

AERO-S Manual

Version 1.1

Edition 0.1/11 April 2014

Farhat Research Group (FRG)
Stanford University

This manual was prepared with Texinfo (<http://www.gnu.org/software/texinfo>).

Short Contents

- [AERO-S](#)
- [1 INTRODUCTION](#)
- [2 INSTALLATION](#)
- [3 ACOUSTIC TIME-DOMAIN ARTIFICIAL BOUNDARY *S*](#)
- [4 ACOUSTIC TIME-DOMAIN DIRICHLET BOUNDARY CONDITIONS *S*](#)
- [5 ACOUSTIC TIME-DOMAIN DISTRIBUTED NEUMANN BOUNDARY CONDITION *S*](#)
- [6 ACOUSTIC TIME-DOMAIN NEUMANN BOUNDARY CONDITIONS *S*](#)
- [7 ACOUSTIC TIME-DOMAIN ROBIN DISTRIBUTED BOUNDARY CONDITION *S*](#)
- [8 ACTUATORS *S*](#)
- [9 AEROELASTICITY](#)
- [10 AEROHEAT \(FEM-HEAT Only\)](#)
- [11 ATTRIBUTES *S*](#)
- [12 BEAM OFFSET](#)
- [13 BEAM REFERENCE FRAMES](#)
- [14 BINARY INPUT / OUTPUT](#)
- [15 BOUNDARY CONVECTION *S*](#)
- [16 BOUNDARY DISPLACEMENTS *S*](#)
- [17 BOUNDARY FLUXES](#)
- [18 BOUNDARY FORCES *S*](#)
- [19 BOUNDARY TEMPERATURES *S*](#)
- [20 BUCKLING](#)
- [21 COMMENTS](#)
- [22 COMPOSITE \(OR ORTHOTROPIC SHELL OR ANISOTROPIC SOLID\)](#)
- [23 COMPOSITE \(ORTHOTROPIC SHELL OR ANISOTROPIC SOLID\) ELEMENT FRAMES](#)
- [24 CONDITION NUMBER](#)
- [25 CONSTRAINTS](#)
- [26 CONTACT SURFACES](#)
- [27 CONTROL STATEMENT](#)
- [28 CONWEP](#)
- [29 DECOMPOSE](#)
- [30 DISCRETE NODAL MASS AND INERTIA](#)
- [31 DYNAMIC ANALYSIS](#)
- [32 EIGENVALUE PROBLEMS](#)
- [33 END](#)
- [34 FLUID/STRUCTURE INTERFACE](#)
- [35 FIELD WEIGHTS FOR MESH DECOMPOSITION](#)
- [36 FORCE TIME TABLE-MECHANICS AND ACOUSTICS](#)
- [37 GEOMETRIC RIGID BODY MODES](#)
- [38 GEOMETRIC STIFFENING DUE TO PRESTRESS](#)
- [39 GRAVITATIONAL ACCELERATION](#)
- [40 GROUPS](#)
- [41 HEAT ZERO ENERGY MODE](#)
- [42 HEAT ZERO ENERGY MODES FILTER](#)
- [43 HYDROELASTIC FLUID/STRUCTURE BOUNDARY](#)
- [44 HYDROELASTIC FREE SURFACE BOUNDARY](#)
- [45 HELMHOLTZ](#)
- [46 HELMHOLTZ ARTIFICIAL BOUNDARY *S*](#)
- [47 HELMHOLTZ DIRICHLET BOUNDARY CONDITIONS *S*](#)
- [48 HELMHOLTZ DISTRIBUTED NEUMANN BOUNDARY CONDITION *S*](#)
- [49 HELMHOLTZ NEUMANN BOUNDARY CONDITIONS *S*](#)
- [50 HELMHOLTZ SCATTERER BOUNDARY *S*](#)
- [51 HELMHOLTZ WET INTERFACE BOUNDARY *S*](#)

- [52 HELMHOLTZ LINEAR MULTIPoint CONSTRAINTS](#)
- [53 HELMHOLTZ LOCATIONS WHERE TO COMPUTE THE KIRCHHOFF INTEGRAL](#)
- [54 IMPEDANCE ANALYSIS](#)
- [55 INITIAL ACCELERATIONS \(Not Supported Yet\)](#)
- [56 INITIAL DISPLACEMENTS *S*](#)
- [57 INITIAL DISPLACEMENT 6 COLUMNS \(IDISP6 completely spelled out\) *S*](#)
- [58 INITIAL OR SEED DISPLACEMENT FOR PITA \(IDISP6PITA completely spelled out\)](#)
- [59 INITIAL OR SEED VELOCITY FOR PITA \(IVEL6PITA completely spelled out\)](#)
- [60 INITIAL TEMPERATURES *S*](#)
- [61 INITIAL VELOCITIES *S*](#)
- [62 INTRUSIVE POLYNOMIAL CHAOS](#)
- [63 LINEAR MATERIAL PROPERTIES *S*](#)
- [64 LINEAR MULTIPoint CONSTRAINTS](#)
- [65 LOAD](#)
- [66 LOADCASE DEFINITION](#)
- [67 LUMPED](#)
- [68 MASS EVALUATION](#)
- [69 MATERIAL LAW](#)
- [70 MATERIAL LAW USAGE](#)
- [71 SAVING EIGENMODES OR PROJECTING ONTO EIGENVECTORS](#)
- [72 NODAL CONTACT](#)
- [73 NODAL FRAMES](#)
- [74 NODES *S*](#)
- [75 NON INTRUSIVE POLYNOMIAL CHAOS](#)
- [76 NON LINEAR ANALYSIS](#)
- [77 OUTPUT OF RESULTS \(OUTPUT completely spelled out\)](#)
- [78 OUTPUT OF RESULTS 6 COLUMNS \(OUTPUT6 completely spelled out\)](#)
- [79 PARALLEL-IN-TIME ALGORITHM \(PITA\)](#)
- [80 PRELOAD](#)
- [81 PRESSURE](#)
- [82 QUASISTATICS ANALYSIS](#)
- [83 RANDOM](#)
- [84 READING EIGEN MODES](#)
- [85 RENUMBERING](#)
- [86 RESTART](#)
- [87 RIGID BODY \(AND OTHER ZERO ENERGY\) MODES](#)
- [88 RIGID BODY MODES FILTER](#)
- [89 CONSTRUCTING A REDUCED-ORDER BASIS](#)
- [90 CONSTRUCTING A REDUCED MESH](#)
- [91 SENSORS *S*](#)
- [92 SLOSHING PROBLEMS](#)
- [93 SLOSHING ZERO ENERGY MODE](#)
- [94 SOURCE TIME TABLE-HEAT CONDUCTION](#)
- [95 STATICS](#)
- [96 STRUCTURAL DAMPING LOSS FACTOR TABLE](#)
- [97 SURFACE TOPOLOGY](#)
- [98 THERMAL EXPANSION TEMPERATURE TABLE](#)
- [99 THERMOELASTICITY](#)
- [100 THERMOELASTICITY: FEM-HEAT \(FEM-HEAT Only\)](#)
- [101 TIED SURFACES](#)
- [102 TOPOLOGY *S*](#)
- [103 USER DEFINED FORCES *S*](#)
- [104 USER DEFINED PRESCRIBED DISPLACEMENTS *S*](#)
- [105 WEIGHTS](#)
- [106 YOUNGS MODULUS TEMPERATURE TABLE](#)

Table of Contents

- [AERO-S](#)
- [1 INTRODUCTION](#)
- [2 INSTALLATION](#)
- [3 ACOUSTIC TIME-DOMAIN ARTIFICIAL BOUNDARY *S*](#)
- [4 ACOUSTIC TIME-DOMAIN DIRICHLET BOUNDARY CONDITIONS *S*](#)
- [5 ACOUSTIC TIME-DOMAIN DISTRIBUTED NEUMANN BOUNDARY CONDITION *S*](#)
- [6 ACOUSTIC TIME-DOMAIN NEUMANN BOUNDARY CONDITIONS *S*](#)
- [7 ACOUSTIC TIME-DOMAIN ROBIN DISTRIBUTED BOUNDARY CONDITION *S*](#)
- [8 ACTUATORS *S*](#)
- [9 AEROELASTICITY](#)
- [10 AEROHEAT \(FEM-HEAT Only\)](#)
- [11 ATTRIBUTES *S*](#)
- [12 BEAM OFFSET](#)
- [13 BEAM REFERENCE FRAMES](#)
- [14 BINARY INPUT / OUTPUT](#)
- [15 BOUNDARY CONVECTION *S*](#)
- [16 BOUNDARY DISPLACEMENTS *S*](#)
- [17 BOUNDARY FLUXES](#)
- [18 BOUNDARY FORCES *S*](#)
- [19 BOUNDARY TEMPERATURES *S*](#)
- [20 BUCKLING](#)
- [21 COMMENTS](#)
- [22 COMPOSITE \(OR ORTHOTROPIC SHELL OR ANISOTROPIC SOLID\)](#)
- [23 COMPOSITE \(ORTHOTROPIC SHELL OR ANISOTROPIC SOLID\) ELEMENT FRAMES](#)
- [24 CONDITION NUMBER](#)
- [25 CONSTRAINTS](#)
- [26 CONTACT SURFACES](#)
- [27 CONTROL STATEMENT](#)
- [28 CONWEP](#)
- [29 DECOMPOSE](#)
- [30 DISCRETE NODAL MASS AND INERTIA](#)
- [31 DYNAMIC ANALYSIS](#)
- [32 EIGENVALUE PROBLEMS](#)
- [33 END](#)
- [34 FLUID/STRUCTURE INTERFACE](#)
- [35 FIELD WEIGHTS FOR MESH DECOMPOSITION](#)
- [36 FORCE TIME TABLE-MECHANICS AND ACOUSTICS](#)
- [37 GEOMETRIC RIGID BODY MODES](#)
- [38 GEOMETRIC STIFFENING DUE TO PRESTRESS](#)
- [39 GRAVITATIONAL ACCELERATION](#)
- [40 GROUPS](#)
- [41 HEAT ZERO ENERGY MODE](#)
- [42 HEAT ZERO ENERGY MODES FILTER](#)
- [43 HYDROELASTIC FLUID/STRUCTURE BOUNDARY](#)
- [44 HYDROELASTIC FREE SURFACE BOUNDARY](#)
- [45 HELMHOLTZ](#)
- [46 HELMHOLTZ ARTIFICIAL BOUNDARY *S*](#)
- [47 HELMHOLTZ DIRICHLET BOUNDARY CONDITIONS *S*](#)
- [48 HELMHOLTZ DISTRIBUTED NEUMANN BOUNDARY CONDITION *S*](#)
- [49 HELMHOLTZ NEUMANN BOUNDARY CONDITIONS *S*](#)
- [50 HELMHOLTZ SCATTERER BOUNDARY *S*](#)
- [51 HELMHOLTZ WET INTERFACE BOUNDARY *S*](#)

- [52 HELMHOLTZ LINEAR MULTIPoint CONSTRAINTS](#)
- [53 HELMHOLTZ LOCATIONS WHERE TO COMPUTE THE KIRCHHOFF INTEGRAL](#)
- [54 IMPEDANCE ANALYSIS](#)
- [55 INITIAL ACCELERATIONS \(Not Supported Yet\)](#)
- [56 INITIAL DISPLACEMENTS *S*](#)
- [57 INITIAL DISPLACEMENT 6 COLUMNS \(IDISP6 completely spelled out\) *S*](#)
- [58 INITIAL OR SEED DISPLACEMENT FOR PITA \(IDISP6PITA completely spelled out\)](#)
- [59 INITIAL OR SEED VELOCITY FOR PITA \(IVEL6PITA completely spelled out\)](#)
- [60 INITIAL TEMPERATURES *S*](#)
- [61 INITIAL VELOCITIES *S*](#)
- [62 INTRUSIVE POLYNOMIAL CHAOS](#)
- [63 LINEAR MATERIAL PROPERTIES *S*](#)
- [64 LINEAR MULTIPoint CONSTRAINTS](#)
- [65 LOAD](#)
- [66 LOADCASE DEFINITION](#)
- [67 LUMPED](#)
- [68 MASS EVALUATION](#)
- [69 MATERIAL LAW](#)
- [70 MATERIAL LAW USAGE](#)
- [71 SAVING EIGENMODES OR PROJECTING ONTO EIGENVECTORS](#)
- [72 NODAL CONTACT](#)
- [73 NODAL FRAMES](#)
- [74 NODES *S*](#)
- [75 NON INTRUSIVE POLYNOMIAL CHAOS](#)
- [76 NON LINEAR ANALYSIS](#)
- [77 OUTPUT OF RESULTS \(OUTPUT completely spelled out\)](#)
- [78 OUTPUT OF RESULTS 6 COLUMNS \(OUTPUT6 completely spelled out\)](#)
- [79 PARALLEL-IN-TIME ALGORITHM \(PITA\)](#)
- [80 PRELOAD](#)
- [81 PRESSURE](#)
- [82 QUASISTATICS ANALYSIS](#)
- [83 RANDOM](#)
- [84 READING EIGEN MODES](#)
- [85 RENUMBERING](#)
- [86 RESTART](#)
- [87 RIGID BODY \(AND OTHER ZERO ENERGY\) MODES](#)
- [88 RIGID BODY MODES FILTER](#)
- [89 CONSTRUCTING A REDUCED-ORDER BASIS](#)
- [90 CONSTRUCTING A REDUCED MESH](#)
- [91 SENSORS *S*](#)
- [92 SLOSHING PROBLEMS](#)
- [93 SLOSHING ZERO ENERGY MODE](#)
- [94 SOURCE TIME TABLE-HEAT CONDUCTION](#)
- [95 STATICS](#)
- [96 STRUCTURAL DAMPING LOSS FACTOR TABLE](#)
- [97 SURFACE TOPOLOGY](#)
- [98 THERMAL EXPANSION TEMPERATURE TABLE](#)
- [99 THERMOELASTICITY](#)
- [100 THERMOELASTICITY: FEM-HEAT \(FEM-HEAT Only\)](#)
- [101 TIED SURFACES](#)
- [102 TOPOLOGY *S*](#)
- [103 USER DEFINED FORCES *S*](#)
- [104 USER DEFINED PRESCRIBED DISPLACEMENTS *S*](#)
- [105 WEIGHTS](#)
- [106 YOUNGS MODULUS TEMPERATURE TABLE](#)

Up: [\(dir\)](#)

AERO-S

- [ATDARB](#)
- [ATDDIR](#)
- [ATDDNB](#)
- [ATDNEU](#)
- [ATDROB](#)
- [ACTUATORS](#)
- [AERO](#)
- [AEROH](#)
- [ATTRIBUTES](#)
- [BOFFSET](#)
- [EFRAMES](#)
- [BINARY](#)
- [CONVECTION](#)
- [DISPLACEMENTS](#)
- [FLUX](#)
- [FORCES](#)
- [TEMPERATURES](#)
- [BUCKLE](#)
- [COMPOSITE](#)
- [CFRAMES](#)
- [CONDITION](#)
- [CONSTRAINTS](#)
- [CONTACTSURFACES](#)
- [CONTROL](#)
- [CONWEP](#)
- [DECOMPOSE](#)
- [DIMASS](#)
- [DYNAMICS](#)
- [EIGEN](#)
- [FSINTERFACE](#)
- [FWEIGHTS](#)
- [MFTT](#)
- [GRBM](#)
- [GEPS](#)
- [GRAVITY](#)
- [GROUPS](#)
- [HZEM](#)
- [HZEMFILTER](#)
- [HEFSB](#)
- [HEFRS](#)
- [HELMHOLTZ](#)
- [HARB](#)
- [HDIR](#)
- [HDNB](#)
- [HNEU](#)
- [HSCB](#)
- [HWIB](#)
- [HLMPC](#)
- [KIRLOC](#)
- [IMPEDANCE](#)

- [IACCELERATIONS](#)
- [IDISPLACEMENTS](#)
- [IDISP6](#)
- [IDISP6PITA](#)
- [IVEL6PITA](#)
- [ITEMPERATURES](#)
- [IVELOCITIES](#)
- [INPC](#)
- [MATERIAL](#)
- [LMPC](#)
- [LOAD](#)
- [LOADCASE](#)
- [LUMPED](#)
- [MASS](#)
- [MATLAW](#)
- [MATUSAGE](#)
- [MODE](#)
- [NODALCONTACT](#)
- [NFRAMES](#)
- [NODES](#)
- [NONINPC](#)
- [NONLINEAR](#)
- [OUTPUT](#)
- [OUTPUT6](#)
- [PITA](#)
- [PRELOAD](#)
- [PRESSURE](#)
- [QSTATICs](#)
- [RANDOM](#)
- [READMODE](#)
- [RENUMBERING](#)
- [RESTART](#)
- [TRBM](#)
- [RBMFILTER](#)
- [ROBC](#)
- [RMSHC](#)
- [SENSORS](#)
- [SLOSH](#)
- [SZEM](#)
- [HFTT](#)
- [STATICS](#)
- [SDETAFT](#)
- [SURFACETOPO](#)
- [TETT](#)
- [THERMOE](#)
- [THERMOH](#)
- [TIEDSURFACES](#)
- [TOPOLOGY](#)
- [USDF](#)
- [USDD](#)
- [WEIGHTS](#)
- [YMTT](#)

Next: [INSTALLATION](#)

1 INTRODUCTION

This section compiles basic information on how to prepare an input data for a finite element analysis using the **AERO-S** code, and how to run this code.

The input data consists of command macros, clustered numerical data, and comment lines. Each data cluster is preceded by a command statement. The following rules of thumb are suggested:

- 1 The users are allowed to have as many comment lines as they wish, as long as the first column of each line starts with * (see [COMMENTS](#)).
- 2 The subsequent data clusters may appear in any order. The command statement preceding each cluster may be in lower, upper, or mixed characters.
- 3 The data in each cluster are free format.
- 4 **Only the first four letters of a command need to be specified unless otherwise noted.**

Any segment of a **AERO-S** input file — for example, the data corresponding to any command macro, or a command macro and its data, or any number of input lines — can be replaced by a statement of the form

INCLUDE	filename (or "filename")
----------------	---------------------------------

where `filename` is the name of the file containing the information. "`filename`" can contain a path or can be replaced by `<filename>` in which case the path is that of the environment variable `\$FEM_INCLUDE`. Furthermore, the included file `filename` can be compressed by gzip, bzip2, or zip.

AERO-S can be executed in serial or parallel mode. However, if the requested finite element analysis requires an equation solver, **AERO-S** runs in parallel mode on a given parallel architecture only if the chosen equation solver (see [STATICS](#)) can be executed in parallel mode on that parallel architecture. In the latter case, performing the core finite element analysis — in addition to the solution of the system(s) of equations — in parallel using MPI and/or OpenMP requires partitioning the finite element mesh using the [DECOMPOSE](#) command. If on the other hand the requested finite element analysis does not require an equation solver, **AERO-S** can be executed in parallel mode on either a shared or distributed memory system as long as the finite element mesh is partitioned using the [DECOMPOSE](#) command.

The command line for executing **AERO-S** in serial mode is

aeros

<code>[-d <decomposition_pathandfilename>]</code>	(Specifies a decomposition file).	<code>[-v <verbose_frequency>]</code>	(Turns on and specifies frequency of printing on screen the FETI iteration count and subspace)
---	-----------------------------------	---	--

		iteration count).
[-c]	(Outputs contact status on screen (FETI solver)).	
[-t]	(Converts input file to XPost format).	
[-T]	(Converts input file to XPost format after removing all numbering gaps).	
[-m]	(Converts input file to XPost format after gathering each material in a separate element set).	
[-M]	(Converts input file to XPost format after removing all numbering gaps and gathering each material in a separate element set).	
[-p]	(Generates automatically XPost patterns for the various XPost element sets. This option is useful only in conjunction with the -m and -M options which can generate multiple XPost element sets. Also, automatically generates a global element set).	
<filename.aeros.aicdf>	(AERO-S ASCII Input Command Data file containing the finite element structural model and analysis commands).	

The command line for executing **AERO-S** in parallel mode on a shared memory system using OpenMP and `number_of_threads` threads is

aeros -n `number_of_threads`

[-d <decomposition_pathandfilename>]	(Specifies a decomposition file).
[-v <verbose_frequency>]	(Turns on verbose and specifies frequency of printing on screen the FETI iteration count and subspace iteration count).
[-c]	(Outputs contact status on screen (FETI solver)).
[-t]	(Converts input file to XPost format).
[-T]	(Converts input file to XPost after removing all numbering gaps).

[-m]	(Converts input file to XPost format after gathering each material in a separate element set).
[-M]	(Converts input file to XPost format after removing all numbering gaps and gathering each material in a separate element set).
[-P]	(Generates automatically XPost patterns for the various XPost element sets. This option is useful only in conjunction with the -m and -M options which can generate multiple XPost element sets. Also, automatically generates a global element set).
<filename.aeros.aicdf>	(AERO-S ASCII Input Command Data file containing the finite element structural model and analysis commands).

The command line for executing **AERO-S** in parallel mode on a distributed system using MPI with `number_of_MPI_processes` MPI processes [and `number_of_threads_within_an_MPI_process` threads within an MPI process] is

mpirun -np <code>number_of_MPI_processes</code> aeros -n <code>number_of_threads_within_an_MPI_process</code>	
[-d <decomposition_pathandfilename>]	(Specifies a decomposition file).
[-v <verbose_frequency>]	(Turns on verbose and specifies frequency of printing on screen the FETI iteration count and subspace iteration count).
[-c]	(Outputs contact status on screen (FETI solver)).
[-t]	(Converts input file to XPost format).
[-T]	(Converts input file to XPost after removing all numbering gaps).
[-m]	(Converts input file to XPost format after gathering each material in a separate element set).
[-M]	(Converts input file to XPost format after removing all numbering gaps and gathering each material in a separate element set).
[-P]	(Generates automatically XPost patterns for the various XPost element sets. This option is useful only in conjunction with the -m and -M options which can generate multiple XPost element sets. Also, automatically generates a global element set).
<filename.aeros.aicdf>	(AERO-S ASCII Input Command Data file containing the finite element structural model and analysis commands).

Next: [ATDARB](#), Previous: [INTRODUCTION](#)

2 INSTALLATION

The installation of **AERO-S** on a given computing system requires the availability on that system of the following tools:

C++ compiler g++	Version 4.1.2 or higher.
Fortran compiler gfortran	Version 4.1.2 or higher.
Flex utility	Version 2.5 or higher. <code>Flex</code> is a lexical analyser required for building the parser of AERO-S 's input command data file.
Bison utility	Version 2.3 or higher. <code>Bison</code> is a parser generator required for building the parser of AERO-S 's input command data file.
CMake utility	Version 2.6 or higher. <code>CMake</code> is a cross-platform open-source build system. It is comparable to the Unix <code>Make</code> program in that the build process is ultimately controlled by configuration files (<code>CMakeLists.txt</code>). However unlike <code>Make</code> , it does not directly build the final software but instead generates standard build files such as makefiles for Unix and projects/workspaces for Windows Visual C++. The <code>cmake</code> version 2.6 utility can be obtained from http://www.cmake.org . (Note: a "README.cmake" file discussing details on <code>cmake</code> options for code configuration and installation is available in the directory containing the source code of AERO-S).

and following libraries:

BLAS library	BLAS is a set of Basic Linear Algebra Subprograms required by various operations performed in AERO-S .
LAPACK library	LAPACK is a high-performance Linear Algebra PACKage with advanced solvers.
MPI library openmpi	Version 1.2.6 or higher. <code>Open MPI</code> is a high-performance implementation of the Message Passing Interface (MPI) required for performing interprocessor communication, among others. More specifically, AERO-S requires an MPI-2 implementation such as the one provided by the Open MPI project.
OpenMP API	Open Multi-Processing is an Application Programming Interface (API) that supports multi-platform shared memory multiprocessing programming in C, C++ and Fortran on many architectures, including Unix. As an option, AERO-F can be compiled with <code>OpenMP</code> to enable multi-threaded execution.

In addition, the following optional libraries extend the capabilities of **AERO-S**:

SPOOLES library	SPOOLES is a library for solving sparse real and complex linear systems of equations with a sparse direct solver, written in the C language using object oriented design.
MUMPS library	MUMPS is a library for solving sparse real and complex linear systems of equations with a multifrontal massively parallel sparse direct solver.
SUPERLU library	SUPERLU is a general purpose library for the direct solution of large sparse nonsymmetric systems on high performance machines.
ARPACK library	ARPACK is the Arnoldi PACKage for the solution of large-scale symmetric, nonsymmetric, and generalized eigenproblems.
ScaLAPACK library	ScaLAPACK is also known as the Scalable LAPACK. This library includes a subset of LAPACK routines redesigned for distributed memory MIMD parallel computers.
BLACS library	BLACS (Basic Linear Algebra Communication Subprograms) is a linear algebra oriented message passing interface designed for linear algebra.
PARPACK library	PARPACK is the parallel version of ARPACK used by AERO-S 's parallel eigensolver.
METIS library	METIS is a library of graph manipulation routines that can be used by AERO-S for reordering of a sparse matrix to reduce the number of fill-in entries created

during factorization.

Zoltan library

The Zoltan library includes among other things a suite of dynamic load-balancing and parallel partitioning tools that are used by **AERO-S** for parallel proximity searches.

Eigen3 library

Eigen3 is a versatile C++ template library for linear algebra (vectors, matrices, and related algorithms (see <http://eigen.tuxfamily.org>)).

To install **AERO-S**, follow the procedure specified below:

- From the directory containing the source code of **AERO-S**, type `cmake -DAERO=1 .` Note the space and the `.` after the command `cmake`. The `.` specifies the current directory.
- Watch the computer screen and verify that all invoked components were found and all build options were correct. A sample computer screen output of the `cmake` command is:

```
-- The C compiler identification is GNU
-- The CXX compiler identification is GNU
-- Check for working C compiler: /usr/bin/gcc
-- Check for working C compiler: /usr/bin/gcc -- works
-- Detecting C compiler ABI info
-- Detecting C compiler ABI info - done
-- Check for working CXX compiler: /usr/bin/c++
-- Check for working CXX compiler: /usr/bin/c++ -- works
-- Detecting CXX compiler ABI info
-- Detecting CXX compiler ABI info - done
-- The Fortran compiler identification is GNU
-- Check for working Fortran compiler: /usr/bin/gfortran
-- Check for working Fortran compiler: /usr/bin/gfortran -- works
-- Detecting Fortran compiler ABI info
-- Detecting Fortran compiler ABI info - done
-- Checking whether /usr/bin/gfortran supports Fortran 90
-- Checking whether /usr/bin/gfortran supports Fortran 90 -- yes
-- Found MPI: /usr/lib/openmpi/lib/libmpi_cxx.so
-- Try OpenMP C flag = [-fopenmp]
-- Performing Test OpenMP_FLAG_DETECTED
-- Performing Test OpenMP_FLAG_DETECTED - Success
-- Try OpenMP CXX flag = [-fopenmp]
-- Performing Test OpenMP_FLAG_DETECTED
-- Performing Test OpenMP_FLAG_DETECTED - Success
-- Found OpenMP: -fopenmp
-- Looking for Fortran sgemm
-- Looking for Fortran sgemm - found
-- A library with BLAS API found.
-- Looking for Fortran cheev
-- Looking for Fortran cheev - found
-- A library with LAPACK API found.
-- A library with ARPACK API found.
-- A library with SPOOLES API found.
-- A library with EIGEN2 API found.
-- A library with METIS API found.
-- A library with MUMPS API found.
-- A library with BLACS API found.
-- A library with SCALAPACK API found.
-- A library with METIS API found.
```

```
=====
Summary of build options
-----
Distributed FETI:      YES
Aeroelastic:           YES
Mumps:                 YES
Arpack:                YES
Spuoles:               YES
Acme:                  YES
Metis:                 YES
OpenMP:                YES
```

```

Build type:           Release
Extra libraries:
=====
-- Configuring done
-- Generating done
-- Build files have been written to: /home/pavery/Codes/FEM

```

- If necessary, edit the `CMakeCache.txt` file to include the file paths to all required and desired optional components that were not automatically found by `cmake`. Typically, the compilers, the utilities `Flex` and `Bison`, the libraries `MPI`, `BLAS` and `LAPACK`, and the API `OpenMP` will be automatically found. However, it may be necessary to specify the paths for the libraries `SPOOLES`, `MUMPS`, `ScalAPACK`, `BLACS`, `ARPACK`, `PARPACK`, `METIS`, `Zoltan` and `Eigen3`.
- Then, also from the directory containing the source code of **AERO-S**, type `make`.

The successful completion of the procedure described above leads to the creation in the bin/ directory of **AERO-S**'s executable **aeros**.

Next: [ATDDIR](#), Previous: [INSTALLATION](#)

3 ACOUSTIC TIME-DOMAIN ARTIFICIAL BOUNDARY *S*

Command Statement:	ATDARB
--------------------	---------------

The ATDARB command statement is used to specify the artificial boundary Σ on which an Antoine absorbing condition is to be applied in the time domain, and the order of this absorbing boundary condition. The input format is given below.

ATDARB	ORDER
--------	-------

FACE	FACE_TYPE	CONNECTIVITY_NODES
------	-----------	--------------------

ORDER	Order of Antoine's absorbing boundary condition in the time domain (real). Currently supported values are 0 and 1.	
FACE	Face (or edge in two dimensions) identification number whose type and connectivity are to be specified (integer). In practice, this identification number is ignored by AERO-S .	
FACE_TYPE		
1	2-node line segment. To be used with two-dimensional linear elements.	
2	3-node line segment. To be used with two-dimensional quadratic elements.	
3	3-node triangular face. To be used with three-dimensional linear tetrahedral element.	
4	4-node quad face. To be used with three-dimensional linear hexahedral element.	
6	6-node triangular face. To be used with three-dimensional quadratic tetrahedral element.	

- 10 n^2 -node arbitrarily higher-order quad face where n is the number of nodes on an edge of this face. To be used with element type 95.
- 11 arbitrarily higher-order triangular face.
- 12 arbitrarily higher-order line segment.
- 13 edge of a full isoparametric triangular element where the nodes are numbered linearly along it.
- 14 n^2 -node arbitrarily higher-order (spectral) quad face where n is the number of nodes on an edge of this face. To be used with spectral element type 105.

CONNECTIVITY_NODES These should be listed in a stacked fashion on a single line, and numbered clockwise (when looking from infinity in three dimensions).

Next: [ATDDNB](#), Previous: [ATDARB](#)

4 ACOUSTIC TIME-DOMAIN DIRICHLET BOUNDARY CONDITIONS *S*

Command Statement:	ATDDIR
--------------------	---------------

The ATDDIR command statement is used to specify *nodal* Dirichlet boundary conditions for a time-domain acoustic scattering problem. The input format is given below.

ATDDIR

NODE#	VALUE
--------------	--------------

NODE#	Node number where the Dirichlet boundary condition is specified (integer).
VALUE	Value of the specified boundary condition (real).

Next: [ATDNEU](#), Previous: [ATDDIR](#)

5 ACOUSTIC TIME-DOMAIN DISTRIBUTED NEUMANN BOUNDARY CONDITION *S*

Command Statement:	ATDDNB
--------------------	---------------

The ATDDNB command statement is used to specify the surface of a scatterer on which a *distributed*

Neumann boundary condition of the form $\frac{\partial p^s}{\partial n} = c$ is applied in a time-domain acoustic computation,

and the value of the constant c .

The input format of this command is given below.

ATDDNB	CONSTANT
--------	----------

FACE	FACE_TYPE	CONNECTIVITY_NODES
------	-----------	--------------------

CONSTANT	Value of the constant c (real).	
FACE	Face (or edge in two dimensions) identification number whose type and connectivity are to be specified (integer). In practice, this identification number is ignored by AERO-S .	
FACE_TYPE		
1	2-node line segment. To be used with two-dimensional linear elements.	
2	3-node line segment. To be used with two-dimensional quadratic elements.	
3	3-node triangular face. To be used with three-dimensional linear tetrahedral element.	
4	4-node quad face. To be used with three-dimensional linear hexahedral element.	
6	6-node triangular face. To be used with three-dimensional quadratic tetrahedral element.	
10	n^2 -node arbitrarily higher-order quad face where n is the number of nodes on an edge of this face. To be used with element type 95.^M	
11	arbitrarily higher-order triangular face.	
12	arbitrarily higher-order line segment.	
13	edge of a full isoparametric triangular element where the nodes are numbered linearly along it.	
14	n^2 -node arbitrarily higher-order (spectral) quad face where n is the number of nodes on an edge of this face. To be used with spectral element type 105.	
CONNECTIVITY_NODES	These should be listed in a stacked fashion on a single line, and numbered clockwise (when looking from infinity in three dimensions).	

Next: [ATDROB](#), Previous: [ATDDNB](#)

6 ACOUSTIC TIME-DOMAIN NEUMANN BOUNDARY

CONDITIONS *S*

Command Statement:	ATDNEU
--------------------	---------------

The ATDNEU command statement is used to specify the *nodal* Neumann boundary conditions for a time-domain acoustic scattering problem. The input format is given below.

ATDNEU

NODE#	VALUE
-------	-------

NODE# Node number where the Neumann boundary condition is specified (integer).

VALUE Value of the specified boundary condition (real).

Next: [ACTUATORS](#), Previous: [ATDNEU](#)

7 ACOUSTIC TIME-DOMAIN ROBIN DISTRIBUTED BOUNDARY CONDITION *S*

Command Statement:	ATDROB
--------------------	---------------

The ATDROB command statement can be used to specify the surface of a scatterer on which a *distributed* Robin boundary condition of the form

$$\alpha \frac{\partial p^s}{\partial n} + \beta p^s = \gamma$$

is applied in a time-domain acoustic computation, and the three constants α , β , and γ .

The input format of this command is given below.

ATDROB	α	β	γ
--------	----------	---------	----------

FACE	FACE_TYPE	CONNECTIVITY_NODES
------	-----------	--------------------

α Non-zero constant premultiplying the normal derivative of the Robin boundary condition (real).

β Constant premultiplying the unknown of the Robin boundary condition (float).



Constant right hand-side of the Robin boundary condition (float).

FACE	Face (or edge in two dimensions) identification number whose type and connectivity are to be specified (integer). In practice, this identification number is ignored by AERO-S .
FACE_TYPE	
1	2-node line segment. To be used with two-dimensional linear elements.
2	3-node line segment. To be used with two-dimensional quadratic elements.
3	3-node triangular face. To be used with three-dimensional linear tetrahedral element.
4	4-node quad face. To be used with three-dimensional linear hexahedral element.
6	6-node triangular face. To be used with three-dimensional quadratic tetrahedral element.
10	n^2 -node arbitrarily higher-order quad face where n is the number of nodes on an edge of this face. To be used with element type 95.^M
11	arbitrarily higher-order triangular face.
12	arbitrarily higher-order line segment.
13	edge of a full isoparametric triangular element where the nodes are numbered linearly along it.
14	n^2 -node arbitrarily higher-order (spectral) quad face where n is the number of nodes on an edge of this face. To be used with spectral element type 105.

CONNECTIVITY_NODES These should be listed in a stacked fashion on a single line, and numbered clockwise (when looking from infinity in three dimensions).

Next: [AERO](#), Previous: [ATDROB](#)

8 ACTUATORS *S*

Command Statement:	ACTUATORS
--------------------	------------------

The ACTUATORS command is used to prescribe at specified nodes and local degrees of freedom a time-variant source term (for example, a force field for a structural model) using a user-defined subroutine, and request the solution state at that node (for example, the displacement, velocity, and acceleration fields for a structural model). In this case, the user has to write his/her own algorithm for specifying the prescribed source term within a subroutine named "control.C". The [SENSORS](#) command must be used to pass to "control.C" the solution state at the nodes of interest (see [SENSORS](#)). The user should grab the special makefile for this command which is located within the AERO-S.d/Control.d directory, and use the [LOAD](#) command to activate this command.

For structural models, all prescribed forces and moments are interpreted by default as being of the *axial* type — that is, as being defined in the *fixed* nodal degree of freedom reference frames (see [NODES](#) and [NFRAMES](#)). However, if a node has rotational degrees of freedom, the user can specify

that the forces and/or moments prescribed at this node are of the *follower* type — that is, they act in a direction that remains constant in the local frame attached to the node where they are applied. This local frame coincides with the nodal degree of freedom reference frame (see [NODES](#) and [NFRAMES](#)) in the undeformed configuration. In the deformed configuration, the orientation of this local frame is defined by the rotation of the node to which it is attached. In other words, the specified nodal force or moment "follows" in this case the rotation of the node to which it is applied.

An example input file using this command can be found in [APPENDIX 1](#).

Note 1: By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: For nonlinear structural models, sensor information on the velocity and acceleration of a nodal degree of freedom is not currently available for use in the "control.C" file. These are currently passed as zero.

Note 3: For structural models, specifying a follower force or moment leads to an unsymmetric tangent "load" stiffness matrix during a [NONLINEAR](#) analysis.

The syntax for invoking this option is given below.

ACTUATORS

NODE#	DOF#	TYPE
-------	------	------

NODE# Node number where an actuating force/moment is specified (integer).
 DOF# Degree of freedom local number where an actuating force/moment is specified (integer).
 TYPE For structural models, all specified nodal source terms are by default of the axial type. However, if this parameter is set to FOLLOWER and the node NODE# has rotational degrees of freedom, the source term specified at this node and degree of freedom DOF# is considered to be of the follower type (characters).

Next: [AEROH](#), Previous: [ACTUATORS](#)

9 AEROELASTICITY

Command Statement:	AERO
--------------------	-------------

The AERO command statement can be used to perform any or all of the following instructions:

- indicate that **AERO-S** is to interact with **AERO-F** to compute a flow-induced load and perform the corresponding static structural analysis, or perform a coupled static or dynamic aeroelastic

simulation

- choose a staggered time-integration algorithm (**ALGORITHM** subcommand) and, if needed, a displacement predictor (**ALGORITHM** subcommand) defined by two coefficients α_0 and α_1 and the formula

$$u^{n+k^p} = u^n + \alpha_0 \Delta t_S \dot{u}^n + \alpha_1 \Delta t_S (\dot{u}^n - \dot{u}^{n-1})$$

when the structural time-integrator is implicit, with $k = 1/2$ if **A6** is the chosen time-integration algorithm and $k = 1$ otherwise, or

$$u^{n+\frac{1}{2}^p} = u^n + \alpha_0 \Delta t_S \dot{u}^{n-\frac{1}{2}} + \alpha_1 \Delta t_S (\dot{u}^{n-\frac{1}{2}} - u^{n-\frac{3}{2}})$$

when the structural time-integrator is explicit, to perform a static or dynamic aeroelastic simulation.

Here, u^{n+1^p} ($u^{n+\frac{1}{2}^p}$) is the predicted displacement field at time-step t^{n+1} ($t^{n+\frac{1}{2}}$).

- select an algorithm for computing the corrected pressure field P_c^{n+1} to be used in computing the aerodynamic forces acting on the structure at time-step t^{n+1} (**PRESSURE** subcommand) of a static or dynamic aeroelastic simulation

The purpose of the displacement predictor and that of the force corrector are to compensate for the effects of time-lagging. More specifically, they improve the time-accuracy and numerical stability of the chosen staggered procedure by minimizing the lack of energy conservation at the fluid-structure interface. The theoretical, algorithmic, and practical aspects of both of these "compensators" are described in S. Piperno and C. Farhat, "Partitioned Procedures for the Transient Solution of Coupled Aeroelastic Problems - Part II: Energy Transfer Analysis and Three-Dimensional Applications," Computer Methods in Applied Mechanics and Engineering, Vol. 190, pp. 3147-3170 (2001).

Before setting the values of α_0 and α_1 that complete the definition of the displacement predictor and choosing the force corrector, the user should note that:

- Currently, the fluid code **AERO-F** can send to **AERO-S** at t^{n+1} either the pressure field at t^{n+1} , P^{n+1} , or the averaged pressure field $\bar{P}^{n+1} = \frac{1}{\Delta t_S} \int_{t^n}^{t^{n+1}} P(t) dt$. Hence, P_c^{n+1} can be set at least to either of these two values. Therefore, the purpose of the **PRESSURE** subcommand is to propose additional choices for the corrected pressure field P_c^{n+1} . These can affect the order of

time-accuracy of the chosen staggered solution procedure (see C. Farhat, G. van der Zee and P. Geuzaine, "Provably Second-Order Time-Accurate Loosely-Coupled Solution Algorithms for Transient Nonlinear Computational Aeroelasticity," Computer Methods in Applied Mechanics and Engineering, Vol. 195, pp. 1973-2001 (2006)).

- The values of α_0 and α_1 that minimize the variation of energy exchanged at the fluid-structure interface and therefore improve time-accuracy and numerical stability of the chosen staggered solution procedure depend on the chosen fluid and structure time-integrators, the chosen force corrector, and the selected staggered solution procedure itself.
- When the Generalized α method is used for time-integrating linear or nonlinear structural

dynamics problems (see [DYNAMICS](#)), **AERO-S** solves the following equilibrium problem

$$M\ddot{u}^{n+1-\alpha_m} + C\dot{u}^{n+1-\alpha_f} + F_{internal}(u^{n+1-\alpha_f}) = F_{external}^{n+1-\alpha_f}$$

- When the central difference method is used for time-integrating structural dynamics problems (see [DYNAMICS](#)), **AERO-S** solves the following equilibrium problem

$$M\ddot{u}^{n+1} + C\dot{u}^{n+1} + F_{internal}(u^{n+1}) = F_{external}^{n+1}$$

Note 1: **AERO-F** offers two different computational frameworks for fluid-structure interaction: an Arbitrary Lagrangian/Eulerian (ALE) framework, and an embedded one. When a functional capability described below is meaningful, applicable, or supported for only one of these two computational frameworks, its designating keyword is followed by [ALE] in the case of the ALE computational framework, and by [EMB] in the case of the embedded computational framework. When this capability is applicable, meaningful, and supported for both computational frameworks, its designating keyword is not followed by any symbol.

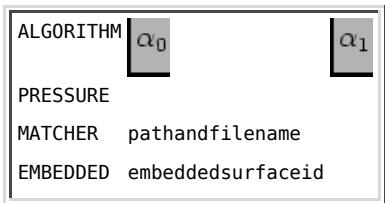
Note 2: The PP, MPP, and c0 algorithms require the additional presence of the [DYNAMICS](#) command in the input file. The MPP algorithm also requires the presence of the MODAL sub-command under the [DYNAMICS](#) command. The A0, A4, A5, and A6 algorithms require the additional presence in the input file of the [QSTATIC](#)S command for a static aeroelastic analysis, and that of the [DYNAMICS](#) command for a dynamic aeroelastic analysis. The B0 algorithm requires the additional presence in the input file of the [QSTATIC](#)S command for a static aeroelastic analysis.

Note 3: The values of α_0 and α_1 derived in the paper by Piperno and Farhat mentioned above correspond to the case where the structure is linear and time-integrated by the Newmark algorithm with $\beta = \frac{1}{4}$ and $\gamma = \frac{1}{2}$ (see [DYNAMICS](#)). For different configurations of the structural time-integrator, that analysis needs to be redone if it is to be used for choosing the parameters of the AERO command.

Note 3: If an aeroelastic analysis is requested with the GEPS and IDISP6 commands present in the input file, and the [IDISPLACEMENTS](#) command is specified to initialize the displacement field, then **AERO-S** sends to the fluid code at each time step the sum of the updated displacement field (interpreted in that case as an increment displacement) and the displacement specified under the IDISP6 command. If in such a case the [IDISPLACEMENTS](#) command is not present in the input file, the displacement field of the structure is then initialized by the IDISP6 command, and **AERO-S** acts as follows: at the first time-step, it sends to the fluid code only the updated displacement field (since initialization accounts in that case for the content of the IDISP6 command), but at each subsequent time-step, it sends the sum of the updated displacement field and the displacement specified under the IDISP6 command.

The syntax for invoking this option is given below.

AERO



ALGORITHM

- [ALE]** **PP** This “Ping-Pong” algorithm sends the initial displacement of the structure, specified under either the `IDISPL6` command or the [IDISPLACEMENTS](#) with the `MODAL` option command (see [IDISPLACEMENTS](#)), to the fluid code which receives it and deforms the fluid mesh accordingly. For this purpose, the `DYNAMICS` command must also be present in the **AERO-S** input file. After the send occurs, the structure code exits gracefully, the fluid code computes the fluid mesh deformation associated with the specified structural displacement and outputs (if requested) the corresponding position and/or displacement of the fluid mesh (characters).
- [ALE]** **MPP** This “Multi-Ping-Pong” algorithm sends several initial displacements of the structure at a time — for example, modal displacements — to the fluid code, which uses them to compute compatible fluid mesh deformations. If a coefficient α_0 is specified after `MPP`, the initial displacements are amplified by α_0 before they are sent to the fluid code. However, the fluid code scales back by $1/\alpha_0$ the corresponding fluid mesh displacements it computes before saving them in an output file, in order to preserve the effect of the mass normalization of the modal structural displacements. The deformed fluid mesh configurations can then be used to generate sources of excitations for the construction of fluid POD bases and ROMs, or for linearized (perturbation) flow simulations whose initial conditions involve a (modal) position or velocity of the fluid surface mesh. The [READMODE](#) command must be used to input the initial displacements of interest — for example, a set of eigenmodes. As for the “Ping-Pong” case, the `DYNAMICS` command must also be present in the **AERO-S** input file, **AERO-S** exits gracefully after the send occurs, the fluid code computes the fluid mesh deformation associated with the specified structural displacements and outputs (if requested) the corresponding positions and/or displacements of the fluid mesh. In addition, for each inputted deformed fluid mesh position, the fluid code outputs an identification tag — for example, the frequency of the corresponding input structural mode in the case of modal displacements (characters).
- A0** This is the basic sequential staggered solution procedure (characters). When subcycling is effected, the same algorithm has been referred to in the AIAA Paper 96-1388 by Farhat and co-workers as the α_1 algorithm (characters). It should not be used without a displacement predictor as it would reduce the overall order of time-accuracy compared to that intrinsic to the structural time-integrator. For the same reason, it is also not recommended — and as a matter of fact not available — when the structural time-integrator is explicit.
- A4** This is the basic staggered solution procedure with fluid-structure inter-parallelism (characters). It has been referred to as α_2 in the AIAA Paper 96-1388 by Farhat and co-workers. When this algorithm is specified, **AERO-F** sends to **AERO-S** P^n or \bar{P}^n rather than the corresponding pressure fields at t^{n+1} . Currently, this algorithm is not available when an explicit structural time-integrator is chosen.

- A5 This staggered solution procedure also features inter-parallelism but offers a better accuracy than A4 when both methods are used without a displacement predictor (characters). Currently, this algorithm is not available when an explicit structural time-integrator is chosen.
- A6 This staggered solution procedure has superior stability and accuracy properties. It is recommended when the fluid is time-advanced by an implicit time-integrator. It has been referred to in most papers by Farhat and co-workers as the ISS (Improved Serial Staggered algorithm) method (characters). It delivers good accuracy without any displacement predictor. However, it is provably second-order time-accurate when equipped with the the displacement predictor defined by $\alpha_0 = 1/2$ and $\alpha_1 = 1/8$, the midpoint implementation of the Newmark time-integrator with $\beta = \frac{1}{4}$ and $\gamma = \frac{1}{2}$ (see [DYNAMICS](#)), and a second-order time-integrator for the discrete fluid problem. Also, this partitioned solution procedure prefers a non time-averaged pressure field. Most importantly, it was designed to be used exclusively with the `NONCOLLOCATED` scheme of aerodynamic force evaluation (see below) (characters). Currently, it does not support an explicit structural time-integrator.
- B0 This algorithm sends the initial displacement of the structure (if any) to the flow solver. After receiving it and updating its mesh, **AERO-F** performs any requested flow simulation and sends to **AERO-S** upon completion the flow-induced load. Then, **AERO-S** receives this load, performs a static analysis if requested, and finally outputs any requested data (such as the computed flow-induced load, and/or resulting displacement, stress, and strain fields).
- C0 This staggered solution procedure is designed for the case where the structural subsystem is time-integrated by the explicit central difference method (see [DYNAMICS](#)), and the fluid subsystem is time-integrated by either an explicit or an implicit scheme. As shown in C. Farhat, A. Rallu, K. Wang and T. Belytschko, "Robust and Provably Second-Order Explicit-Explicit and Implicit-Explicit Staggered Time-Integrators for Highly Nonlinear Fluid-Structure Interaction Problems," International Journal for Numerical Methods in Engineering, (2010), it is genuinely second-order time-accurate when equipped with the second-order displacement predictor obtained by setting $\alpha_0 = 0.5$ and $\alpha_1 = 0.375$ (see below), the second-order force corrector obtained by setting `PRESSURE` to `COLLOCATED` (see below), and a second (or higher)-order accurate explicit or implicit ALE fluid time-integrator. Currently, this algorithm supports only explicit structural time-integrators.
- α_0 When used with the `MPP` command, this coefficient can be specified to amplify the initial displacements to be sent to the fluid code (however, as noted above, the fluid mesh displacements computed by the fluid code are scaled back by $1/\alpha_0$ before they are saved in an output file). In this case, the default value is 1. Otherwise, this coefficient is part of the construction of a predictor for the position of the structure. For $\alpha_0 = 0$ (and $\alpha_1 = 0$), no prediction is effected. Setting $\alpha_0 = 1$ (and $\alpha_1 = 0$) generates a first-order prediction, and setting $\alpha_0 = 1$ (and $\alpha_1 = \frac{1}{2}$) generates a second-order prediction. The optimal values of α_0 and α_1 depend on the specifics of the fluid and structure field time-integrators (see the aforementioned paper on interface energy conservation by Piperno and Farhat) (real). In this case, the default value is 0.5 for algorithm C0, and 0 for all other algorithms.

α_1 Unlike α_0 , this parameter cannot be used with the MPP command (real). It is part of the construction of a predictor for the position of the structure and therefore should be used only as such (see above). Its default value is 0.375 for algorithm c0, and 0 for all other algorithms.

PRESSURE

COLLOCATED

In this case, P_c^{n+1} is set to the received pressure field P^{n+1} (or \bar{P}^{n+1}), then converted to $P_c^{n+1-\alpha_f} = (1 - \alpha_f)P_c^{n+1} + \alpha_f P_c^n$ before the aerodynamic forces are constructed and fed into the structural equations of dynamic equilibrium (see the introduction to this command). Therefore, this option covers the cases $P_c^{n+1} = P^{n+1}$ and $P_c^{n+1} = \bar{P}^{n+1}$, which explains the origin of the word “collocated” (characters).

NONCOLLOCATED

In this case, which is also the default value of PRESSURE, the pressure field P_c^{n+1} is set to $P_c^{n+1} = \frac{1}{\gamma}P^{n+1} - \frac{(1-\gamma)}{\gamma}P_c^n$ (or $P_c^{n+1} = \frac{1}{\gamma}\bar{P}^{n+1} - \frac{(1-\gamma)}{\gamma}P_c^n$), then converted to $P_c^{n+1-\alpha_f} = (1 - \alpha_f)P_c^{n+1} + \alpha_f P_c^n$ (here, γ is the γ parameter of the Generalized α method). This specific choice for P_c^{n+1} is consistent with the quadrature rule of the Generalized α method for evaluating the work done by an external force in the time-interval $[t^n, t^{n+1}]$. In other words, the pressure forces fed into the structural equations of equilibrium are based on $P_c^{n+1-\alpha_f} = \frac{(1-\alpha_f)}{\gamma}P^{n+1} + (1 - \frac{(1-\alpha_f)}{\gamma})P_c^n$. For $\gamma = \alpha_f = \frac{1}{2}$, $P_c^{n+1-\alpha_f} = P^{n+1}$, which explains the origin of the word “non collocated” (characters).

MATCHER

pathandfilename

Name (including path, if needed) of the structural matcher file (characters).

EMBEDDED

embeddedsurfaceid

Integer identification number of a surface defined in SURFACETOPO using 3-noded triangles (integer). This surface, which is to be embedded in an **AERO-F** CFD mesh for fluid-structure computations, is assumed to be identical to the wet surface of the finite element structural model and defined using the same nodes of this model. For this reason, when the EMBEDDED option is used, it is not currently necessary to provide a structure matcher file.

Next: [ATTRIBUTES](#), Previous: [AERO](#)

10 AEROHEAT (FEM-HEAT Only)

Command Statement:	AEROH
--------------------	--------------

The AEROH command statement is used to indicate that FEM-HEAT is to interact with a fluid code to perform an aerothermal (thermostructure-thermofluid) coupled simulation, to select a staggered time-integration algorithm (ALGORITHM subcommand), and specify the α_0 and α_1 parameters of the following prediction of the temperature of the structure at time-step t^{n+1} (ALGORITHM subcommand)

$$T_S^{n+1P} = T_S^n + \alpha_0 \Delta t_S^H \dot{T}_S^n + \alpha_1 \Delta t_S^H (\dot{T}_S^n - \dot{T}_S^{n-1})$$

The syntax for invoking this option is given below.

AEROH

ALGORITHM α_0 α_1

ALGORITHM

A0 This is the basic sequential partitioned procedure. It is similar to the A0 algorithm of the AERO command.

α_0 For $\alpha_0 = 0$ (and $\alpha_1 = 0$), no prediction is effected. Setting $\alpha_0 = 1$ (and $\alpha_1 = 0$) generates a first-order prediction, and setting $\alpha_0 = 1$ (and $\alpha_1 = \frac{1}{2}$) generates a second-order prediction. However, the optimal values of α_0 and α_1 depend on the specifics of the fluid and structure field time-integrators (see the CAS paper on energy conservation by Piperno and Farhat) (real).

α_1 See above (real).

Next: [BOFFSET](#), Previous: [AEROH](#)

11 ATTRIBUTES *S*

Command Statement: **ATTRIBUTES**

The ATTRIBUTES command statement is used to label an element with an attribute identification number for linear material and/or geometric properties, a composite or orthotropic shell, or anisotropic solid element attribute identification number, a frame or fiber-angle attribute identification number if this element has been defined as a composite or orthotropic shell or anisotropic solid element, and a hyper reduction coefficient when the mesh containing this element is sampled for the purpose of hyper reduction.

The attribute identification number for linear material properties and/or geometric properties is used in the MATERIAL command when specifying the linear material properties and/or geometric properties of a group of elements (see [MATERIAL](#)).

An “empty” (or “phantom”) element — that is, an element with zero generalized stiffness and mass matrices — can be useful in aeroelastic, aerothermal, and aerothermoelastic computations to facilitate the exchange of elastodynamic, thermal, and aerodynamic data between the fluid, thermal, and structural analyzers. Such an element can be specified by assigning it a negative attribute number. In this case, it is not necessary to specify a material for such an element as its properties will be ignored.

It is also not necessary to assign an attribute number to a rigid element or simple joint element (types 118–125 and 127) for which the default constraint method (see [CONSTRAINTS](#)) is to be used for enforcing the associated constraints. On the other hand, an attribute number should always be assigned to a revolute-joint-with-joint-driver element (type 126) or a joint spring combination element (types 220–227), regardless of whether the default constraint method is to be used or not for this element.

The hyper reduction coefficients are computed by the [RMSHC](#) command and outputted in the result file [SAMPLMSH](#) (see [OUTPUT](#)) which includes the header ATTRIBUTES. Hence, this file can be simply included in the **AERO-S** input file using the [INCLUDE](#) command (see [INTRODUCTION](#)).

AERO-S supports three input formats for this command that are described below. In the case of the first format, there should be as many lines as the number of finite elements. All formats can be mixed.

Note 1: When “empty” (or “phantom” elements) are used in the mesh, the FETI solvers are not guaranteed to work because of issues related to subdomain singularities. Hence, it is not recommended to use FETI solvers in such cases.

Note 2: Phantom elements should be used with care to avoid the generation of degrees of freedom without stiffness (and mass) and therefore avoid introducing artificial singularities in the solution of the problem of interest. In particular, it is strongly recommended to ensure that every node of every phantom element is also a node of a non-phantom element, and that every local degree of freedom of a phantom element is also a local degree of freedom of a non-phantom element. In other words, it is strongly recommended, for example, to avoid connecting a phantom shell element to a face of a solid (brick element) and use instead a phantom three-dimensional plane stress/plane strain element in this case.

Note 3: Only phantom shell elements and phantom plane stress/plane strain elements can transfer aeroelastic loads to a structure.

ATTRIBUTES

ELEMENT#	MAT_ATT#	CMP_ATT#	CMP_FRM#	HRC	HRCOEFF	[EXTFOL]
----------	----------	----------	----------	-----	---------	----------

or

ELEMENT#	MAT_ATT#	CMP_ATT#	THETA	θ_{Ref}	HRC	HRCCOEFF
[EXTFOL]						

or

STARTING_ELEMENT#	ENDING_ELEMENT#	MAT_ATT#	CMP_ATT#	CMP_FRM#

or

STARTING_ELEMENT#	ENDING_ELEMENT#	MAT_ATT#	CMP_ATT#	THETA	θ_{Ref}

or

STARTING_ELEMENT#	ENDING_ELEMENT#	IDENTITY

or

ELEMENT#	HRC	HRCCOEFF	[EXTFOL]

ELEMENT# Element number whose attribute numbers are to be specified (integer).
MAT_ATT# Group identification number (integer); **AERO-S** supports gaps in numbering.
Identifies material properties.
CMP_ATT# Composite (orthotropic or anisotropic) identification number (integer); **AERO-S** supports gaps in numbering. Identifies composite (orthotropic or anisotropic) properties. May be left blank if element is not a composite (orthotropic or anisotropic) element.
CMP_FRM# Composite frame identification number (integer); **AERO-S** supports gaps in numbering. May be left blank if element is not a composite (orthotropic or anisotropic) element.

THETA Keyword that must be spelled as **THETA** (characters). This keyword announces that the next item on the same input line is the value θ_{Ref} of **THETA**. This unusual **AERO-S** input is necessary to avoid parsing conflicts.

θ_{Ref} Reference angle in degrees between the axis defined by local nodes 1 and 2 of the element, and the material local x-axis defining the direction of the σ_{xx} and ϵ_{xx} of the constitutive law (real). When also using the **LAYN** and **LAYC** commands, the material x-axis corresponds to the direction of $E_1^{(k)}$ when $\theta_F^{(k)}$ is zero. If

this angle rather than `CMP_FRM#` is specified, the composite frame is internally generated by **AERO-S**.

<code>STARTING_ELEMENT#</code>	First element of a sequence of elements that have the same <code>MAT_ATT#</code> , <code>CMP_ATT#</code> , and <code>CMP_FRM#</code> (integer).
<code>ENDING_ELEMENT#</code>	Last element of a sequence of elements that have the same <code>MAT_ATT#</code> , <code>CMP_ATT#</code> , and <code>CMP_FRM#</code> (integer).
<code>MAT_ATT#</code>	Group identification number. Identifies material properties (integer).
<code>CMP_ATT#</code>	Composite identification number (integer). Identifies composite (orthotropic or anisotropic) properties. May be left blank if element is not a composite (orthotropic or anisotropic) element.
<code>CMP_FRM#</code>	Composite frame identification number (integer). May be left blank if element is not a composite (orthotropic or anisotropic) element.
<code>IDENTITY</code>	This keyword signals to the AERO-S code that each element in the range delimited by <code>STARTING_ELEMENT#</code> and <code>ENDING_ELEMENT#</code> has a material attribute identification number equal to its element identification number. For example, the sequence 1 3 <code>IDENTITY</code> means that element#1 has the material attribute identification 1, element#2 has the material attribute identification 2, and element#3 has the material attribute identification 3.
<code>HRC</code>	Sub-command keyword to request the application of a specified hyper reduction coefficient to the element-level reduced internal forces and moments and acquiring this coefficient (characters).
<code>HRCOEFF</code>	Hyper reduction coefficient computed by the RMSHC command and stored in the result file <code>SAMPLMSH</code> under OUTPUT (float).
<code>[EXTFOL]</code>	Optional sub-command keyword to request the application of the specified hyper reduction coefficient to all of the element-level reduced internal forces and moments and the element-level reduced follower external forces and moments (see FORCES) (characters).

Next: [EFRAMES](#), Previous: [ATTRIBUTES](#)

12 BEAM OFFSET

Command Statement:	BOFFSET
--------------------	----------------

The `BOFFSET` command statement is used to specify a beam's neutral axis offset from the line passing through its two end nodes. This command may be used for both Euler-Bernoulli and Timoshenko beam elements. For beam elements not listed under `BOFFSET`, a zero offset is used. If the `BOFFSET` command is absent altogether, then an offset of zero is used for all beam elements. If a listed element is neither an Euler-Bernoulli nor a Timoshenko beam element, then the offset is ignored for that element. The input format is as follows.

BOFFSET

	STARTING_ELEMENT#	ENDING_ELEMENT#	x	y	z
STARTING_ELEMENT#					First element of a sequence of elements that have the same offset specified by {x, y, z} (integer).
ENDING_ELEMENT#					Last element of a sequence of elements that have the same offset specified by {x, y, z} (integer). If ENDING_ELEMENT# is the same as STARTING_ELEMENT#, then the specified offset is applied to that element only (integer).
x					x component of the offset vector expressed in the global frame (float).
y					y component of the offset vector expressed in the global frame (float).
z					z component of the offset vector expressed in the global frame (float).

Next: [BINARY](#) Previous: [BOFFSET](#)

13 BEAM REFERENCE FRAMES

Command Statement: **EFRAMES**

The EFRAMES command statement is used to specify a beam's orientation which defines the position of the beam with respect to the global frame. The requirements for each beam frame are that one of the axis, **the local x-axis**, concurs with the longitudinal axis of the beam, and the remaining two axes complete an orthogonal triad. If this requirement is violated, **AERO-S** regenerates the local x-axis as well as the local z-axis.

For *flexible* beams, **AERO-S** also supports a run-time generation of frames that is activated either when the target flexible beam element is identified under this command and a third node is specified to generate the frame, or when a third node is found in the definition of a flexible beam element within the TOPOLOGY command. The only requirement for the third node is that it does not be colinear with the other two beam nodes that define the local x-axis (X). Under *this* command, the third node defines the X,Y plane. Under the TOPOLOGY command, it defines the X,Z plane. Using the third node option under the TOPOLOGY command also relieves the user from specifying the EFRAMES command. An example illustrating the third node option in the TOPOLOGY command can be found in FEM.d/fem\examples/Third_Node.d

Otherwise, the input format for this command is given below with the number of lines equal to the number of beams in the problem.

EFRAMES

ELEMENT#	S_{1x} S_{1y} S_{1z}	S_{2x} S_{2y} S_{2z}	S_{3x} S_{3y} S_{3z}
----------	----------------------------	----------------------------	----------------------------

or

ELEMENT#	THIRDNODE	third_node
----------	-----------	------------

ELEMENT# Element number where the beam frame is specified (integer).

$S_{1x} \ S_{1y} \ S_{1z}$ The first axis, **the local x-axis**, of the beam frame expressed in the global frame. This axis must concur with the longitudinal axis of the beam (floats).

$S_{2x} \ S_{2y} \ S_{2z}$ The second axis of the beam frame expressed in the global frame (floats).

$S_{3x} \ S_{3y} \ S_{3z}$ The third axis of the beam frame expressed in the global frame (floats).

THIRDNODE Keyword that must be spelled as THIRDNODE. This keyword announces that the next item on the same input line is the value `third_node#` of THIRDNODE. This unusual **AERO-S** input is necessary to avoid parsing conflicts (characters).

THIRD_NODE# Id number of an existing node to be considered as a third node of the *flexible* beam element identified by ELEMENT#, and which must be in the local x-y plane of the beam. The only requirement for the third node is that it does not be colinear with the other two beam nodes. The normalized vector node1 to node2 defines the local x axis. The normalized vector node1 to node3 defines the local y-axis. The local z-axis is automatically generated as cross product of the other two normalized axes.

Note 1: The above axes have to be normalized. By default, the beam frames concur with the global frame.

Next: [CONVECTION](#), Previous: [EFRAMES](#)

14 BINARY INPUT / OUTPUT

Command Statement:	BINARY
--------------------	---------------

When the finite element model of interest is very large, it may become necessary (for example, because of memory limitations), to organize the input data of an **AERO-S** computation in a set of binary distributed files, and more efficient to output its results in another set of binary distributed files. This requires:

- Using a version of **aeros** that is compiled in the distributed memory execution mode.
- Generating and using a mesh partition for the target simulation.
- Generating the binary distributed input files associated with the above mesh partition, as explained below.
- Inputting these binary distributed input files and if desired requesting the binary distributed format for the output files, using the command **BINARY** explained herein.

To decompose the input data contained in an initial instance of the ASCII Input Command Data file according to a generated mesh partition and reorganize it in a set of binary distributed input files, the user should use the specialized version of the software **SOWER** that is embedded in **AERO-S**. The generated binary distributed input files can then be inputted via this command in the final instance of the ASCII Input Command Data file. This command can also be used to request outputting the results marked for output in [OUTPUT](#) and/or [OUTPUT6](#) in the binary distributed format compatible with the generated mesh partition. In that case, the original **SOWER** (see **SOWER**'s User's Reference Manual) should be used to convert these output files into the ASCII format suitable for postprocessing by **XPost**.

If memory is not an issue, **AERO-S** can also operate directly on the global set of input data — that is, on the standard content of the ASCII Input Command Data file.

Note 1: If **aeros** is not compiled in the distributed memory execution mode, it can only generate ASCII output files. Most of these are in a format that is suitable for postprocessing by the **XPost** software, but some are in a format that is suitable for postprocessing by **gnuplot**.

Note 2: Any command present in the initial instance of the ASCII Input Command Data file and whose description in this User's Reference Manual is marked by "**S**" can be assumed to be contained in one of the aforementioned binary distributed input files. However, if that command contains both an algorithmic parameter as well as data, only the data will be contained in the appropriate distributed binary input file. Therefore, the command itself should be kept in the updated ASCII Input Command Data file together with the algorithmic parameter, but without the data. For example, the command [IDISPLACEMENTS](#) specifies both an amplification factor and an initial displacement data. The initial displacement data will be included in the aforementioned binary distributed input file, but the amplification factor will not. This is to allow the user to change simple things such as this amplification factor without having to regenerate the binary distributed input files.

The syntax for this command is as follows.

BINARY

```
BINARYINPUT flagBIN [<pathandfileprefix>]  
BINARYOUTPUT flagBOUT
```

BINARYINPUT	flagBIN	This pair of sub-command keyword (characters) and corresponding value (characters) can be used to choose between reading the binary distributed input files designated by <pathandfileprefix>, or the standard ASCII global input data contained in the initial instance of the ASCII Input Command Data file. flagBIN should be inputted on the same line as BINARYINPUT; it can take one of the following values:
	on	This setting chooses the specified binary distributed input files as input. In this case, the user should check first that all binary distributed input files have been generated and that the associated filenames and path are those referred to in <pathandfileprefix>.
	off	This setting, which is also the default setting, chooses the standard ASCII global data. In this case, the user should check first that all needed input data is included in the ASCII Input Command Data file.
<pathandfileprefix>		This entry is relevant only when flagBIN is set to on. By default, AERO-S expects the binary input data command to be contained in four binary distributed files named INPUT.dec, INPUT.con, INPUT.sub, and INPUT.msh. Alternatively, this data can be contained in four equivalent files that are named differently, as long as they share the same path and filename prefix — that is, these four binary distributed files must have the same path and can be named fileprefix.dec (substitute for INPUT.dec), fileprefix.con (substitute for INPUT.con), fileprefix.sub (substitute for INPUT.sub), and prefixfile.msh (substitute for INPUT.msh). In the latter case, <pathandfileprefix>

BINARYOUTPUT	flagBOUT	specifies the common filename prefix (and file path if needed) (characters).
		This pair of sub-command keyword (characters) and corresponding value (characters) can be used to choose between outputting the results specified in OUTPUT and/or OUTPUT6 in the binary distributed format, or the standard ASCII global format. flagBOUT should be inputted on the same line as BINARYOUTPUT; it can take one of the following values:
	On	This setting chooses the binary distributed format for all output files that support it.
	Off	This setting, which is also the default setting, chooses the standard ASCII global format for all output files.

Next: [DISPLACEMENTS](#), Previous: [BINARY](#)

15 BOUNDARY CONVECTION *S*

Command Statement:	CONVECTION [LOADSET_ID]
--------------------	--------------------------------

The CONVECTION command statement is used to specify nodal convection type boundary conditions. Each node can have only one degree of freedom. This command statement can be used to solve a prescribed boundary convection problem for both statics and dynamics. The input format is given below.

Note 1: This command contributes to the construction of the right-hand side vector only. For the left-hand side (stiffness matrix) contribution, AERO-S uses boundary convection elements and information specified in the MATERIAL command.

CONVECTION [LOADSET_ID]

NODE#	H-COEFF	AREA	TA
-------	---------	------	----

LOADSET_ID	Optional non-negative integer which identifies explicitly the "load" set to which the source term generated by this command belongs to (integer). The default value is 0. Hence, the CONVECTION command can be repeated as many times as desired within the same input file using each time a different value for LOADSET_ID and different data. The LOADCASE command can refer to LOADSET_ID to define one or multiple "load" cases for static analysis (see the STATICS command and the explanation of its sub-command keyword CASES), and/or the "load" case for dynamic analysis.		
NODE#	Node number where the convection is specified (integer).		
H_COEFF	Convection coefficient at the prescribed node (float).		
AREA	Value of the node cross-sectional area (float).		
TA	Ambient temperature around the node (float).		

Next: [FLUX](#), Previous: [CONVECTION](#)

16 BOUNDARY DISPLACEMENTS *S*

Command Statement:	DISPLACEMENTS
--------------------	----------------------

The **DISPLACEMENTS** command is used to prescribe nodal displacements and/or rotations, either directly, or via the definition of a surface using the command [SURFACETOPO](#). In the latter case, the specified displacement or rotation is applied to each specified local degree of freedom of each node of that surface. The user can specify up to three displacements per node if a node can have up to three degrees of freedom, and up to three displacements and three rotations if it can have up to six degrees of freedom.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: This command can also be used to specify nodal Dirichlet boundary conditions for a time-domain or frequency-domain acoustic simulation by setting **DOF#** to 8 (see below). Hence, it can also serve as an alternative to the [ATDDIR](#) and [HDIR](#) commands.

Note 3: In the context of a linearized (perturbation) analysis, the displacements boundary conditions specified under this command are interpreted as displacement boundary perturbations.

The following two formats are available for this command and can be mixed.

DISPLACEMENTS

NODE#	DOF#	VALUE
--------------	-------------	--------------

SURFACE	SURFACE#	DOF#	VALUE
----------------	-----------------	-------------	--------------

NODE#	Node number where the displacement or rotation is specified (integer).
DOF#	Degree of freedom local number where the displacement or rotation is specified (integer).
VALUE	Value of the specified displacement or rotation (real).
SURFACE	Keyword indicating that a surface defined in SURFACETOPO is to be identified next by its integer identification number (characters).
SURFACE#	Integer identification of the surface defined in SURFACETOPO where the force VALUE is specified (integer).

??? from here until ???END lines may have been inserted/deleted

Next: [FORCES](#), Previous: [DISPLACEMENTS](#)

17 BOUNDARY FLUXES

Command Statement:	FLUX [LOADSET_ID]
--------------------	--------------------------

The **FLUX** command statement is used to specify nodal heat sources (for example, products of nodal areas and nodal values of finite element fluxes) either directly, or via the definition of a surface using the command [SURFACETOPO](#). In the latter case, a specified heat source is applied to each node of the identified surface. This command statement can be used to define both statics and dynamics heat problems with prescribed temperature flux boundary conditions. Its input format is given below.

FLUX [LOADSET_ID]

NODE#	VALUE
--------------	--------------

SURFACE	SURFACE#	VALUE
----------------	-----------------	--------------

LOADSET_ID	Optional non-negative integer which identifies explicitly the "load" set to which the source term generated by this command belongs to (integer). The default value is 0. Hence, the FLUXES command can be repeated as many times as desired within the same input file using each time a different value for LOADSET_ID and different data. The LOADCASE command can refer to LOADSET_ID to define one or multiple "load" cases for static analysis (see the STATICS command and the explanation of its sub-command keyword CASES), and/or the "load" case for dynamic analysis.
NODE#	Node number where the flux is specified (integer).
VALUE	Value of the prescribed boundary flux (float).
SURFACE	Keyword indicating that a surface defined in SURFACETOPO is to be identified next by its integer identification number (characters).
SURFACE#	Integer identification of the surface defined in SURFACETOPO where the nodal heat source VALUE is specified (integer).

Next: [TEMPERATURES](#), Previous: [FLUX](#)

18 BOUNDARY FORCES *S*

Command Statement:	FORCES [LOADSET_ID]
--------------------	----------------------------

The **FORCES** command is used to prescribe external nodal forces and/or moments, either directly, or via the definition of a surface using the [SURFACETOPO](#) command. In the latter case, the specified force

or moment is applied to each specified local degree of freedom of each node of that surface. The user can specify up to three forces per node if a node has three degrees of freedom attached to it, and up to three forces and three moments if it has six degrees of freedom attached to it.

By default, all prescribed forces and moments are interpreted as being of the *axial* type — that is, as being defined in the *fixed* nodal degree of freedom reference frames (see [NODES](#) and [NFRAMES](#)).

However, if a node has rotational degrees of freedom, the user can specify that the forces and/or moments prescribed at this node are of the *follower* type — that is, they act in a direction that remains constant in the local frame attached to the node where they are applied. This local frame coincides with the nodal degree of freedom reference frame (see [NODES](#) and [NFRAMES](#)) in the undeformed configuration. In the deformed configuration, the orientation of this local frame is defined by the rotation of the node to which it is attached. In other words, the specified nodal force or moment "follows" in this case the rotation of the node to which it is applied.

Finally, the user can also time-vary the specified forces and moments using the [MFTT](#) command.

Note 1: By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: This command can also be used to specify nodal Neumann boundary conditions for a time-domain acoustic simulation by setting `DOF#` to 8 (see below). Hence, it can also serve as an alternative to the [ATDNEU](#) command (see [ATDNEU](#)).

Note 3: Specifying a follower force or moment leads to an unsymmetric tangent "load" stiffness matrix during a [NONLINEAR](#) analysis.

The following two formats are available for this command and can be mixed.

FORCES [LOADSET_ID]		
NODE#	DOF#	VALUE TYPE
SURFACE SURFACE#	DOF#	VALUE TYPE

LOADSET_ID	Optional non-negative integer which identifies explicitly the "load" set to which the source term generated by this command belongs to (integer). The default value is 0. Hence, the <code>FORCES</code> command can be repeated as many times as desired within the same input file using each time a different value for <code>LOADSET_ID</code> and different data. The LOADCASE command can refer to <code>LOADSET_ID</code> to define one or multiple "load" cases for static analysis (see the STATICS command and the explanation of its sub-command keyword <code>CASES</code>), and/or the "load" case for dynamic analysis.
NODE#	Node number where the force or moment is specified (integer).
DOF#	Degree of freedom local number where the force or rotation is specified (integer).
VALUE	Value of the specified force or moment (real).

TYPE	By default, all specified nodal forces and moments are considered to be of the axial type. However, if this parameter is set to FOLLOWER and the node where a force or moment is specified has rotational degrees of freedom, this specified nodal force or moment is considered to be of the follower type (characters).
SURFACE	Keyword indicating that a surface defined in SURFACETOPO is to be identified next by its integer identification number (characters).
SURFACE#	Integer identification of the surface defined in SURFACETOPO where the force or moment VALUE is specified (integer).

Next: [BUCKLE](#), Previous: [FORCES](#)

19 BOUNDARY TEMPERATURES *S*

Command Statement:	TEMPERATURES
--------------------	---------------------

The TEMPERATURES command statement is used to specify prescribed nodal temperature type boundary conditions either directly, or via the definition of a surface using the command [SURFACETOPO](#). In the latter case, a specified temperature is applied to each node of the identified surface. For structural analysis using **AERO-S**, this command statement causes **AERO-S** to construct a thermal load based on the prescribed nodal temperatures and the reference temperatures (see [MATLAW](#), see [TOPOLOGY](#), and use this load as usual in a structural analysis).

The input format is given below.

TEMPERATURES

NODE#	VALUE
-------	-------

SURFACE	SURFACE#	VALUE
---------	----------	-------

NODE#	Node number where the temperature is specified (integer).
VALUE	Value of the prescribed temperature (float).
SURFACE	Keyword indicating that a surface defined in SURFACETOPO is to be identified next by its integer identification number (characters).
SURFACE#	Integer identification of the surface defined in SURFACETOPO where the temperature VALUE is specified (integer).

Next: [COMMENTS](#), Previous: [TEMPERATURES](#)

20 BUCKLING

Command Statement:	BUCKLE
--------------------	---------------

The BUCKLE command statement is to be used together with the GEPS, IDISP6 (with $\alpha = -1$), and EIGEN command statements to request the buckling analysis of a given structure.

When the EIGEN, GEPS, and IDISP6 (with $\alpha = -1$) command statements are specified in the input file, the BUCKLE command statement signals to FEM that the eigenvalue problem to be solved is

$KU = -\lambda K_G U$, where K_G is the geometric stiffness matrix associated with a displacement field specified under the IDISP6 command statement, and whose computation is triggered by the presence of the GEPS command statement in the input file. For each eigenvalue λ , the corresponding buckling load is λf where f is the load that created the displacement field specified under the IDISP6 command.

An example input file that illustrates a buckling analysis can be found in FEM.d/fem\examples/Buckle.d. The syntax for invoking this option is given below.

BUCKLE

Next: [COMPOSITE](#), Previous: [BUCKLE](#)

21 COMMENTS

The '*' at the beginning of a line indicates that the subsequent input on the same line is a comment. For FEM, it can be placed anywhere.

Next: [CFRAMES](#), Previous: [COMMENTS](#)

22 COMPOSITE (OR ORTHOTROPIC SHELL OR ANISOTROPIC SOLID)

Command Statement: **COMPOSITE**

The COMPOSITE command statement is used to specify the material properties of a composite shell, or orthotropic shell, or anisotropic solid element. All information concerning the constitutive coefficients or the composite layering is input using this command.

Currently, this command supports only the solid elements type 17, type 23, type 24, type 25, type 72, type 91 and type 92, and the (Kirchhoff) shell elements type 20 and type 2020.

For the above solid elements, the constitutive matrix is inputted using the COEF sub-command. This matrix is defined in the local frame of the element specified in the CFRAMES command. Entries in the matrix that are not specified are determined from symmetry if appropriate, or are set to zero.

For the shell elements, the composite (or orthotropic) constitutive law can be prescribed either by entering the $C_{i,j}$ coefficients of the 6 by 6 constitutive matrix, or by defining the material properties

and geometrical characteristics for each layer of the composite. In the first case, the sub-command COEF should be used. In the second case, either of the sub-commands LAYC or LAYN can be used, depending on whether the coupling between bending and membrane effects is to be enforced or not,

respectively. When many layers of the composite are made of the same orthotropic material, the sub-commands `LAYD` and `LAYMAT` can be used instead of the sub-command `LAYC` to simplify the data entry process. Likewise, in similar circumstances, the sub-command `LAYO` and `LAYMAT` can be used instead of the sub-command `LAYN`.

The input format for the `COMPOSITE` command is given below.

`COMPOSITE`

<code>COEF</code>	<code>attribute#</code>	
<code>ROW# (<i>i</i>)</code>	<code>COLUMN# (<i>j</i>)</code>	<code>VALUE ($C_{i,j}$)</code>
.		
.		
.		
<code>ROW# (<i>i</i>)</code>	<code>COLUMN# (<i>j</i>)</code>	<code>VALUE ($C_{i,j}$)</code>

`COEF` Sub-command keyword used to directly input the coefficients of the constitutive law. Since the 6 by 6 constitutive matrix is symmetric, up to 21 independent coefficients may follow.

`attribute#` Integer value that corresponds to the composite (or orthotropic or anisotropic) attribute of the element.

`ROW#` Integer value that corresponds to the row *i* of the coefficient $C_{i,j}$.

A row can be skipped if all its entries are zero.

`COLUMN#` Integer value that corresponds to the column *j* of the coefficient $C_{i,j}$.

`VALUE` Real value of the coefficient $C_{i,j}$.

For a solid element, the constitutive matrix C relates the stresses to the engineering strains in the element's local frame system {x ; y ; z} as follows:

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{pmatrix} = \begin{pmatrix} C_{1,1} & C_{1,2} & C_{1,3} & C_{1,4} & C_{1,5} & C_{1,6} \\ C_{1,2} & C_{2,2} & C_{2,3} & C_{2,4} & C_{2,5} & C_{2,6} \\ C_{1,3} & C_{2,3} & C_{3,3} & C_{3,4} & C_{3,5} & C_{3,6} \\ C_{1,4} & C_{2,4} & C_{3,4} & C_{4,4} & C_{4,5} & C_{4,6} \\ C_{1,5} & C_{2,5} & C_{3,5} & C_{4,5} & C_{5,5} & C_{5,6} \\ C_{1,6} & C_{2,6} & C_{3,6} & C_{4,6} & C_{5,6} & C_{6,6} \end{pmatrix} \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{pmatrix}$$

For a (Kirchhoff) shell element, the constitutive matrix C relates the forces and moments to the mid-surface strains and curvatures in the shell element's local frame system {x ; y ; z} as follows:

$$\begin{pmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{pmatrix} = \begin{pmatrix} C_{1,1} & C_{1,2} & C_{1,3} & C_{1,4} & C_{1,5} & C_{1,6} \\ C_{1,2} & C_{2,2} & C_{2,3} & C_{2,4} & C_{2,5} & C_{2,6} \\ C_{1,3} & C_{2,3} & C_{3,3} & C_{3,4} & C_{3,5} & C_{3,6} \\ C_{1,4} & C_{2,4} & C_{3,4} & C_{4,4} & C_{4,5} & C_{4,6} \\ C_{1,5} & C_{2,5} & C_{3,5} & C_{4,5} & C_{5,5} & C_{5,6} \\ C_{1,6} & C_{2,6} & C_{3,6} & C_{4,6} & C_{5,6} & C_{6,6} \end{pmatrix} \begin{pmatrix} \epsilon_x^0 \\ \epsilon_y^0 \\ \gamma_{xy}^0 \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{pmatrix}$$

The element's local axes are defined with respect to the global reference frame by the three vectors defined in the **CFRAMES** command for the corresponding composite (orthotropic, or anisotropic) frame number that is specified in the **ATTRIBUTES** section. For shell elements, the constitutive matrix, C , can be decomposed into sub-matrices containing the bending and membrane properties of the shell element:

$$C = \begin{pmatrix} C_{mm} & C_{mb} \\ C_{bm} & C_{bb} \end{pmatrix}$$

Note the absence of the transverse shear behavior, as can be expected from a Kirchhoff type shell element. In the case of an isotropic material, or a single layer orthotropic or orthotropic material, there is no coupling between the bending and membrane behavior:

$$C_{mb} = C_{bm} = 0_{3 \times 3}$$

For an isotropic material, the membrane constitutive matrix can be defined in terms of the Young's modulus, E , Poisson's ratio, ν , and shell thickness, h as:

$$C_{mm} = \begin{pmatrix} \frac{Eh}{(1-\nu^2)} & \frac{\nu Eh}{(1-\nu^2)} & 0 \\ \frac{\nu Eh}{(1-\nu^2)} & \frac{Eh}{(1-\nu^2)} & 0 \\ 0 & 0 & \frac{Eh}{2(1+\nu)} \end{pmatrix}$$

while the bending constitutive matrix is given by:

$$C_{bb} = \begin{pmatrix} \frac{Eh^3}{12(1-\nu^2)} & \frac{\nu Eh^3}{12(1-\nu^2)} & 0 \\ \frac{\nu Eh^3}{12(1-\nu^2)} & \frac{Eh^3}{12(1-\nu^2)} & 0 \\ 0 & 0 & \frac{Eh^3}{24(1+\nu)} \end{pmatrix}$$

Using the **COEF** sub-command, different values of E , ν , and h can be utilized for building the membrane and bending components of the constitutive law, for example, if one wishes to adjust in some specific manner the bending and membrane behaviors of the shell element.

When the sub-command keyword **COEF** is used, the **MATERIAL** (see [MATERIAL](#)) command must be used as follows: the **density per unit surface** must be input at the fourth position after the attribute number, and for shells, the total thickness must also be defined at the seventh position after the attribute number if a stress analysis is requested.

For shell elements, if the pre-integrated constitutive matrix C is not available, the sub-command keywords **LAYC** or **LAYN** (or their counterparts **LAYD**, **LAYO** and **LAYMAT**) should be used to input the material properties and geometrical characteristics of each composite layer. If either **LAYN** or **LAYO** is used, no coupling between bending and membrane effects is enforced explicitly during integration through

the thickness — that is: $C_{1,4} = C_{1,5} = C_{1,6} = 0$, $C_{2,4} = C_{2,5} = C_{2,6} = 0$ and $C_{3,4} = C_{3,5} = C_{3,6} = 0$.

The input format given below is the same for both sub-commands LAYC and LAYN.

LAYC (or LAYN)		attribute#
k	$E_1^{(k)}$	$E_2^{(k)}$
	$\nu_{12}^{(k)}$	$G_{12}^{(k)}$
	$\mu_{1,12}^{(k)}$	$\mu_{2,12}^{(k)}$
	$\rho^{(k)}$	$h^{(k)}$
		$\theta_F^{(k)}$
.		
.		
.		
k	$E_1^{(k)}$	$E_2^{(k)}$
	$\nu_{12}^{(k)}$	$G_{12}^{(k)}$
	$\mu_{1,12}^{(k)}$	$\mu_{2,12}^{(k)}$
	$\rho^{(k)}$	$h^{(k)}$
		$\theta_F^{(k)}$

The input format given below is the same for both sub-commands LAYD and LAYO.

LAYD (or LAYO)		attribute#
k	LAYER_MATERIAL_ID	$h^{(k)}$
		$\theta_F^{(k)}$
LAYMAT		
LAYER_MATERIAL_ID	$E_1^{(k)}$	$E_2^{(k)}$
	$\nu_{12}^{(k)}$	$G_{12}^{(k)}$
	$\mu_{1,12}^{(k)}$	$\mu_{2,12}^{(k)}$
	$\rho^{(k)}$	
or		
LAYER_MATERIAL_ID	$E_1^{(k)}$	$E_2^{(k)}$
	$\nu_{12}^{(k)}$	$G_{12}^{(k)}$
	$\rho^{(k)}$	
.		
.		
.		
LAYER_MATERIAL_ID	$E_1^{(k)}$	$E_2^{(k)}$
	$\nu_{12}^{(k)}$	$G_{12}^{(k)}$
	$\mu_{1,12}^{(k)}$	$\mu_{2,12}^{(k)}$
	$\rho^{(k)}$	
or		
LAYER_MATERIAL_ID	$E_1^{(k)}$	$E_2^{(k)}$
	$\nu_{12}^{(k)}$	$G_{12}^{(k)}$
	$\rho^{(k)}$	

LAYC, LAYD, LAYN, or LAYO Sub-command keyword used for inputting geometrical and material properties for each layer of a composite shell element (characters).

attribute# Integer value that corresponds to the composite (orthotropic) attribute of the element.

k Integer value that corresponds to the layer number.

$E_1^{(k)}$ Young's modulus in the local direction $1^{(k)}$ (that is, in the direction of the fibers for the k^{th} layer).

$E_2^{(k)}$	Young's modulus in the local direction $2^{(k)}$ (that is, in the direction orthogonal to the fibers for the k^{th} layer).
$\nu_{12}^{(k)}$	Poisson's ratio for transverse strain in the local direction $2^{(k)}$ when stressed in the local direction $1^{(k)}$ for $\sigma_1^{(k)} = \text{constant}$ and all other stresses zero.
$G_{12}^{(k)}$	Transverse shear modulus in the plane $1^{(k)} - 2^{(k)}$ of the layer.
$\mu_{1,12}^{(k)}$	Coefficient of mutual influence of the first kind which characterizes stretching in the local direction $1^{(k)}$ caused by shear in the plane $1^{(k)} - 2^{(k)}$ of the layer for $\tau_{12}^{(k)} = \text{constant}$ and all other stresses zero.
$\mu_{2,12}^{(k)}$	Coefficient of mutual influence of the first kind which characterizes stretching in the local direction $2^{(k)}$ caused by shear in the plane $1^{(k)} - 2^{(k)}$ of the layer for $\tau_{12}^{(k)} = \text{constant}$ and all other stresses zero.
$\rho^{(k)}$	Density (mass per unit volume) of the material of the k^{th} layer.
$h^{(k)}$	Thickness of the k^{th} layer.
$\theta_F^{(k)}$	Angle between a reference vector and the fibers in the layer that defines the orientation of these fibers. If <code>CMP_FRM#</code> is specified as an attribute for an element containing this layer under the command ATTRIBUTES , the reference vector is the projection onto the plane of that element of the first of the three vectors defining a local frame for this element and specified in the CFRAMES command. On the other hand, if a reference angle θ_{Ref} is specified as an attribute for an element containing this layer under the command ATTRIBUTES , the reference vector is the vector obtained by rotating the directional edge connecting local nodes 1 and 2 of this element around its normal by an angle equal to θ_{Ref} . In both cases, this angle must be inputed in degrees.
LAYER_MATERIAL_ID	Identifier of a set of material properties (integer).
LAYMAT	Sub-command keyword that can be used for inputting the properties of a layer when it is made of a two-dimensional orthotropic material (characters).

The report number CU-CAS-94-16, "The 3-node Composite Shell and Isotropic Timoshenko Beam Elements" by Fran c cois M. Hemez, provides a detailed description of the theory and implementation for the type-20 composite (orthotropic) shell element.

Next: [CONDITION](#), Previous: [COMPOSITE](#)

23 COMPOSITE (ORTHOTROPIC SHELL OR ANISOTROPIC SOLID) ELEMENT FRAMES

Command Statement:	CFRAMES
--------------------	----------------

The CFRAMES command statement is used to specify the orientation of composite laminates and orthotropic or anisotropic elements with respect to the global reference frame. The input format of this command is given below with the number of lines equal to the number of different composite (or

orthotropic or anisotropic) element frames referenced in the ATTRIBUTES command.

CFRAMES

CMP_FRM#	$S_{1x} \ S_{1y} \ S_{1z}$	$S_{2x} \ S_{2y} \ S_{2z}$	$S_{3x} \ S_{3y} \ S_{3z}$
----------	----------------------------	----------------------------	----------------------------

CMP_FRM# Identification of a composite frame (integer).

$S_{1x} \ S_{1y} \ S_{1z}$ The first axis, **the local x-axis**, expressed in the global frame. For LAYC and LAYN type composites, the orientation of the fibers of the layers should be defined with respect to the projection of this axis onto the plane of the element.

$S_{2x} \ S_{2y} \ S_{2z}$ The second axis of the frame expressed in the global frame (floats).

$S_{3x} \ S_{3y} \ S_{3z}$ The third axis of the frame expressed in the global frame (floats).

Next: [CONSTRAINTS](#), Previous: [CFRAMES](#)

24 CONDITION NUMBER

Command Statement:	CONDITION
--------------------	------------------

The CONDITION command statement is used to request the evaluation of the condition number of the system being solved by FEM. The input format is given below.

CONDITION

TOLERANCE	MAXITR
-----------	--------

TOLERANCE Error tolerance for computing the lowest and highest eigenvalues of the finite element model using the inverse power and power methods, respectively (float). The default value is 10^{-3} .

MAXITR Maximum number of iterations for computing the lowest and highest eigenvalues of the finite element model using the inverse power and power methods, respectively (integer). The default value is 100.

Next: [CONTACTSURFACES](#), Previous: [CONDITION](#)

25 CONSTRAINTS

Command Statement:	CONSTRAINTS
--------------------	--------------------

The **CONSTRAINTS** command statement is used to specify a default method for enforcing the constraints defined or associated with a problem. Its input format is given below.

Note 1: The Lagrange multiplier method for enforcing constraints associated with contact (see [CONTACTSURFACES](#)) or tied (see [TIEDSURFACES](#)) surfaces is currently available for all analyses. However for enforcing constraints of other origins, it is currently available for all but explicit dynamic analysis. This method leads to an indefinite system of equations. Currently, only the `spooles` and `mumps` direct solvers with pivoting turned on, the `superlu` direct solver, and the `gmres` and `FETI DP` iterative solvers support this approach for solving constrained systems (see [STATICS](#)).

Note 2: The augmented Lagrangian method for enforcing constraints is currently available only for [NONLINEAR](#) static and [NONLINEAR](#) implicit dynamic analyses.

Note 3: The elimination method is supported only by the following equation solvers (see [STATICS](#)): `skyline`, `sparse`, `mumps` in the context of a single domain, and `spooles`. Furthermore, this method is currently supported only when applied to all constraints: in other words, it cannot be combined with another method for enforcing constraints.

CONSTRAINTS

METHOD

METHOD	Specified default method for enforcing constraints (characters). This specified method can be overruled in the CONTACTSURFACES , MATERIAL , LMPC , NODALCONTACT , and TIEDSURFACES commands.	
multipliers	In this case, which is also the default case, AERO-S uses the Lagrange multiplier method for enforcing all constraints associated with the LMPC , NODALCONTACT , CONTACTSURFACES , and TIEDSURFACES commands, and with joint and rigid elements (see TOPOLOGY). Warning: before relying on this default value of <code>METHOD</code> , the user should read above the notes describing the scope of the Lagrange multiplier method.	
elimination	In this case, AERO-S uses an elimination method for enforcing all constraints associated with the LMPC and TIEDSURFACES commands, and with joint and rigid elements (see TOPOLOGY).	
beta	penalty	In this case, AERO-S uses the penalty method for enforcing all constraints associated with the LMPC , NODALCONTACT , CONTACTSURFACES , and TIEDSURFACES commands, and with joint and rigid elements (see TOPOLOGY). The parameter <code>beta</code> should be a large positive number, typically of the order of 10^8 (no default value is provided).
beta	augmented	In this case, AERO-S uses the augmented Lagrangian method for enforcing all constraints associated with the LMPC , NODALCONTACT , CONTACTSURFACES , and TIEDSURFACES commands, and with joint and rigid elements (see TOPOLOGY). The parameter <code>beta</code> should be a large positive number, typically of the order of 10^8 (no default value is provided).

Next: [CONTROL](#), Previous: [CONSTRAINTS](#)

26 CONTACT SURFACES

Command Statement:	CONTACTSURFACES
--------------------	------------------------

The CONTACTSURFACES command statement can be used to enforce contact laws between pairs of surfaces defined using the [SURFACETOPO](#) command, for static and dynamic analyses only. Surface interactions are detected using the ACME library's search module.

Note 1: In general, the master surface should be chosen as that with the coarser discretization. Setting the master and slave contact surfaces to the same surface activates the numerical treatment of self-contact.

Note 2: For an explicit dynamic analysis, both contact detection and enforcement steps are performed using ACME. For an implicit dynamic analysis, both contact detection and enforcement steps are performed using **AERO-S**.

Note 3: For the enforcement of contact surface constraints by ACME's Lagrange multiplier method in explicit dynamic analysis, the discrete kinematic constraint equations are defined using a node-to-segment approach, and the contact forces are computed using one of four enforcement models that are supported in this case and this case only: frictionless (default), constant friction, velocity dependent friction, and pressure dependent friction.

Note 4: For the enforcement of contact surface constraints by either the Lagrange multiplier method in static and implicit dynamic analyses, or the penalty method in static, explicit or implicit dynamic analyses, other discrete kinematic constraint equations are defined using a segment-to-segment (mortar) approach. In these cases, only the frictionless contact model is supported.

Note 5: The enforcement of contact surface constraints by the Lagrange multiplier method in static and implicit dynamic analyses is supported only by the FETI-DP family of solvers.

Note 6: The enforcement of contact surface constraints by the elimination method is not supported.

CONTACTSURFACES

For statics or implicit dynamics

SURF_PAIR_ID#	MASTER	SLAVE	MORTAR_TYPE	NORMAL_TOL	TANGENTIAL_TOL	CONSTRAINT_METHOD
---------------	--------	-------	-------------	------------	----------------	-------------------

For explicit dynamics, frictionless contact, and ACME's Lagrange multiplier method for enforcing contact

SURF_PAIR_ID#	MASTER	SLAVE	KPART_TYPE	NORMAL_TOL	TANGENTIAL_TOL	NUM_ITER	CONVERG_TOL
---------------	--------	-------	------------	------------	----------------	----------	-------------

For explicit dynamics, ACME's constant friction contact, and ACME's Lagrange multiplier method for enforcing contact

SURF_PAIR_ID#	MASTER	SLAVE	KPART_TYPE	NORMAL_TOL	TANGENTIAL_TOL	NUM_ITER	CONVERG_TOL
FRIC_COEF							

For explicit dynamics, ACME's velocity dependent friction contact model, and ACME's Lagrange multiplier method for enforcing contact

SURF_PAIR_ID#	MASTER	SLAVE	KPART_TYPE	NORMAL_TOL	TANGENTIAL_TOL	NUM_ITER	CONVERG_TOL
---------------	--------	-------	------------	------------	----------------	----------	-------------

STATIC_COEF	DYNAMIC_COEF	VELOCITY_DECAY
-------------	--------------	----------------

For explicit dynamics, ACME's pressure dependent friction contact model, and ACME's Lagrange multiplier method for enforcing contact

SURF_PAIR_ID#	MASTER	SLAVE	KPART_TYPE	NORMAL_TOL	TANGENTIAL_TOL	NUM_ITER	CONVERG_TOL
FRIC_COEF	REF_PRES	OFFSET_PRES	PRES_EXP				

SURF_PAIR_ID#	Id number of the surface pair to be described (integer).						
MASTER	Identification of the master (mortar method) surface (see SURFACETOPO) (integer).						
SLAVE	Identification of the slave (mortar method) surface (see SURFACETOPO) (integer).						
CONSTRAINT_METHOD	Method for enforcing the associated constraints (characters). The default method is set in CONSTRAINTS and used whenever this entry is omitted.						
multipliers	The Lagrange multiplier method.						
elimination	The elimination method.						
penalty	The penalty method. The parameter beta should be a large positive number, typically of the order of 10^8 (no default value is provided).						
[beta]	Mortar type: 0 = standard, 1 = dual, default value is 0 (integer).						
MORTAR_TYPE							
NORMAL_TOL	Normal search tolerance used by ACME to identify interactions, default value is 0.1 (float) (see Figs. 1.2 and 1.3 in Section 1.3 of ACME's User Reference Manual). This tolerance should be larger than the maximum relative displacement occurred during one load step of a static simulation or one time-step of a dynamic one. If penetration occurs, this tolerance should be increased.						
TANGENTIAL_TOL	Tangential search tolerance used by ACME to identify interactions, default value is 0.001 (float) (see Figs. 1.2 and 1.3 in Section 1.3 of ACME's User Reference Manual).						
KPART_TYPE	Kinematic partitioning type for explicit dynamics: 0 = fixed, 1 = automatic, default value is 0 (integer).						
NUM_ITER	Maximum number of predictor-corrector iterations to be performed at each time step for explicit dynamics, default value is 5 (integer).						
CONVERG_TOL	Convergence tolerance of the predictor-corrector iteration loop for explicit dynamics, default value is 1.0e-10 (float).						
FRIC_COEF	Friction coefficient (float).						
STATIC_COEF	Static friction coefficient (float).						
DYNAMIC_COEF	Dynamic friction coefficient (float).						
VELOCITY_DECAY	Velocity decay parameter (float).						
REF_PRES	Reference pressure (float).						
OFFSET_PRES	Offset pressure (float).						
PRES_EXP	Pressure exponent (float).						

Next: [CONWEP](#), Previous: [CONTACTSURFACES](#)

27 CONTROL STATEMENT

Command Statement:	CONTROL
--------------------	----------------

AERO-S always generates a performance file named `FNAME.timing` which reports on the complexity, memory, and CPU resources associated with the performed computation. By default, the prefix `FNAME` is set to the name of the **AERO-S** input file (**AERO-S** also outputs some of these performance results on the screen). This optional command statement can be used to reset the value of the prefix `FNAME`. This prefix is also used by **AERO-S**, when executed with the “-t” (or “-T”) option, to name the ASCII output file (`FNAME.top`) containing the geometry of the performed computation in the **XPost** format. It is also used by **AERO-S** for outputting, when requested, a number of domain decomposition (or mesh partitioning) files (see [DECOMPOSE](#)).

Currently, this command is also needed for specifying the type of analysis to be performed by **AERO-S**. For this reason, it features the entry `ANATYPE` described below. For parsing reasons, `ANATYPE` must be specified whenever the `CONTROL` command is included in the input file, even this entry is not needed.



<code>FNAME</code>	Prefix of the names of the files reporting on the performance, geometry, and mesh decomposition of the performed computation, when applicable. The default value is the name of the AERO-S input file.
<code>ANATYPE</code>	Analysis type identifier (integer). It is needed for <i>any</i> thermal (heat conduction) analysis involving <i>radiation</i> , in which case it must be set to 2. Otherwise, this parameter should be set to 1.
<code>NODESET</code>	Name of the node set describing the grid points of the geometry of the computation in the file <code>FNAME.top</code> outputted in the XPost format, when AERO-S is executed with the “-t” (or “-T”) option. The default name is “nodes”.
<code>ELEMSET</code>	Name of the element set describing the connectivity of the geometry of the computation in the file <code>FNAME.top</code> outputted in the XPost format, when AERO-S is executed with the “-t” (or “-T”) option. The default name is “elems”.

Next: [DECOMPOSE](#), Previous: [CONTROL](#)

28 CONWEP

Command Statement:	CONWEP
--------------------	---------------

The `CONWEP` command statement is used to define, in conjunction with the [PRESSURE](#) command, a pressure load due to air blast in the free field using the software module [CONWEP](#). More specifically, the `CONWEP` command is used to specify the position of the charge, its mass, and the detonation time. The [PRESSURE](#) command is used to specify those elements of the finite element model on which to apply the generated pressure load.

The input format of this command is given below.

CONWEP

XCHARGE	YCHARGE	ZCHARGE	MCHARGE	DTIME
---------	---------	---------	---------	-------

XCHARGE x-ordinate of the charge (real).
 YCHARGE y-ordinate of the charge (real).
 ZCHARGE z-ordinate of the charge (real).
 MCHARGE Mass of the explosive charge (real).
 DTIME Time at which the charge is detonated (real).

Next: [DIMASS](#), Previous: [CONWEP](#)

29 DECOMPOSE

Command Statement:	DECOMPOSE
--------------------	------------------

The **DECOMPOSE** command statement can be used to perform an element-based mesh partitioning (or domain decomposition) of the computational domain, and exit or continue with a finite element analysis. The mesh partitioning algorithm is organized in two steps. The first step, which is based on the Greedy algorithm, generates an initial mesh partition. The second step optimizes the subdomain aspect ratios for the benefit of the FETI (see [STATICS](#)) iterative solution algorithms.

DECOMPOSE

NSUBS nsubs
DETER
FSGL
OUTFILE decomposition_filename
OUTLOAD
OUTMEM
EXIT
SKIP

NSUBS nsubs	Number of subdomains. Default is 1.
DETER	By default, the second step of the decomposition is carried out by a simulated annealing procedure. This sub-command keyword requests instead a deterministic approach to reduce the CPU time associated with the second step of the decomposition.

FSGL	Sub-command that is relevant only for vibro-acoustic (aka elastoacoustic and fluid-structure) analysis (for example, see IMPEDANCE). It requests performing the mesh decomposition in such a way that the fluid-structure interface is embedded within the subdomains as much as possible.
OUTFILE	
decomposition_filename	Requests outputting the decomposition in <code>decomposition_filename</code> . The format of this output is essentially the number and list of the elements in each subdomain. Default is <code>FNAME.optDec</code> where <code>FNAME</code> is the prefix specified under CONTROL (or its default value).
OUTLOAD	Requests outputting the load distribution in file <code>decomposition_filename.load</code> . The load distribution is based on the weights of the elements in the subdomains (see WEIGHTS and FWEIGHTS). Default value is <code>off</code> .
OUTMEM	This sub-command is relevant only when a FETI method is chosen as an equation solver (see STATICS). It requests outputting in file <code>decomposition_filename.mem</code> an estimate of the subdomain-based memory consumption. This estimate is based on the requirements for factoring the subdomain matrices by a direct skyline method. By default, mid-side nodes are ignored by the overall mesh partitioning algorithm. However, when this option is specified, mid-side nodes are taken into account. Default value is <code>off</code> .
EXIT	Requests exiting after the mesh partition is generated. Default value is <code>off</code> .
SKIP	Requests skipping this command. Default value is <code>off</code> .

When specified on the command line, the syntax of this command is as follows.

```
fem_executable --dec --nsub [number_of_subdomains] --deter --load --mem --exit <FEM_input_filename>
where --dec requests a mesh decomposition and --nsub specifies the number of subdomains. In this case, the prefix for the names of the files associated with the mesh decomposition and analysis is the content of the FNAME entry of the CONTROL command or its default value.
```

Note 1: When this command is executed from the command line and specified in the input file, any parameter specified on the command line overrides its value specified in the input file.

Note 2: When a decomposition file is already available, the syntax for running **AERO-S** with this decomposition file on a shared memory is

```
fem_executable -d [decomposition_filename] -v [verbose_frequency] -n [decomposition_filename]
<FEM_input_filename>
```

The above command can be combined with the decomposition command to obtain

```
fem_executable --dec --nsub [number_of_subdomains] --load --mem [decomposition_filename] -v
[verbose_frequency] -n [decomposition_filename] <FEM_input_filename>
```

Next: [DYNAMICS](#), Previous: [DECOMPOSE](#)

30 DISCRETE NODAL MASS AND INERTIA

Command Statement:	DIMASS
--------------------	---------------

The command **DIMASS** is used to lump a discrete mass or inertia on a specified degree of freedom attached to a specified node, either for a diagonal contribution to the mass matrix or an off-diagonal one. If a node number appears more than once under this command with the same degree of freedom (or pair of degrees of freedom), **AERO-S** sums all the lumped masses at the implied entries of the mass matrix. If a gravity field is also specified in the input file using the command [GRAVITY](#), **AERO-S** generates at each specified translational degree of freedom at each specified node under this command a weight-force component equal to the product of the specified discrete mass and specified gravity acceleration in the corresponding direction.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)), using the usual local numbering convention. By default, the nodal degree of freedom reference frames are the same as the global reference frame.

The input format is given below.

DIMASS

NODE#	DOF#	VALUE
-------	------	-------

or

NODE#	DOF1#	DOF2#	VALUE
-------	-------	-------	-------

NODE#	Node number where the mass or inertia will be added (integer).		
DOF#	Degree of freedom for which the mass or inertia will contribute a diagonal entry to the mass matrix (integer).		
VALUE	Value of the prescribed discrete mass or inertia (float).		
DOF1#	Degree of freedom for which the specified mass or inertia will contribute an off-diagonal entry (and its symmetric counterpart) to the mass matrix at the location implied by this degree of freedom and that specified in DOF2# (integer). Note that only the contribution to the lower triangular part of the mass matrix should be specified, and therefore DOF1# should be greater than DOF2#. Note also that for an explicit time-integration scheme or when the mass is requested to be LUMPED , this contribution will be part of the lumping process.		
DOF2#	Degree of freedom for which the mass or inertia will contribute an off-diagonal entry (and its symmetric counterpart) to the mass matrix at the location implied by this degree of freedom and that specified in DOF1# (integer). Note that only the contribution to the lower triangular part of the mass matrix should be specified, and therefore DOF2# should be smaller than DOF1#. Note also that for an explicit time-integration scheme or when the mass is requested to be LUMPED , this contribution will be part of the lumping process.		

If more than one line addressing the same degree of freedom of the same node are encountered under this command, the effect is the accumulation of the lumped masses or

inertias at this degree of freedom.

Next: [EIGEN](#), Previous: [DIMASS](#)

31 DYNAMIC ANALYSIS

Command Statement:	DYNAMICS
--------------------	-----------------

The **DYNAMICS** command is used to specify a time-integrator for a desired dynamic (transient, time-domain) structural, acoustic, or heat transfer (thermal) analysis and other relevant parameters. Currently, the following time-integrators are available in **AERO-S**:

1. The implicit generalized α method and the explicit central difference method for linear and nonlinear (in this case, the [NONLINEAR](#) command must also be present in the input file) structural dynamic analyses.
2. The implicit generalized α method, the explicit central difference method, and the explicit modified wave equation algorithm for linear, time-domain acoustic analysis.
3. The generalized midpoint family of methods for linear heat transfer analysis.
4. The implicit midpoint rule for nonlinear heat transfer analysis.

Linear and Nonlinear Structural Dynamic Analyses and Linear Time-Domain Acoustic Analyses

- The implicit generalized α method for linear and nonlinear structural dynamic and linear time-domain acoustic analyses enforces the following equation of equilibrium

$$M\ddot{u}^{n+1-\alpha_f} + D\dot{u}^{n+1-\alpha_f} + F_{internal}(u^{n+1-\alpha_f}) = F_{external}^{n+1-\alpha_f}$$

where

$$u^{n+1-\alpha_f} = (1 - \alpha_f)u^{n+1} + \alpha_f u^n$$

$$\dot{u}^{n+1-\alpha_f} = (1 - \alpha_f)\dot{u}^{n+1} + \alpha_f \dot{u}^n$$

$$\ddot{u}^{n+1-\alpha_m} = (1 - \alpha_m)\ddot{u}^{n+1} + \alpha_m \ddot{u}^n$$

$$F_{external}^{n+1-\alpha_f} = F_{external}(t^{n+1-\alpha_f})$$

$$t^{n+1-\alpha_f} = (1 - \alpha_f)t^{n+1} + \alpha_f t^n$$

and for linear problems

$$F_{internal}(u^{n+1-\alpha_f}) = Ku^{n+1-\alpha_f}$$

Hence, the above equation of equilibrium can also be written as

$$M((1 - \alpha_m)\ddot{u}^{n+1} + \alpha_m \ddot{u}^n) + D((1 - \alpha_f)\dot{u}^{n+1} + \alpha_f \dot{u}^n) + F_{internal}((1 - \alpha_f)u^{n+1} + \alpha_f u^n) = F_{external}^{n+1-\alpha_f}$$

Given $F_{external}(t)$, u^0 and \dot{u}^0 , the implicit generalized α method solves the above equation as follows

$$\ddot{u}^0 = M^{-1}(F_{external}^0 - F_{internal}(u^0) - D\dot{u}^0)$$

$$u^{n+1} = u^n + \Delta t \dot{u}^n + \Delta t^2 \left(\left(\frac{1}{2} - \beta \right) \ddot{u}^n + \beta \ddot{u}^{n+1} \right)$$

$$\dot{u}^{n+1} = \dot{u}^n + \Delta t \left(\left(1 - \gamma \right) \ddot{u}^n + \gamma \ddot{u}^{n+1} \right)$$

where $\beta \neq 0$. The Newmark method is obtained by setting $\alpha_m = \alpha_f = 0$. The midpoint rule is obtained by setting $\alpha_m = \alpha_f = \frac{1}{2}$, $\beta = \frac{1}{4}$ and $\gamma = \frac{1}{2}$. The *classical* central difference method is obtained by setting $\alpha_m = \alpha_f = 0$, $\beta = 0$ and $\gamma = \frac{1}{2}$.

The Newmark method ($\alpha_m = \alpha_f = 0$) is second-order time-accurate if and only if $\gamma = \frac{1}{2}$. For undamped linear systems, it is unconditionally stable if and only if $\gamma \geq \frac{1}{2}$ and $\beta \geq \frac{1}{4}(\gamma + \frac{1}{2})^2$.

The generalized α method is second-order time-accurate if and only if $\gamma = \frac{1}{2} - \alpha_m + \alpha_f$. For undamped systems, it is unconditionally stable for linear problems if and only if $\alpha_m \leq \alpha_f \leq \frac{1}{2}$ and $\beta \geq \frac{1}{4} + \frac{1}{2}(\alpha_f - \alpha_m)$. It maximizes high-frequency dissipation when

$$\beta = \frac{1}{4}(1 - \alpha_m + \alpha_f)^2, \quad \alpha_f = \frac{\rho_\infty}{\rho_\infty + 1}, \quad \alpha_m = \frac{2\rho_\infty - 1}{\rho_\infty + 1}$$

where ρ_∞ is the user-specified high-frequency dissipation ($0 \leq \rho_\infty \leq 1$).

- The central difference method for linear and nonlinear structural dynamic and time-domain acoustic analyses implemented in **AERO-S** enforces the following equation of equilibrium

$$M\ddot{u}^{n+1} + D\dot{u}^{n+\frac{1}{2}} + F_{internal}(u^{n+1}) = F_{external}^{n+1}$$

where

$$\dot{u}^{n+\frac{1}{2}} = \dot{u}^n + \frac{\Delta t}{2} \ddot{u}^n$$

Given $F_{external}(t)$, u^0 and \dot{u}^0 , it solves the above equation as follows

$$\begin{aligned} \ddot{u}^0 &= M^{-1}(F_{external}^0 - F_{internal}(u^0) - D\dot{u}^0) \\ u^{n+1} &= u^n + \Delta t \dot{u}^n + \frac{\Delta t^2}{2} \ddot{u}^n \\ \dot{u}^{n+1} &= \dot{u}^n + \frac{\Delta t}{2} (\ddot{u}^n + \ddot{u}^{n+1}) \end{aligned}$$

This central difference method for linear and nonlinear structural dynamic and linear time-domain acoustic analyses computes the acceleration as follows

$$\ddot{u}^{n+1} = M^{-1} \left(F_{external}^{n+1} - F_{internal}(u^{n+1}) - D(\dot{u}^n + \frac{\Delta t}{2} \ddot{u}^n) \right)$$

For this reason, **AERO-S** automatically lumps the mass matrix M when this method is selected so that it becomes explicit.

- The explicit modified wave equation method for linear time-domain acoustic analysis implemented in **AERO-S** enforces the following equation of equilibrium

$$M\ddot{u}^{n+1} + D\dot{u}^{n+1} + \left(K - \frac{\Delta t^2}{q} KM^{-1} K\right) u^{n+1} = F^{n+1}$$

where D is the damping matrix associated with a non-absorbing boundary condition when such a condition is imposed, and q is a real-valued parameter.

Given $F(t)$, u^0 and \dot{u}^0 , it solves the above equation as follows

$$\begin{aligned}\ddot{u}^0 &= M^{-1} \left(F^0 - D\dot{u}^0 - K(u^0 - \frac{\Delta t^2}{q} M^{-1} K u^0) \right) \\ \dot{u}^{\frac{1}{2}} &= \dot{u}^0 + \frac{\Delta t}{2} \ddot{u}^0 \\ u^{n+1} &= u^n + \Delta t \dot{u}^{n+\frac{1}{2}} \\ G^{n+1} &= K \left(I - \frac{\Delta t^2}{q} (M + \frac{\Delta t}{2} D)^{-1} K \right) u^{n+1} \\ \dot{u}^{n+\frac{3}{2}} &= (M + \frac{\Delta t}{2} D)^{-1} \left(M\dot{u}^{n+\frac{1}{2}} + \frac{\Delta t}{2} (2F^{n+1} - 2G^{n+1} - D\dot{u}^{n+\frac{1}{2}}) \right)\end{aligned}$$

For $D = 0$ and $q = 12$, the explicit modified wave equation method is fourth-order time-accurate. For $D = 0$ and $12 \leq q \leq 12.5$, experience reveals that for many applications, this method delivers an almost sixth-order time-accuracy. For $D \neq 0$, the explicit modified wave equation method is second-order time-accurate.

Linear Heat Transfer Analysis

- The generalized midpoint family of methods for linear heat transfer analysis enforces the equation of equilibrium

$$Q\dot{u}^{n+\frac{1}{2}} + Hu^{n+\frac{1}{2}} = F^{n+\frac{1}{2}}$$

where

$$u^{n+\frac{1}{2}} = \frac{1}{2}(u^{n+1} + u^n)$$

$$\dot{u}^{n+\frac{1}{2}} = \frac{1}{2}(\dot{u}^{n+1} + \dot{u}^n)$$

and solves it as follows

$$\dot{u}^0 = Q^{-1}(F^0 - Hu^0)$$

$$\dot{u}^{n+\alpha} = \alpha\dot{u}^{n+1} + (1 - \alpha)\dot{u}^n$$

$$u^{n+1} = u^n + \Delta t (\alpha\dot{u}^{n+1} + (1 - \alpha)\dot{u}^n)$$

The generalized midpoint family of methods outlined above is unconditionally unstable for

$\alpha > \frac{1}{2}$. Hence, only the choices satisfying $\alpha \leq \frac{1}{2}$ are allowed. The forward Euler method is obtained for $\alpha = 0$. The backward Euler implicit method is obtained for $\alpha = 1$. The implicit trapezoidal rule is obtained for $\alpha = \frac{1}{2}$.

Nonlinear Heat Transfer Analysis

- The implicit midpoint rule for nonlinear heat transfer analysis enforces the following equation of equilibrium

$$Q\dot{u}^{n+\frac{1}{2}} + F_{internal}(u^{n+\frac{1}{2}}) = F_{external}^{n+\frac{1}{2}}$$

where

$$u^{n+\frac{1}{2}} = \frac{1}{2}(u^{n+1} + u^n)$$

$$\dot{u}^{n+\frac{1}{2}} = \frac{1}{2}(\dot{u}^{n+1} + \dot{u}^n)$$

Given $F_{external}(t)$ and u^0 , it solves the above equation as follows

$$\dot{u}^0 = Q^{-1} (F_{external}^0 - F_{internal}(u^0))$$

$$u^{n+1} = u^n + \frac{\Delta t}{2}(\dot{u}^{n+1} + \dot{u}^n)$$

Structural Damping

Note that for the purpose of structural dynamic analysis, structural damping can be represented in **AERO-S** using two different approaches:

- The Rayleigh proportional damping, in which case the finite element damping matrix D takes the form

$$D = aK + bM$$

where a and b are two real scalars that may be specified in this **DYNAMICS** command for the entire structure, and/or in the **MATERIAL** command at the material and therefore element level (see the sub-command keyword **DAMPING_TYPE** in **MATERIAL**).

- Modal damping ratios that can be specified in this command **DYNAMICS**. However, as its name suggests, this approach is applicable only to modal dynamic analysis.

Note 1: Explicit time-integration algorithms are currently not supported for finite element models with rigid elements, massless (or moment-of-inertia-less) degrees of freedom, or linear multipoint constraints.

Note 2: Whenever an implicit time-integration algorithm is selected, an equation solver must be specified under the **STATICS** command.

Note 3: For nonlinear heat transfer analysis (see **NONLINEAR**), the appropriate midpoint rule

algorithm is automatically selected by this command.

The input format of this command is given below.

DYNAMICS

IACC	flagIACC				
STABLE	OPTION COEFF TOL MAXITR FREQ				
MODAL	(optional)				
MECH	β	γ			(midpoint implementation of Newmark — that is, $\alpha_m = \alpha_f = \frac{1}{2}$)
MECH	β	γ	α_f	α_m	(generalized α)
MECH	ρ_∞				(generalized α)
ACOU	β	γ			(midpoint implementation of Newmark — that is, $\alpha_m = \alpha_f = \frac{1}{2}$)
ACOU	β	γ	α_f	α_m	(generalized α)
ACOU	ρ_∞				(generalized α)
ACOU	β	γ	q		(modified wave equation method)
HEAT	α				
TIME	TH	TM	TT		
RAYDAMP	a	b			
MODDAMP					
MODE#	MDV				
.					
.					
MODE#	MDV				

IACC For linear and nonlinear explicit dynamic computations, and for nonlinear implicit dynamic computations, the initial acceleration is always computed to satisfy equilibrium at $t = 0$. For linear implicit dynamic computations, the initial acceleration is always set to zero if a FETI solver is specified under the [STATICS](#) command; otherwise, the user can request to have it either set to zero or computed to satisfy equilibrium at $t = 0$. `iacc` is the sub-command keyword that enables the user to make this choice in the latter case (characters).

flagIACC On/Off flag (characters). The default value is On.

on In this case, for linear implicit dynamic computations, as long as a FETI method is not chosen as the equation solver under the [STATICS](#) command, the acceleration is initialized to satisfy equilibrium at $t = 0$. This initialization requires the

factorization of the mass matrix. The sparse direct solver is always used for this purpose, no matter which direct equation solver is specified under the [STATICS](#) command. If for some reason the finite element model has one or several massless or moment-of-inertia-less degrees of freedom, the sparse solver will typically encounter zero pivots and set the initial acceleration to zero at the corresponding degrees of freedom.

off	In this case, for linear implicit dynamic computations, the initial acceleration is set to zero.
STABLE	Sub-command keyword for managing the time-step of a linear or nonlinear, structural dynamic or acoustic time-integration using the central difference method (characters). Currently, option 1 and option 2 (see below) are automatically disabled for fluid-structure computations using the tandem AERO-F/AERO-S .
OPTION	This integer parameter specifies how to manage the computational time-step for an explicit simulation.
0	In this case, the time-step is not managed and the computations are performed using the specified computational time-step TM (see below).
1	In this case, which is also the default case, AERO-S estimates automatically the critical time-step — that is, the maximum stability time-step — and multiplies it by the specified coefficient COEFF (see below). In the case of the linear explicit central difference method, AERO-S estimates the critical time-step by computing the largest eigenvalue of the finite element model using the power method. In the case of the nonlinear explicit central difference method, AERO-S estimates the critical time-step by an element-dependent method and updates this estimate every FREQ time-steps. If the specified computational time-step TM (see below) is larger than the product of the critical time-step and COEFF, this product is used instead for the computation.
2	In this case, AERO-S computes automatically the critical time-step as described above for OPTION = 1. However in this case, the explicit computations are performed using a computational time-step equal to the product of the estimated critical time-step and COEFF (see below), regardless of the value of the specified computational time-step TM (see below).
COEFF	Real-valued coefficient for managing the computational time-step (see above). The default value is 0.8 .
TOL	Error tolerance for computing the highest eigenvalue of the finite element model using the power method (float). The default value is 10⁻³ .
MAXITR	Maximum number of iterations for computing the highest eigenvalue of the finite element model using the power method (integer). The default value is 100 .
FREQ	Integer parameter specifying the frequency (every so many time-steps) at which the critical time-step is re-estimated during nonlinear explicit computations (see above). The default value is 1000 .
MODAL	Sub-command keyword for requesting a modal superposition dynamic analysis (characters). In this case, the READMODE command must be present in the input file to provide the eigenmodes to be used (characters). As in the case of a direct time-integration rather than modal analysis, the initial displacement (see IDISPLACEMENTS) and velocity (see IVELOCITIES) conditions can be input in either the finite element or generalized coordinates (or reduced system), or in both.
MECH OR ACOU OR HEAT	The sub-command keyword MECH signals a structural dynamic analysis (characters). The sub-command keyword ACOU signals a time-domain acoustic analysis (characters). The sub-command keyword HEAT signals a thermal analysis (characters).
β	Newmark coefficient β for a second-order system (real). Set $\beta = 0$ ($\alpha_m = \alpha_f = 0$)

and $\gamma = \frac{1}{2}$ to specify the explicit central difference method.

γ Newmark coefficient γ for a second-order system (real).

α_f Generalized α coefficient α_f for a second-order system (real). To obtain the standard Newmark algorithm, set $\alpha_f = 0$. The default is $\alpha_f = \frac{1}{2}$.

α_m Generalized α coefficient α_m for a second-order system (real). To obtain the standard Newmark algorithm, set $\alpha_m = 0$. The default is $\alpha_m = \frac{1}{2}$.

ρ_∞ Generalized α infinite frequency spectral radius for a second-order system (real). When this option is used and this coefficient is specified, α_f , α_m , β and γ are automatically set to

$$\alpha_f = \frac{\rho_\infty}{\rho_\infty + 1}, \quad \alpha_m = \frac{2\rho_\infty - 1}{\rho_\infty + 1}, \quad \beta = \frac{1}{4}(1 - \alpha_m + \alpha_f)^2, \quad \gamma = \frac{1}{2} - \alpha_m + \alpha_f$$

, which results in a second-order unconditionally stable scheme that minimizes low frequency dissipation.

q For $q = 12$, the explicit modified wave equation method is fourth-order time-accurate. Furthermore, for many applications, experience shows that for $12 \leq q \leq 12.5$, this method behaves almost like a sixth-order time-accurate scheme.

HEAT Sub-command keyword that precedes the parameter of the Newmark algorithm for a first-order time-dependent system (characters).

α Main parameter of the generalized trapezoidal family of methods for first-order systems (real). This parameter must be chosen within $\alpha \leq \frac{1}{2}$.

TIME Sub-command keyword for specifying time-interval parameters (characters).

TH Integration time-step for heat transfer analysis (real). This time-step should be put to zero if a structural dynamic or time-domain acoustic analysis is to be performed.

TM Integration time-step for a structural dynamic or time-domain acoustic analysis (real).

TT End of time-interval for time-integration (real); beginning of time-interval for time-integration is zero for all types of time-dependent analyses.

RAYDAMP Sub-command keyword for specifying Rayleigh proportional damping coefficients for the entire structure (characters). This option is active only when **MECH** is specified.

a Rayleigh damping stiffness coefficient a (real). This option is active only when **MECH** is specified. In the nonlinear case, this coefficient is assigned at each iteration to the *initial* linearized stiffness matrix and the product of these two quantities, which therefore remains constant throughout the nonlinear iterations, constitutes the contribution to the damping term.

b Rayleigh damping mass coefficient b (real). This option is active only when **MECH** is specified.

MODDAMP Sub-command keywords for signaling that the following lines specify modal damping ratios (characters). This option is active only when the **MECH** and **MODAL** sub-commands are also specified — that is, for a dynamic analysis of a mechanical system using mode superposition. If a mode is repeated, the modal damping ratio values are added for that mode. If both of the **RAYDAMP** and **MODDAMP** commands are specified, and the keyword **MODAL** is also specified, then modal damping takes precedence over Rayleigh damping.

MODE# Mode id number (integer).
 MDV Modal damping value (ξ) for mode MODE# (real).

Next: [END](#), Previous: [DYNAMICS](#)

32 EIGENVALUE PROBLEMS

Command Statement: **EIGEN**

The EIGEN command statement is used to request the construction and solution of a generalized eigenvalue problem of the form

$$AX = BX\Omega^2$$

where A and B are two finite element square matrices, X is the rectangular matrix of generalized eigenvectors, and Ω^2 is the diagonal matrix of corresponding eigenvalues. Currently, only the subspace iteration method (the default choice) and the ARPACK package are available in **AERO-S** for this purpose.

The input format of this command is given below. An example input file can be found in FEM.d/fem_examples/Eigen.d.

Note 1: The usage of the EIGEN command requires the simultaneous usage of the [STATICS](#) command to specify an equation solver. This solver must be chosen according to the properties of A and B (most importantly, watch out for the case where A — or in the shifted case $A\sigma B$ — is singular).

Note 2: When A — or in the shifted case $A\sigma B$ — is singular, and the equation solver specified in [STATICS](#) is [sparse](#), [skyline](#), [mumps](#), [pivot](#), or [FETI DP](#), the null space computed by this equation solver is exploited by the eigen solver described herein in the solution of the above generalized eigenvalue problem.

EIGEN

VERSION			
SHIFT	σ		
NSBSPV	nsbspv		
NEIGPA	neigpa		
TOLEIG	toleig		
TOLJAC	toljac		
ARPACK	which	mode	
ARPACK	lbound	nshifts	
ARPACK	lbound	ubound	neigps
MAXITR	maxitr		

VERSION

explicit	In this case, the A matrix is used explicitly when constructing the reduced generalized eigen problem in the subspace iteration method. Otherwise (default), the reduced generalized eigen problem is constructed by exploiting some mathematical identities to avoid using explicitly the A matrix which has been factored by then. Note that specifying the explicit version of the subspace iteration algorithm incurs additional memory storage.
SHIFT σ	Value of a specified shift for the A matrix (real).
NSBSPV	
nsbspv	In the case of the subspace iteration method (default eigensolver), nsbspv denotes the number of subspace iteration vectors and should be set to the minimum between twice the number of requested eigen pairs, $2 \times$ neigpa, and $\text{neigpa} + 8$. If rigid body modes are expected, the previous formula should be increased by the total number of rigid body modes. In the case of ARPACK, nsbspv denotes the number of Krylov vectors (per shift, if a shifting strategy is used).
NEIGPA	
neigpa	Number of requested eigen pairs. If rigid body modes are found, their number is included in the specified value of neigpa.
TOLEIG	
toleig	Tolerance for the convergence of the subspace iteration method.
TOLJAC	
toljac	Tolerance for the Jacobi Determinant algorithm used in the subspace iteration method.
ARPACK	The presence of this keyword under the EIGEN command specifies the usage of the ARPACK eigensolver in the specified mode (see mode below). Otherwise, the default choice is the subspace iteration method. The ARPACK choice is required whenever one of the two matrices A or B is indefinite, or one or both of them are singular. Indefinite systems arise: (1) if the analysis involves the HELMHOLTZ, IMPEDANCE, or EIGEN command with a positive shift (see SHIFT in EIGEN) or for buckling analysis (see ARPACK in EIGEN), or (2) the structural model includes rigid and/or joint elements (see TOPOLOGY), linear multi-point constraints (see LMPC), or tied surfaces (see TIEDSURFACES), and the Lagrange multiplier method is chosen for enforcing the associated constraints (see CONSTRAINTS). In either case, the equation solver must be properly chosen in STATICS.
which	Keyword (characters) to specify which eigenpairs to be computed by ARPACK. This keyword can take one of the following values.
LA	In this case, the neigpa eigenpairs whose eigenvalues are just to the right of the shift σ are computed. This is the default value of which when the shift is zero ($\sigma = 0$).
SA	In this case, the neigpa eigenpairs whose eigenvalues are just to the left of the shift σ are computed.
BE	In this case, neigpa eigenpairs with eigenvalues on either side of the shift σ are computed. This is the default value of which when the shift is non zero ($\sigma \neq 0$).
mode	Integer identifier to specify the mode in which to run ARPACK. It can take one of the following values.
3	This default mode, which is the shift-invert mode, is recommended for all generalized eigenvalue problems except those for which the matrix B is

	indefinite. Hence, this mode is particularly not recommended for buckling analysis.
4	This mode is recommended for generalized eigenvalue problems where the matrix B is indefinite. Hence, this mode is particularly recommended for buckling analysis. In this case, a nonzero shift should be specified in SHIFT (see above).
lbound	Lower bound of a set or range of eigenvalues to be computed by ARPACK (real).
nshifts	Number of shifts to be used by ARPACK when computing the <code>neigpa</code> eigenvalues that are greater than <code>lbound</code> (integer). The specific values of the shifts are automatically selected by ARPACK, and <code>neigpa/nshifts</code> eigenpairs are computed per shift.
ubound	Upper bound of a range of eigenvalues to be computed by ARPACK (real).
neigps	When a range of eigenvalues [<code>lbound</code> , <code>ubound</code>] is specified, ARPACK computes all eigenpairs whose eigenvalues lie within this range. In this case, a first shift is set to <code>lbound</code> and <code>neigps</code> consecutive eigenvalues within the range [<code>lbound</code> , <code>ubound</code>] are computed. Then, a recursive procedure in which the largest previously computed eigenvalue is chosen as a new shift and <code>neigps</code> new eigenvalues within the range [<code>lbound</code> , <code>ubound</code>] are computed using this shift is applied until all eigenvalues within the specified range are captured. The default value for <code>neigps</code> is 50.
MAXITR	Maximum number of iterations for the eigensolver. The default value is <code>10 × nspbv.</code>
maxitr	Maximum number of iterations for the eigensolver. The default value is <code>10 × nspbv.</code>

Next: [FSINTERFACE](#), Previous: [EIGEN](#)

33 END

The `END` statement is used to indicate the end-of-file. It should always be the last statement.

Next: [FWEIGHTS](#), Previous: [END](#)

34 FLUID/STRUCTURE INTERFACE

Command Statement:	FSINTERFACE
--------------------	--------------------

The `FSINTERFACE` command statement can be used to define the fluid/structure interface in a coupled elastoacoustic frequency response problem where: (1) the computational acoustic fluid and structural domains are discretized using either a single mesh, or two different meshes with matching or non-matching discrete interfaces, and (2) in either case, each of these two computational domains has its own representation of the fluid/structure interface in the form of a discrete surface defined using [SURFACETOPO](#). In this scenario, the command statement `FSINTERFACE` is used to define more specifically the fluid/structure interface as the pairing of these two surfaces.

The input format of this command is given below.

Note 1: In this case, the capabilities of the ACME library are used to generate the fluid-structure coupling coefficients.

FSINTERFACE

SURF_PAIR_ID#	FLUID_SURF	STR_SURF
---------------	------------	----------

or

SURF_PAIR_ID#	FLUID_SURF	STR_SURF	NORMAL_TOL	TANGENTIAL_TOL
---------------	------------	----------	------------	----------------

SURF_PAIR_ID#	Id number of the surface pair to be described (integer).
FLUID_SURF	Identification of the fluid surface (see SURFACETOPO) (integer).
STR_SURF	Identification of the structure surface (see SURFACETOPO) (integer).
NORMAL_TOL	Normal search tolerance used by ACME to identify interactions, default value is 0.1 (float) (see Figs. 1.2 and 1.3 in Section 1.3 of ACME's User Reference Manual).
TANGENTIAL_TOL	Tangential search tolerance used by ACME to identify interactions, default value is 0.001 (float) (see Figs. 1.2 and 1.3 in Section 1.3 of ACME's User Reference Manual).

Next: [MFTT](#), Previous: [FSINTERFACE](#)

35 FIELD WEIGHTS FOR MESH DECOMPOSITION

Command Statement:	FWEIGHTS
--------------------	-----------------

In order to achieve load balancing when generating the subdomains, the mesh partitioning algorithm (see [DECOMPOSE](#)) can account for different weights (or weighting coefficients) attributed to the elements of the given mesh according to their type (see [TOPOLOGY](#)). The default values attributed by **AERO-S** to these elements types are based on their relative computational complexity. For each different field — that is, Mechanic, Heat, Coupled Thermoelastic, Fluid, or Acoustic — these weights range between 1 and some higher value that is field-dependent. The command [WEIGHTS](#) can be used to reset, if desired, some or all of these weights to some user-specified values.

For a multidisciplinary simulation such as fluid-structure using **AERO-S** alone, the command [FWEIGHTS](#) can be used to attribute to each involved field (for example, Mechanic and Acoustic) a weight to allow [DECOMPOSE](#) to account additionally for the relative computational complexity of a typical element of this field to those of the typical elements of the other fields involved in the simulation. In this case, the weight w_e of each element type (whether its default value or that set using [WEIGHTS](#)) is automatically reset by **AERO-S** to the ratio

$$\frac{w_e \times w_{field_of_e}}{\sum_{involvedfields} w_{field}}$$

where $w_{field_of_e}$ is the weight of the field of element type e , the scope of *involvedfields* is defined by all fields /bin/bash: jkkkkk: command not found specified using this command.

The input format of this command is given below.

FWEIGHTS
ACOU acou_weight
FLUI flui_weight
MECH mech_weight
HEAT heat_weight

ACOU	Sub-command keyword for specifying a weight to the field Acoustic (see TOPOLOGY).
acou_weight	Weight to be attributed to the field Acoustic (real).
FLUI	Sub-command keyword for specifying a weight to the field Fluid (see TOPOLOGY).
flui_weight	Weight to be attributed to the field Fluid (real).
MECH	Sub-command keyword for specifying a weight to the field Mechanic (see TOPOLOGY).
mech_weight	Weight to be attributed to the field Mechanic (real).
HEAT	Sub-command keyword for specifying a weight to the field Heat (see TOPOLOGY).
heat_weight	Weight to be attributed to the field Heat (real).

Next: [GRBM](#), Previous: [FWEIGHTS](#)

36 FORCE TIME TABLE-MECHANICS AND ACOUSTICS

Command Statement:	MFTT [TABLE_ID]
--------------------	------------------------

The **MFTT** command statement can be used to implement time-dependent tensor-product forms of the boundary conditions enforced by the commands **FORCE**, **ATDDNB**, **ATDROB**, and **PRESSURE**. Pairs of time and amplification values are input. Linear interpolation is also used for "in between" points.

When applied with the **FORCE** command, the result is the amplification of the force value. When applied with the **ATDDNB** command, the result is the amplification of the resulting distributed Neumann boundary condition. When applied with the **ATDROB** command, the result is the amplification of the right hand-side of the distributed Robin boundary condition.

MFTT	[TABLE_ID]
------	------------

TIME_1	AMP_1
.	.
.	.

.	.
TIME_n	AMP_n

TABLE_ID

Optional non-negative integer which uniquely identifies a force-time table so that it can be associated with a "load" set to define the "load" case for a dynamic analysis using the [LOADCASE](#) command. The default value is 0. Hence, the [MFTT](#) command can be repeated as many times as desired within the same input file using each time a different value for TABLE_ID and different data.

TIME_1

A specified time point (float).

AMP_1

A specified amplification value at time point TIME_1 (float). This amplification factor is automatically set to zero for all times prior to the earliest specified time point and all times later than the latest specified time point.

Next: [GEPS](#), Previous: [MFTT](#)

37 GEOMETRIC RIGID BODY MODES

Command Statement:	GRBM
--------------------	-------------

The GRBM command is useful for a number of analyses of unrestrained or partially restrained finite element models. It has two separate effects:

- 1. The GRBM command requests the computation of the rigid body modes (or zero energy modes in general) of a given (unrestrained, partially restrained, or even restrained) finite element model using the hybrid geometric-algebraic method published in "C. Farhat and M. Geradin, *On the General Solution by a Direct Method of a Large-Scale Singular System of Linear Equations: Application to the Analysis of Floating Structures*, International Journal for Numerical Methods in Engineering, Vol. 41, pp. 675-696 (1998)". This hybrid method combines a geometry-based algorithm and the SVD factorization of a matrix of constraints corresponding to the given boundary conditions. The SVD factorization relies on the first tolerance specified under this command for identifying the deemed zero singular values, which are associated with the sought-after singular modes. Whenever AERO-S finds the GRBM in its input file, it computes the rigid body modes (or zero energy modes in general) of the given finite element model and prints on the screen the found number of rigid body or zero energy modes. Then, if the equation solver specified in [STATICS](#) is the sparse or skyline direct solver, these rigid body or zero energy modes are exploited by the [RBMFILTER](#) and [EIGEN](#) commands when these are also invoked in the input file. Otherwise, this aspect of the GRBM command can be used only to verify that the rigid body mode information (their number and the modes themselves) computed by the [TRBM](#) method, which is supported by all equation solvers of the [STATICS](#) command that can handle singular matrices, is correct.

- 2. When the requested analysis incurs the solution of a singular system of finite element equations, and the equation solver chosen under the [STATICS](#) command is sparse or skyline, the GRBM command requests the computation of the generalized inverse of the representative singular finite element matrix in factored form using the algorithm published also in "C. Farhat and M. Geradin, *On the General Solution by a Direct Method of a Large-Scale Singular System of Linear Equations: Application to the Analysis of Floating Structures*, International Journal for Numerical Methods in Engineering, Vol. 41, pp. 675-696 (1998)". This algorithm relies on the

information computed by the hybrid geometric-algebraic method outlined above (number of rigid body or zero energy modes of the finite element model, and in some cases, the rigid body or zero energy modes themselves).

Singular finite element systems may arise in linear or nonlinear static problems, and in eigenvalue problems. In the former case, the tangent finite element matrices associated with unrestrained or partially restrained nonlinear finite element models can have a number of rigid body or zero energy modes that is different from that which can be deduced from the analysis of the matrix of constraints associated with the specified boundary conditions. For this reason, the second effect of the **GRBM** command takes effect only for **linear** static, quasistatic, eigen (structural), aeroelastic, and thermoelastic analyses, and when the **sparse** or **skyline** equation solver is selected under the **STATICS** command. In that case, the generalized inverse of the stiffness matrix is computed in factored form using the aforementioned hybrid geometric-algebraic algorithm, and its pivots are monitored for small values (deemed to be zero) using the second tolerance specified under the **GRBM** command — such deemed zero pivots are associated with zero energy modes corresponding to mechanisms or model oddities that cannot be detected by the hybrid geometric-algebraic method.

If a FETI-1 or FETI-2 solver is specified under the **STATICS** command, the rigid body modes of the floating subdomains are computed using the same hybrid geometric-algebraic method and their singular matrices are factored using the same aforementioned algorithm (see section on FETI parameters in **STATICS**). If a FETI-DP solver is specified under the **STATICS** command, the tolerance for detecting and computing the rigid body modes of the global structure is set to the first tolerance specified under the **GRBM** command.

Note 1: See [TRBM](#) for an alternative option for analyzing singular systems.

Note 2: If both the **GRBM** and **TRBM** commands are specified in the same **AERO-S** input file, the one specified last is chosen. However, if that one is **TRBM**, it does not de-activate the **GRBM** command but simply overwrites the value of its second tolerance, **VALUE_2**, by the value of the tolerance specified under **TRBM**.

GRBM

VALUE_1	VALUE_2
----------------	----------------

VALUE_1	Tolerance for identifying the zero singular values of the constraint matrix analyzed by the hybrid geometric-algebraic method for finding the rigid body or zero energy modes of a finite element model (real). Its default value is 1.0e-6.
VALUE_2	Tolerance for capturing algebraically the mechanisms and other oddities that cause a finite element matrix to be singular and which are not visible by the geometric-algebraic method for finding the rigid body or zero energy modes of a finite element model (real). Its default value is 1.0e-16.

Next: [GRAVITY](#) Previous: [GRBM](#)

38 GEOMETRIC STIFFENING DUE TO PRESTRESS

Command Statement:	GEPS
--------------------	-------------

The **GEPS** command statement is essentially a switch that turns on the accounting of pre-stress effects in the form of a geometric stiffness matrix K_G . This geometric stiffness matrix is computed around

a displacement field specified under the **IDISP6** command which must be present in the input file. A sample input file illustrating the use of **GEPS** can be found in FEM.d/fem\examples/Buckle.d/

The effect of this switch on the **EIGEN** command statement is to replace the stiffness matrix by the tangent stiffness matrix and therefore compute the modes of the structure around the deformed position implied by the displacement field specified under the **IDISP6** command.

Its effect on the **DYNAMICS** and **STATICS** commands is to replace the stiffness matrix by the tangent stiffness matrix around the displacement field specified under the **IDISP6** command, including in aeroelastic applications.

Note 1: The **GEPS** command can be used only in a linear analysis.

Note 2: If an aeroelastic analysis is requested with the **GEPS** and **IDISP6** commands present in the input file, and the **IDISPLACEMENTS** command is specified to initialize the displacement field, then **AERO-S** sends to the fluid code at each time step the sum of the updated displacement field (interpreted in that case as an increment displacement) and the displacement specified under the **IDISP6** command. If in such a case the **IDISPLACEMENTS** command is not present in the input file, the displacement field of the structure is then initialized by the **IDISP6** command, and **AERO-S** acts as follows: at the first time-step, it sends to the fluid code only the updated displacement field (since initialization accounts in that case for the content of the **IDISP6** command), but at each subsequent time-step, it sends the sum of the updated displacement field and the displacement specified under the **IDISP6** command.

GEPS

Next: [GROUPS](#), Previous: [GEPS](#)

39 GRAVITATIONAL ACCELERATION

Command Statement:	GRAVITY
--------------------	----------------

The **GRAVITY** command is used to specify directional gravitational acceleration constants. The input format is given below.

Note 1: By default, **AERO-S** computes all element gravity loads by a consistent approach, **except for explicit dynamic computations** (see [DYNAMICS](#)), in which case **AERO-S** always uses a lumped approach for this purpose. If a consistent mass matrix is not available for a particular element (see [TOPOLOGY](#)), then **AERO-S** uses in all cases a lumped gravity load for that element. Alternatively, the [LUMPED](#) command can be used to instruct **AERO-S** to compute all element mass matrices and gravity loads by a lumped approach.

GRAVITY

COEFF_X	COEFF_Y	COEFF_Z
---------	---------	---------

COEFF_X	Gravitational Acceleration Constant in the X direction.
COEFF_Y	Gravitational Acceleration Constant in the Y direction.
COEFF_Z	Gravitational Acceleration Constant in the Z direction.

Next: [HZEM](#), Previous: [GRAVITY](#)

40 GROUPS

Command Statement:	GROUPS
--------------------	---------------

The GROUPS command can be used to organize elements into groups according to their attribute number (see [ATTRIBUTES](#)), and nodes into groups according to the integer number identifying them or the surface to which they belong (see [SURFACETOPO](#)). For example, the notion of a group of elements is used by the [RANDOM](#) command to attribute a defined random material property to a group of elements. That of a group of nodes is used by the [OUTPUT](#) command to output computed results for groups of nodes.

Note 1: Group identifiers must be contiguous integers starting from 1.

GROUPS

A	ATTRIBUTE#	GROUP#
---	------------	--------

or

A	STARTING_ATTRIBUTE#	ENDING_ATTRIBUTE#	GROUP#
---	---------------------	-------------------	--------

or

N	NODE#	GROUP#
---	-------	--------

or

N	STARTING_NODE#	ENDING_NODE#	GROUP#
---	----------------	--------------	--------

or

N SURFACE	SURFACE#	GROUP#
-----------	----------	--------

A	Sub-command keyword (character) signaling that elements are to be grouped according to their attribute number — that is, all elements whose attribute is the same as ATTRIBUTE# or within the range [STARTING_ATTRIBUTE#, ENDING_ATTRIBUTE#] are to be put in the same group GROUP#.
ATTRIBUTE#	Attribute ID number of the elements to be placed in the same group GROUP# (integer).
STARTING_ATTRIBUTE#	First attribute ID number of a sequence of attribute ID numbers that define the group GROUP# (integer).
ENDING_ATTRIBUTE#	Last attribute ID number of a sequence of attribute ID numbers that define the group GROUP# (integer).
N	Sub-command keyword (character) signaling that nodes are to be grouped according to their node identifying number — that is, the node whose identifying number is NODE# or all nodes whose identifying numbers are within the range [STARTING_NODE#, ENDING_NODE#] are to be put in the same group GROUP#.
N SURFACE	Sub-command keywords (characters) signaling that nodes are to be grouped according to the surface to which they belong, and therefore according to the integer number identifying that surface (see SURFACETOPO).
NODE#	ID number of the node to be placed in the group GROUP# (integer).
STARTING_NODE#	First node ID number of a sequence of node ID numbers that define the group GROUP# (integer).
ENDING_ATTRIBUTE#	Last node ID number of a sequence of node ID numbers that define the group GROUP# (integer).
GROUP#	Group ID number of the constructed group of elements or nodes (integer).

Next: [HZEMFILTER](#), Previous: [GROUPS](#)

41 HEAT ZERO ENERGY MODE

Command Statement:	HZEM
--------------------	-------------

This command is effective only for quasistatic and transient thermal, thermoelastic, aerothermal, and aerothermoelastic analyses involving thermal zero energy modes — that is, a singular conductivity matrix — when the equation solver specified under the [STATICS](#) command is the `sparse` or `skyline` solver. In this case, the constant temperature mode and the generalized inverse of the conductivity matrix are computed using a hybrid physics-algebraic algorithm, rather than the tolerance-based algorithm associated with the [TRBM](#) command.

HZEM

Next: [HEFSB](#), Previous: [HZEM](#)

42 HEAT ZERO ENERGY MODES FILTER

Command Statement:	HZEMFILTER
--------------------	-------------------

This command is effective for thermal simulations involving thermal zero energy modes. When this command is specified in the input file and the thermal problem contains zero energy modes, a non-trivial projector is constructed. In a static or quasistatic analysis, the transpose of this projector is applied to the (possibly variable) right hand-side to make sure that the external load is self-equilibrated and therefore admissible. In a dynamic analysis: a) this projector is applied to the initial solution (initial displacement *and* velocity fields), and b) at each time-step, the transpose of this projector is applied to the (time-dependent) right-hand side of the equation being solved. In all cases, the net effect is to work with (and output) a solution where the thermal zero energy modes have been filtered out.

The constructed projector can be written as $P = I - Z(Z^T Q Z)^{-1} Z^T Q$ where I is the identity

matrix, Z is the matrix of the zero energy modes of the thermal system (always computed with the HZEM method), and Q is either the identity matrix for static analysis, or the capacity matrix for quasistatic and dynamic analyses. Hence, in a quasistatic analysis, this command requires the specific heat to be present in the material properties specified under the MATERIAL command.

HZEMFILTER

Next: [HEFRS](#), Previous: [HZEMFILTER](#)

43 HYDROELASTIC FLUID/STRUCTURE BOUNDARY

Command Statement:	HEFSB
--------------------	--------------

The HEFSB command is used to specify the fluid-structure interface boundary in a hydroelastic eigenvalue problem. The underlying discrete model assumes that each node on this interface is shared by fluid and structural elements. The input format of this command is given below.

Note 1: AERO-S automatically determines that a hydroelastic eigenvalue problem is to be solved once it finds in the input file the HEFSB command, or the HEFRS command which specifies its free-surface boundary. This problem is solved using the added mass approach. The computed frequencies and structural mode shapes can be output as in a regular structural eigen computation.

HEFSB

FACE#	FACE_TYPE	CONNECTIVITY_NODES
-------	-----------	--------------------

FACE# Face (or edge in two dimensions) identification number whose type and connectivity are to be specified (integer).

FACE_TYPE

1	2-node line segment. To be used with the appropriate two-dimensional linear elements.
2	3-node line segment. To be used with the appropriate two-dimensional quadratic elements.
3	3-node triangular face. To be used with the appropriate three-dimensional linear tetrahedral elements.
4	4-node quadrilateral face. To be used with the appropriate three-dimensional linear hexahedral elements.
6	6-node triangular face. To be used with the appropriate three-dimensional quadratic tetrahedral elements.
10	Quadrilateral face. To be used with the appropriate higher-order iso-parametric hexahedral elements.
11	Triangular face. To be used with the appropriate higher-order iso-parametric tetrahedral elements.
12	Line segment. To be used with the appropriate higher-order iso-parametric quadrilateral elements.
13	Line segment. To be used with the appropriate higher-order iso-parametric triangular elements.
CONNECTIVITY_NODES	These should be listed in a stacked fashion on a single line. The nodes of low-order faces should be ordered counter-clockwise (when looking from infinity in three dimensions). Those of higher-order faces (type 10-13) should be ordered from left to right and bottom to top using any convenient axis system (when viewed from infinity in three dimensions).

Next: [HELMHOLTZ](#), Previous: [HEFSB](#)

44 HYDROELASTIC FREE SURFACE BOUNDARY

Command Statement:	HEFRS
--------------------	--------------

The **HEFRS** command can be used to specify the free surface boundary either in a hydroelastic eigenvalue problem, or in an acoustic or elastoacoustic (fluid-structure interaction) problem where the fluid is assumed to be weightless. At each node located on this free surface, **AERO-S** automatically applies a zero Dirichlet boundary condition for the fluid potential equation in the first case, and for the scattered pressure in the second case.

The input format of this command is given below.

Note 1: **AERO-S** automatically determines that a hydroelastic eigenvalue problem is to be solved once it finds in the input file the **HEFRS** command, or the [HEFSB](#) command which specifies its hydroelastic fluid/structure interface boundary. This problem is solved using the added mass approach. The computed frequencies and structural mode shapes can be output as in a regular structural eigen computation.

HEFRS

FACE#	FACE_TYPE	CONNECTIVITY_NODES
-------	-----------	--------------------

FACE#	Face (or edge in two dimensions) identification number whose type and connectivity are to be specified (integer).
FACE_TYPE	Currently, this entry is ignored by AERO-S . Still, it must be assigned a dummy integer value (integer).
CONNECTIVITY_NODES	These should be listed in a stacked fashion on a single line, and ordered clockwise (when looking from infinity in three dimensions).

Next: [HARB](#), Previous: [HEFRS](#)

45 HELMHOLTZ

Command Statement:	HELMHOLTZ
--------------------	------------------

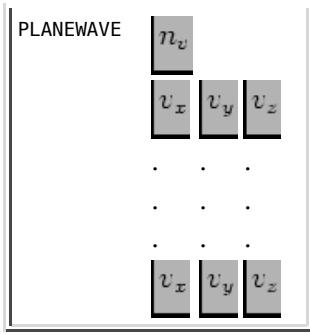
The command statement `HELM` can be used to specify an absorbing boundary condition, and/or a set of incident plane waves or point source terms associated with spherical waves for a frequency-domain acoustic or elastoacoustic (Helmholtz) problem. Its input format is given below.

Note 1: For **AERO-S**, the time-harmonic form of the solution of a formulated Helmholtz problem is $P = p e^{-i\omega t}$, where P denotes the scalar field of interest — for example, the pressure perturbation — p is its amplitude, and ω denotes its circular frequency. When the formulated Helmholtz problem includes an incident wave of amplitude p^i (which can also be specified as a source term), **AERO-S** solves this problem for the *scattered* amplitude $p^s = p - p^i$.

Note 2: If an incident plane wave or a point source term is specified under this command (currently, only one or the other can be specified under `HELM`), and other Dirichlet boundary conditions are also specified under [HDIR](#), only the Dirichlet boundary condition $p^s = -p^i$ associated with the amplitude of the incident wave or point source terms specified under `HELM` are applied to the formulated Helmholtz problem.

HELM

BGTL	i_1	r_1	r_2
POINTSOURCE	n_s		
	x_s	y_s	z_s



BGTL

 i_1

Order (0, 1, or 2) of the generalized Bayliss-Gunburger-Turkel non reflecting condition to be applied on the artificial boundary Σ specified under the [HARB](#) command (integer).

 r_1

This real number (float) is to be used only when the artificial boundary Σ (see [HARB](#)) is supposed to be either a sphere or an ellipsoid, and some geometric approximations generated by **AERO-S** are to be overwritten by values that can in that case be evaluated exactly. Otherwise, r_1 should

simply not be inputted. For a sphere, the curvature of Σ , which is otherwise automatically approximated by **AERO-S** is overwritten by $1/r_1$.

Similarly, for an ellipsoid of the form $(x/a)^2 + (y/b)^2 + (z/c)^2 = 1$,

 $r_1 = a$. r_2

This real number (float) is to be used only when the artificial boundary Σ (see [HARB](#)) is supposed to be an ellipsoid, and some geometric approximations generated by **AERO-S** are to be overwritten by values that can in that case be evaluated exactly. Otherwise, r_2 should simply not be

inputted. For an ellipsoid of the form $(x/a)^2 + (y/b)^2 + (z/c)^2 = 1$,

 $r_2 = b = c$.

POINTSOURCE

Keyword indicating that the Dirichlet and/or Neumann boundary conditions (see [HDIR](#) and [HDNB](#)) of the acoustic (Helmholtz) or elastoacoustic problem defined by this command are associated with a time-harmonic spherical wave of the form $\frac{1}{4\pi} \frac{e^{ikr}}{r}$, where $r = |X - X_s|$,

and propagated by a point source located at the point X_s (character).

Multiple spherical waves and point sources can be specified under this command leading to multiple acoustic or elastoacoustic analyses. Hence, this line should be followed by a line specifying the number of point sources and spherical waves, n_s , and n_s lines specifying for each wave its point source X_s . When used to define Neumann boundary conditions

(see [HDNB](#)), this option is available only for the faces of type 10 and 11 (see [HDNB](#)).

 n_s

Number of point sources propagating spherical waves (integer).

	x coordinate of the location of a point source (float).
	y coordinate of the location of a point source (float).
	z coordinate of the location of a point source (float).
PLANEWAVE	<p>Keyword indicating that the Dirichlet and/or Neumann boundary conditions (see HDIR and HDNB) of the acoustic (Helmholtz) or elastoacoustic problem defined by this command are associated with an incident time-harmonic plane wave of the form $e^{ik\vec{z}\cdot\vec{v}}$ (character). Multiple such waves can be specified under this command leading to multiple acoustic or elastoacoustic analyses. Hence, this line should be followed by a line specifying the number of plane waves to be considered,  , and  lines specifying for each wave its direction of propagation  .</p> <p>Number of incident planar waves (integer).</p>
	Normalized x component of the direction of a planar wave (float).
	Normalized y component of the direction of a planar wave (float).
	Normalized z component of the direction of a planar wave (float).
	

Next: [HDIR](#), Previous: [HELMHOLTZ](#)

46 HELMHOLTZ ARTIFICIAL BOUNDARY *S*

Command Statement:	HARB
--------------------	-------------

The **HARB** command statement is used to specify the artificial boundary Σ on which the absorbing condition specified in the **HELMHOLTZ** command (see [HELMHOLTZ](#)) is to be applied. The input format is given below.

Note 1: In frequency-domain computations, the absorbing condition is applied in general to the solution variable. Because it does not make sense to absorb the incident field, **AERO-S** works in this case with the scattered field as the solution variable.

Note 2: The BGTL (Bayliss-Gunzberger-Turkel-Like) absorbing boundary condition (see [HELMHOLTZ](#)) of order 0 can be applied to any face type described below. The BGTL of order 1 or 2 can be applied only to the face types 1, 2, 3, and 6 described below.

Note 3: When using a BGTL absorbing boundary condition, the face types chosen for constructing the artificial boundary Σ must be compatible with the faces of the Helmholtz elements (see [TOPOLOGY](#)) they overlay (see summary table given below).

FACE_TYPE	Problem Dimension	BGTL Order	ELEMENT_TYPE
1	2D	0,1,2	30,31,33,34,35,36
2	2D	0,1,2	32,38

3	3D	0,1,2	40,41
4	3D	0	44,45
6	3D	0,1,2	42
10	3D	0	95
11	3D	0	96
12	2D	0	98
13	2D	0	99

[HARB]

FACE#	FACETYPE	CONNECTIVITY_NODES
-------	----------	--------------------

FACE# Face (or edge in two dimensions) id number whose type and connectivity are to be specified (integer). In practice, this id number is ignored by **AERO-S**.

FACE_TYPE

- 1 2-node line segment. To be used with two-dimensional linear elements.
- 2 3-node line segment. To be used with two-dimensional quadratic elements.
- 3 3-node triangular face. To be used with three-dimensional linear tetrahedral element.
- 4 4-node quad face. To be used with three-dimensional linear hexahedral element.
- 6 6-node triangular face. To be used with three-dimensional quadratic tetrahedral element.
- 10 4, 9, 16 or 25-node quadrilateral face. To be used with three-dimensional full isoparametric hexahedral elements.
- 11 3, 6, 10 or 15-node triangular face. To be used with three-dimensional full isoparametric tetrahedral elements.
- 12 2, 3, 4 or 5-node line segment. To be used with two-dimensional full isoparametric quadrilateral elements.
- 13 2, 3 or 4-node line segment. To be used with two-dimensional full isoparametric triangular elements.

CONNECTIVITY_NODES These should be listed in a stacked fashion on a single line, and numbered clockwise (when looking from outside in three dimensions).

Next: [HDNB](#), Previous: [HARB](#)

47 HELMHOLTZ DIRICHLET BOUNDARY CONDITIONS *S*

Command Statement:	HDIR
--------------------	-------------

The **HDIR** command statement is used to specify *nodal* Dirichlet boundary conditions for a frequency-domain acoustic scattering (Helmholtz) problem. The input format is given below.

Note 1: When the keyword PLANEWAVE or POINTSOURCE is specified under [HELMHOLTZ](#), it overwrites the REAL_PART_VALUE and IMAGINARY_PART_VALUE (see below) by $p^s = -p^i$, where p^i is the amplitude of an incident time-harmonic plane wave specified by PLANEWAVE, or an that of an incident time-harmonic spherical wave propagating from a source located at a point specified by POINTSOURCE.

[HDIR]

NODE#	DOF#	REAL_PART_VALUE	IMAGINARY_PART_VALUE
-------	------	-----------------	----------------------

NODE#	Node number where the Dirichlet boundary condition is specified (integer).
DOF#	Degree of freedom local number where the boundary condition is specified (integer). This number should be set to 8.
REAL_PART_VALUE	Real part of the value of the specified boundary condition (float).
IMAGINARY_PART_VALUE	Imaginary part of the value of the specified boundary condition (float).

Next: [HNEU](#), Previous: [HDIR](#)

48 HELMHOLTZ DISTRIBUTED NEUMANN BOUNDARY CONDITION *S*

Command Statement: **[HDNB]**

The HDNB command statement can be used to specify on all or segments of the surface of a scatterer a *distributed* Neumann boundary condition of the form $\frac{\partial p^s}{\partial n} = -\frac{\partial p^i}{\partial n}$, where the superscripts *s* and *i*

designate scattered and incident quantities, respectively. The incident time-harmonic wave can be a time-harmonic plane wave specified by PLANEWAVE, or a time-harmonic spherical wave propagating from a source located at a point specified by POINTSOURCE in [HELMHOLTZ](#). Note that for problems involving a nonhomogenous fluid, HDNB can only be used to define a surface that is submerged entirely in one and only one fluid.

The input format of this command is given below.

[HDNB]

FACE#	FACE_TYPE	CONNECTIVITY_NODES
-------	-----------	--------------------

FACE# Face (or edge in two dimensions) id number whose type and connectivity are to

be specified (integer). In practice, this id number is ignored by FEM.

FACE_TYPE	
1	2-node line segment. To be used with two-dimensional linear elements.
2	3-node line segment. To be used with two-dimensional quadratic elements.
3	3-node triangular face. To be used with three-dimensional linear tetrahedral elements.
4	4-node quad face. To be used with three-dimensional linear hexahedral elements.
6	6-node triangular face. To be used with three-dimensional quadratic tetrahedral elements.
10	4, 9, 16 or 25-node quadrilateral face. To be used with three-dimensional full isoparametric hexahedral elements.
11	3, 6, 10 or 15-node triangular face. To be used with three-dimensional full isoparametric tetrahedral elements.
12	2, 3, 4 or 5-node line segment. To be used with two-dimensional full isoparametric quadrilateral elements.
13	2, 3 or 4-node line segment. To be used with two-dimensional full isoparametric triangular elements.
CONNECTIVITY_NODES	These should be listed in a stacked fashion on a single line, and numbered counter clockwise (when looking from infinity in three dimensions).

Next: [HSCB](#), Previous: [HDNB](#)

49 HELMHOLTZ NEUMANN BOUNDARY CONDITIONS *S*

Command Statement:	HNEU [LOADSET_ID]
--------------------	--------------------------

The HNEU command statement is used to specify the *nodal* Neumann boundary conditions for a frequency-domain acoustic scattering (Helmholtz) problem. The input format is given below.

HNEU [LOADSET_ID]

NODE#	REAL_PART_VALUE	IMAGINARY_PART_VALUE
-------	-----------------	----------------------

LOADSET_ID

Optional non-negative integer which identifies explicitly the "load" set to which the source term generated by this command belongs to (integer). The default value is 0. Hence, the HNEU command can be repeated as many times as desired within the same input file using each time a different value for LOADSET_ID and different data. The [LOADCASE](#) command can refer to LOADSET_ID to define one or multiple "load" cases for static analysis (see the [STATICS](#) command and the explanation of its sub-command keyword CASES), and/or the "load" case for dynamic analysis.

NODE#	Node number where the Neumann boundary condition is specified (integer).
REAL_PART_VALUE	Real part of the value of the specified boundary condition (float).
IMAGINARY_PART_VALUE	Imaginary part of the value of the specified boundary condition (float).

Next: [HWIB](#), Previous: [HNEU](#)

50 HELMHOLTZ SCATTERER BOUNDARY *S*

Command Statement:	HSCB
--------------------	-------------

The HSCB command statement is used to specify the surface of a scatterer. It is used to compute the farfield pattern of a frequency-domain acoustic solution, and therefore is necessary for the output of that farfield pattern (see [OUTPUT](#)).

The input format of this command is given below.

HSCB

FACE#	FACE_TYPE	CONNECTIVITY_NODES
-------	-----------	--------------------

FACE#	Face (or edge in two dimensions) id number whose type and connectivity are to be specified (integer). In practice, this id number is ignored by FEM.	
FACE_TYPE	<ul style="list-style-type: none"> 1 2-node line segment. To be used with two-dimensional linear elements. 2 3-node line segment. To be used with two-dimensional quadratic elements. 3 3-node triangular face. To be used with three-dimensional linear tetrahedral elements. 4 4-node quad face. To be used with three-dimensional linear hexahedral elements. 6 6-node triangular face. To be used with three-dimensional quadratic tetrahedral elements. 10 4, 9, 16 or 25-node quadrilateral face. To be used with three-dimensional full isoparametric hexahedral elements. 11 3, 6, 10 or 15-node triangular. To be used with three-dimensional full isoparametric tetrahedral elements. 12 2, 3, 4 or 5-node line segment. To be used with two-dimensional full isoparametric quadrilateral elements. 13 2, 3 or 4-node line segment. To be used with two-dimensional full isoparametric triangular elements. 	
CONNECTIVITY_NODES	These should be listed in a stacked fashion on a single line, and numbered counter clockwise (when looking from outside in three dimensions).	

Next: [HLMPC](#), Previous: [HSCB](#)

51 HELMHOLTZ WET INTERFACE BOUNDARY *S*

Command Statement:	HWIB
--------------------	-------------

The **HWIB** command statement can be used to describe the wet interface boundary of the structure in a coupled frequency response elastoacoustic (fluid-structure) problem where a single mesh is used to discretize both of the computational acoustic fluid and structural domains. Its input format is given below.

Note 1: This command uses the same input format as **HARB** and **HDNB** but is supported only for face types 10, 11, 12 and 13.

Note 2: In this case, the capabilities of the ACME library are not used to generate the fluid-structure coupling coefficients.

HWIB

FACE#	FACETYPE	CONNECTIVITY_NODES
-------	----------	--------------------

FACE# Face (or edge in two dimensions) id number whose type and connectivity are to be specified (integer). In practice, this id number is ignored by FEM.

FACETYPE

10 4, 9, 16 or 25-node quadrilateral face. To be used with three-dimensional full isoparametric hexahedral elements.

11 3, 6, 10 or 15-node triangular. To be used with three-dimensional full isoparametric tetrahedral elements.

12 2, 3, 4 or 5-node line segment. To be used with two-dimensional full isoparametric quadrilateral elements.

13 2, 3 or 4-node line segment. To be used with two-dimensional full isoparametric triangular elements.

CONNECTIVITY_NODES These should be listed in a stacked fashion on a single line, and numbered as shown in the following examples:

9-noded quad:

7 8 9

4 5 6

1 2 3

6-noded triangle:

6

4 5

1 2 3

etc.

Next: [KIRLOC](#), Previous: [HWIB](#)

52 HELMHOLTZ LINEAR MULTIPOINT CONSTRAINTS

Command Statement: **HLMPC**

The **HLMPC** command statement is used to specify a set of *complex* linear multipoint constraint equations of the form

$$\sum_{j=1}^{j=n} c_{ij} u_j = r_i, \quad i = 1, 2, \dots$$

in a frequency-domain acoustic model. There is no limitation on the number of multipoint constraints, or number of degrees of freedom related by the same constraint equation. The format of this command statement is as follows.

Note 1: The Lagrange multiplier method for enforcing the constraints associated with this command is supported only by the FETI-DP family of solvers, the GMRES solver, and the SPOOLES and MUMPS direct sparse solvers (see [STATICS](#)).

HLMPC

CONSTRAINT#	RHS	CONSTRAINT_METHOD	
NODE#	DOF#	REAL-PART-COEFF	IMAGINARY-PART-COEFF
.	.		
NODE#	DOF#	REAL-PART-COEFF	IMAGINARY-PART-COEFF

CONSTRAINT#	This corresponds to the constraint equation number i (integer).		
RHS	This is the right-hand side r_i of the i -th constraint equation (float). It can have both a real and an imaginary part, in which case the real part is specified first and followed by the imaginary part.		
CONSTRAINT_METHOD	This is the method for enforcing the constraint (characters). The default method is set in CONSTRAINTS and used whenever this entry is omitted.		
multipliers	The Lagrange multiplier method.		
elimination	The elimination method.		
penalty [beta]	The penalty method. The parameter β should be a large positive number, typically of the order of 10^8 (no default value is provided).		
NODE#	This is the number of the node contributing the coefficient c_{ij} of the i -th constraint equation (integer).		

DOF#	This is the local number of the degree of freedom at the node specified above contributing the coefficient c_{ij} of the i -th constraint equation (integer).
REAL_PART_VALUE	This is the real part of the coefficient c_{ij} of the i -th constraint equation (float).
IMAGINARY_PART_VALUE	This is the imaginary part of the coefficient c_{ij} of the i -th constraint equation (float).

Next: [IMPEDANCE](#), Previous: [HLMPC](#)

53 HELMHOLTZ LOCATIONS WHERE TO COMPUTE THE KIRCHHOFF INTEGRAL

Command Statement:	KIRLOC
--------------------	---------------

When KIRCHHOFF is selected as an [OUTPUT](#) result and the surface of the scatterer is defined using the command [HSCB](#), this command should be used to specify the locations of the points where the (far-field) solution of a frequency-domain acoustic or elastoacoustic (Helmholtz) problem is to be evaluated using the Kirchhoff integral and outputted. These points do not need to be nodes of the mesh. Usually, but not necessarily, they are outside the computational domain, in the far-field. The keyword KIRLOC should be followed by as many lines as there are points where it is desired to evaluate the solution of the aforementioned Helmholtz problem. Each line contains the coordinates of such a point.

The syntax of this command is given below.

KIRLOC

X-ORDINATE	Y-ORDINATE	Z-ORDINATE
------------	------------	------------

X-ORDINATE	X-ordinate of a point where to evaluate the solution of the Helmholtz problem to be solved using the Kirchhoff integral (real).
Y-ORDINATE	Y-ordinate of a point where to evaluate the solution of the Helmholtz problem to be solved using the Kirchhoff integral (real).
Z-ORDINATE	Z-ordinate of a point where to evaluate the solution of the Helmholtz problem to be solved using the Kirchhoff integral (real).

Next: [IACCELERATIONS](#), Previous: [KIRLOC](#)

54 IMPEDANCE ANALYSIS

Command Statement:	IMPEDANCE	[SWEEP_ID]
--------------------	------------------	------------

The IMPEDANCE command can be used for two different purposes:

- Signal that the problem to be solved is a forced frequency response vibro-acoustic (aka elastoacoustic and fluid-structure), structural dynamic, or acoustic problem of the form

$$Zq = r$$

or more specifically,

$$\begin{bmatrix} K_s - i\omega_j D_s - \omega_j^2 M_s & C^T \\ \omega_j^2 C & \frac{1}{\rho_f} K_f - \frac{\omega_j^2}{\rho_f c_f^2} M_f - \frac{1}{\rho_f} S_a(\omega_j) \end{bmatrix} \begin{bmatrix} u_s \\ p_f \end{bmatrix} = \begin{bmatrix} g_s(\omega_j) \\ \frac{1}{\rho_f} g_f(\omega_j) \end{bmatrix}, \quad j = 1, 2, \dots$$

and specify a solution algorithm for this problem which may involve one or multiple frequency sweep analyses. In the above block matrix equation, i is the pure imaginary number satisfying $i^2 = -1$, ω_j is a circular frequency related to the frequency f_j by $\omega_j = 2\pi f_j$, K_s , M_s , and

D_s are the structural stiffness, mass, and damping matrices, respectively, K_f and M_f are the acoustic fluid stiffness and mass matrices arising from the discretization of the Helmholtz operator, respectively, ρ_f is the acoustic fluid density specified in [MATERIAL](#), c_f is the speed of sound in the acoustic fluid and is also specified in [MATERIAL](#), C is a fluid-structure coupling matrix, u_s , p_f , g_s , and g_f denote the amplitude vectors of the time-harmonic structural displacement ($u_s e^{-i\omega_j t}$), pressure fluctuation field ($u_f e^{-i\omega_j t}$), structural external forcing input ($g_s e^{-i\omega_j t}$), and acoustic fluid external forcing input ($g_f e^{-i\omega_j t}$), respectively, and the superscript T designates the transpose operation.

- To identify within a specified range of interest the eigenvalues missed by a previous eigenvalue computation and their corresponding eigenvectors.

For the purpose of frequency response analysis, structural damping can be represented in **AERO-S** using two different approaches:

- The Rayleigh proportional damping, in which case the matrix D_s takes the form

$$D_s = aK_s + bM_s$$

where a and b are two real scalars that may be specified in this [IMPEDANCE](#) command for the entire structure, and/or in the [MATERIAL](#) command at the material and therefore element level (see the sub-command keyword [DAMPING_TYPE](#) in [MATERIAL](#)).

- The concept of loss factor introduced in the Young modulus of a given material — and thererfore at the element level of a finite element model — as follows

$$E^{e*} = E^e(1 - i\eta)$$

where E^e denotes the element-level Young modulus, the star superscript designates the "damped" Young modulus, i denotes the pure imaginary number satisfying $i^2 = -1$, the minus sign in $-i$ is due to the convention $u_s e^{-i\omega_j t}$, and η is the *loss factor*. In general, η is a

function of the frequency. This function may be specified in the [MATERIAL](#) command (see the sub-command keyword [DAMPING_TYPE](#) in [MATERIAL](#)), or more generally in the form of one or multiple lookup tables (or curves) in [SDETAFT](#).

If the physical domain of the acoustic fluid is unbounded but its computational support is truncated by an artificial boundary surface, S_a is a complex sparse matrix that arises from the discretization on this surface of the absorbing boundary specified in [HELMHOLTZ](#). Alternatively, if the

computational support of an infinite acoustic fluid domain is truncated using a perfectly matched layer (PML) whose properties are specified in [MATERIAL](#). S_a is a complex, non-Hermitian matrix

with non-zero entries only at the degrees of freedom inside the PML. If the parameters of the PML are kept constant when the frequency is varied, or the physical acoustic fluid domain is bounded and delimited by boundary surfaces of the structural domain, the above block matrix equation can be re-written in terms of complex non-Hermitian matrices K_f and M_f so that $S_a = 0$.

The case of a purely acoustic problem is governed by the second row of the above block matrix equation after setting $u_s = 0$. That of a purely structural dynamic problem is given by the first row of the above block matrix equation after setting $p_f = 0$.

Note 1: When structural damping is modeled using the concept of loss factor η outlined above, and the variation of η with the frequency f is described using curves (or lookup tables) that are defined in [SDETAFT](#), special attention should be paid to ensure that:

- In each frequency band of interest — that is, in each frequency band where a sweep is requested — $\eta(f)$ is defined.
- Any frequency band of interest defined as above does not contain within it, except perhaps at its end points, any of the frequency points specified in [SDETAFT](#) to define a curve $\eta(f)$. This is because the fast frequency response algorithms implemented in **AERO-S** assume that $\eta(f)$ is linear within the frequency band where the sweep is to be performed.

The input format of this command is given below.

IMPEDANCE	
FREQ	f
RAYDAMP	$a \ b$
FREQSWEEP1	$f_0 \ \Delta f \ n_{\Delta f}$
FREQSWEEP	$f_l \ f_u \ n_s^* \ n_s$
FREQSWEEP	$f_l \ f_j^* \ n_j$
	$\vdots \ \vdots$
FREQSWEPA	$f_l \ f_u \ n_s \ IMOR \ [\epsilon \ n_s^{*max} \ n_v^{min} \ n_v^{max} \ \Delta n_v]$
RECONS	alg para_1 para_2 para_3

PADEPOLES σ_l σ_u

SWEEP_ID	Optional non-negative integer which identifies explicitly a sweep case (integer). The default value is 0. Hence, the IMPEDANCE command can be repeated as many times as desired within the same input file using each time a different value for SWEEP_ID and different parameters. Doing so defines a multiple sweep case that is managed by the IMPEDANCE command itself.
FREQ	Sub-command keyword for specifying a forced frequency in the case of a single frequency response analysis (characters).
f	Value of the forced frequency (real). The corresponding value of ω is $2\pi f$.
RAYDAMP	Sub-command keyword for specifying Rayleigh proportional damping coefficients for the entire structure (characters).
a	Rayleigh damping stiffness coefficient (real).
b	Rayleigh damping mass coefficient (real).
FREQSWEEP1	Sub-command keyword for requesting a frequency sweep analysis using a one-point configuration of the reconstruction algorithm specified after the sub-command keyword RECONS. In this case, the following ($2n_{\Delta_f} + 1$) frequencies f_j are swept in the frequency band $[f_0 - n_{\Delta_f}\Delta_f, f_0 + n_{\Delta_f}\Delta_f]$
	$f_j = f_0 + j\Delta_f, \quad -n_{\Delta_f} \leq j \leq n_{\Delta_f}$
f_0	Frequency defining the center of the frequency band $[f_0 - n_{\Delta_f}\Delta_f, f_0 + n_{\Delta_f}\Delta_f]$ (real).
Δ_f	Frequency sweep increment (real).
n_{Δ_f}	Together with f_0 and Δ_f , this parameter defines the frequency band $[f_0 - n_{\Delta_f}\Delta_f, f_0 + n_{\Delta_f}\Delta_f]$ (integer).
FREQSWEEP	Sub-command keyword for requesting a frequency sweep analysis in a frequency band $[f_l, f_u]$ of interest using a reconstruction algorithm specified after the sub-command keyword RECONS (characters). In this case, the user can choose between two different schemes for specifying a set of interpolation frequency points within $[f_l, f_u]$ — that is, a set of frequencies where the response and its first few consecutive frequency derivatives are to be computed using the brute force approach — and then sampling frequencies in $[f_l, f_u]$ and rapidly computing the frequency response function at these points using the reconstruction algorithm alg specified after the sub-command keyword RECONS discussed below.
	The first scheme, referred to here as the "regular" sampling scheme, introduces $n_s^* \geq 2$ equally-spaced interpolation frequencies f_j^* in $[f_l, f_u]$, including f_l and f_u . Hence, these frequencies are given by

$$f_j^* = f_l + (j - 1) \frac{(f_u - f_l)}{(n_s^* - 1)}, \quad j = 1, \dots n_s^*$$

Then, the regular scheme samples each interval $[f_{j-1}^*, f_j^*]$ into $n_s \geq 1$ equal frequency increments, and therefore into $(n_s + 1)$ frequency points including f_{j-1}^* and f_j^* , where it rapidly reconstructs the frequency response function using the algorithm `alg`. Hence, the first scheme computes the frequency response at $((n_s^* - 1)n_s + 1)$ frequencies in the frequency band $[f_l, f_u]$ (counting f_l and f_u).

The second scheme, referred to here as the "irregular" sampling scheme, requires the user to specify first on the first line after the keyword `FREQSWEET` the lower end of the frequency spectrum, f_l , and then every *other* desired interpolation frequency f_j^* (in ascending order) on a separate line together with an integer number n_j specifying the sampling of the interval $[f_{j-1}^*, f_j^*]$ into n_j equal frequency increments. Hence, it defines $(n_j + 1)$ sampling frequencies in $[f_{j-1}^*, f_j^*]$ including f_{j-1}^* and f_j^* . The upper end of the frequency spectrum, f_u , is in this case the last inputted interpolation frequency. It follows that if, for example, n_s^* interpolation frequencies are inputted by the user, and n_j is set to $n_j = n_s$ for all j (again, for example), the second scheme computes the frequency response also at $((n_s^* - 1)n_s + 1)$ frequencies in the frequency band $[f_l, f_u]$, including f_l and f_u .

f_l

f_u

n_s^*

Lower end of the frequency band of interest $[f_l, f_u]$ (real).

Upper end of the frequency band of interest $[f_l, f_u]$ (real).

Number of equally-spaced interpolation frequencies f_j^* to introduce in the frequency band of interest $[f_l, f_u]$, including f_l and f_u (integer). Hence, these frequencies are given by

$$f_j^* = f_l + (j - 1) \frac{(f_u - f_l)}{(n_s^* - 1)}, \quad j = 1, \dots n_s^*$$

n_s

When part of the definition of `FREQSWEET`, this parameter specifies the number of equally-spaced frequency intervals in which a frequency "sub-band" $[f_{j-1}^*, f_j^*]$ is to be sampled (integer). In this case, $[f_{j-1}^*, f_j^*]$ is sampled into $(n_s + 1)$

frequencies, including f_{j-1}^* and f_j^* . When part of the definition of FREQSWEPA, this parameter specifies the number of equal size frequency intervals in which the frequency band of interest $[f_l, f_u]$ is to be sampled (integer). In this case,

$[f_l, f_u]$ is sampled into $(n_s + 1)$ frequencies, including f_l and f_u . When part of the definition of FREQSWEEP, the inputted value of this parameter must satisfy $n_s \geq 1$. When part of the definition of FREQSWEPA, it must satisfy $n_s \geq 2$. In both cases, the frequency response function is computed at all sampled frequency points.

 f_j^*

Interpolation frequency (real).

 n_j

Number of equal size frequency increments in which a frequency interval of the form $[f_{j-1}^*, f_j^*]$ is to be sampled (integer). In this case, $[f_{j-1}^*, f_j^*]$ is sampled into $(n_j + 1)$ frequency points.

FREQSWEPA

Sub-command keyword requesting an *adaptive* frequency sweep analysis in a frequency band $[f_l, f_u]$ of interest using a reconstruction algorithm specified after the sub-command keyword RECONS (characters). In this case, the user can set the maximum number of interpolation frequencies, specify the number of frequencies to sample and at which to reconstruct the frequency response function, specify the reconstruction algorithm, and tune some of the parameters of the automatic adaptation procedure.

IMOR

Specifies an interpolatory model order reduction algorithm for the adaptive frequency sweep procedure associated with the sub-command keyword FREQSWEPA. Currently, two options are available: (1) GalProjection, and (2) KrylovGalProjection. These two algorithms are the same as those associated with the sub-command keyword RECONS and are described below. It is recommended to use KrylovGalProjection as much as possible, except for acoustic scattering problems and coupled fluid-structure interaction problems with structural damping and/or absorbing boundary conditions where KrylovGalProjection is not valid.

 ϵ

Specifies the tolerance level for assessing the convergence of the adaptive frequency sweep procedure associated with the sub-command keyword FREQSWEPA using the criterion

$$\frac{\|Z(f_i)\tilde{q} - r(f_i)\|_2}{\|r(f_i)\|_2} \leq \epsilon$$

where \tilde{q} is the reconstructed value of q using the chosen interpolatory model order reduction algorithm. The recommended setting for this parameter is $10^{-6} \leq \epsilon \leq 10^{-2}$, and the default setting is $\epsilon = 10^{-2}$.

 n_s^{max}

Specifies the maximum number of interpolation frequencies for the adaptive frequency sweep procedure associated with the sub-command keyword FREQSWEPA. The recommended practice is $6 \leq n_s^{max} \leq 8$. If for such a setting of

this parameter the value of the tolerance level ϵ (see below) is not reached, a higher value of n_s^{max} should be used, or the frequency band of interest should be split in two smaller bands and a separate frequency sweep should be

performed in each one of them instead. The default value of $n_s^{*\max}$ is 6.

n_v^{\min}

Specifies the minimum number of solution vectors to be computed per interpolation frequency for the adaptive frequency sweep procedure associated with the sub-command keyword FREQSWEPA. The recommended value is the default value 8.

n_v^{\max}

Specifies the maximum number of solution vectors to be computed per interpolation frequency for the adaptive frequency sweep procedure associated with the sub-command keyword FREQSWEPA. The recommended values are 48 for the KrylovGalProjection algorithm, and 16 - 24 for the GalProjection algorithm. The default values for this parameter are 16 when IMOR = GalProjection, and 48 when IMOR = KrylovGalProjection.

Δn_v

Specifies the increment number of solution vectors per interpolation frequency to be considered for computation for the adaptive frequency sweep procedure associated with the sub-command keyword FREQSWEPA. The recommended value is the default value 4.

RECONS

Sub-command keyword for defining a reconstruction algorithm in the case of a frequency sweep analysis and setting its parameters (characters).

alg

Name of the reconstruction algorithm (characters). Five such algorithms are available and listed below. The default value is Taylor with 8 derivatives.

Taylor

Taylor series expansion algorithm (characters).

Pade

Conventional multipoint Pade series expansion algorithm (characters).

PadeLanczos

Multipoint Pade series expansion algorithm based on a Lanczos procedure (characters). This algorithm is less prone to ill-conditioning and therefore better performing than the Pade algorithm. However, it is available only for purely structural or acoustic *undamped* frequency response problems where furthermore K and M are symmetric.

GalProjection

Interpolatory model order reduction algorithm based on a Galerkin projection and orthogonalized frequency derivatives. This algorithm is also known as the DGP algorithm.

KrylovGalProjection

Interpolatory model order reduction algorithm based on a Galerkin projection and a Krylov subspace. This algorithm, which cannot be used for acoustic scattering problems and coupled fluid-structure problems with structural damping and/or absorbing boundary conditions, delivers nevertheless a much better performance than both of the Pade and GalProjection algorithms for all other frequency response problems of the form given at the beginning of the description of the IMPEDANCE command. It is also known as the KGP algorithm.

para_1

For the Taylor algorithm, this parameter specifies the order of the Taylor series (integer). For the Pade, PadeLanczos, GalProjection, and KrylovGalProjection algorithms, it specifies how many of the n_s^* interpolation frequencies to use at a time to

apply a multipoint version of the chosen reconstruction algorithm. (Note however that for GalProjection and KrylovGalProjection, para_1 = 1 and para_1 = n_s^*)

are currently the only options).

para_2

This parameter is relevant only for the Pade, GalProjection, and KrylovGalProjection reconstruction algorithms (integer). For Pade, it specifies the order of the numerator of the rational function (L of [L/M]) (integer). For GalProjection, it specifies the number of frequency derivatives of the response to compute at each interpolation frequency. For KrylovGalProjection, it specifies the number of Krylov vectors to compute at each interpolation frequency.

para_3

This parameter is relevant only for the `Pade` and `PadeLanczos` reconstruction algorithms. It specifies the order of the denominator of the rational function (M of $[L/M]$) (integer).

PADEPOLES

This sub-command keyword is active only with `alg = PadeLanczos` and needs input from the sub-command `FREQSWEEP` (see above). It also requires the presence in the input file of the [MODE](#) command to retrieve the EIGENMODES file or files associated with a previous eigen computation. This (these) file(s) should be placed in the execution path. In this case, this sub-command instructs **AERO-S** to: (1) compute the approximation by a multipoint Pade expansion of a rational transfer function whose poles are exactly the eigenvalues of the symmetric pencil of interest (K, M), and (2) exploit these poles to identify, in a specified range of interest $[\sigma_l, \sigma_u]$, the eigenvalues that may have been missed by a previous eigen computation in which the [MODE](#) command was used to store the results in the EIGENMODES file or files (characters). The multipoint Pade expansion is constructed using points $\sigma_j \in [\sigma_l, \sigma_u]$ that should be generated by the sub-command `FREQSWEEP` with σ_j playing the role of f_j , $f_l = \sigma_l$ and $f_u = \sigma_u$ (and for example, $n_s = 0$). The parameters of this expansion should be specified in `para_1`, `para_2` and `para_3` (see above). At the end of the computation, **AERO-S** outputs on the screen the poles of the Pade rational function in the specified range $[\sigma_l, \sigma_u]$, excluding those poles corresponding to the modes read from the EIGENMODES file or files. The specified range $[\sigma_l, \sigma_u]$ can be narrower than that of the eigenvalues read in the EIGENMODES file or files. However, if it is wider, some of the poles of the Pade rational function may, in some cases, not correspond to missed eigenvalues. In any case, the output of the eigenvectors associated with the poles or missed eigenvalues is not currently implemented in **AERO-S**.

 σ_l

Lower end of an eigenvalue interval of interest (real).

 σ_u

Upper end of an eigenvalue interval of interest (real).

Next: [IDISPLACEMENTS](#), Previous: [IMPEDANCE](#)

55 INITIAL ACCELERATIONS (Not Supported Yet)

Command Statement:

IACCELERATIONS

The `IACCELERATIONS` command statement is used to specify a nodal acceleration type of initial conditions. Each node can have up to six degrees of freedom.

IACCELERATIONS

NODE#	DOF#	VALUE
-------	------	-------

NODE# Node number where the initial acceleration is specified (integer).
 DOF# Degree of freedom local number where the initial acceleration is specified (integer).
 VALUE Value of the specified initial acceleration (float).

Next: [IDISP6](#), Previous: [IACCELERATIONS](#)

56 INITIAL DISPLACEMENTS *S*

Command Statement:	IDISPLACEMENTS
--------------------	-----------------------

For a structural analysis, the IDISPLACEMENTS command statement can be used to specify a nodal (or generalized coordinate) displacement type of initial conditions (in this case, each node can have up to six degrees of freedom). For a time-domain acoustic analysis, this command statement can be used to specify initial nodal values of the solution by setting DOF# to 8 (see below).

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: If the [GEPS](#) command is not specified, and both IDISPLACEMENTS and IDISP6 commands are present in the input file, **AERO-S** selects the [IDISP6](#) command to initialize the displacement field.

Note 3: If the [GEPS](#) command is not specified and both IDISPLACEMENTS and IDISP6 are present in the input file, **AERO-S** selects [IDISP6](#) to initialize the displacement field. If on the other hand the [GEPS](#) command is present in the input file, then **AERO-S** uses the content of the [IDISP6](#) command to construct the geometric stiffness, and that of IDISPLACEMENTS to initialize the displacement field. If in this case IDISPLACEMENTS is not specified in the input file, then **AERO-S** initializes the displacement field to zero.

Note 4: In the context of a linearized (perturbation) analysis, the initial displacements specified under this command are interpreted as initial displacement perturbations.

When working with the usual basis, the format of this command is as follows.

IDISPLACEMENTS

or

IDISPLACEMENTS ZERO

NODE#	DOF#	VALUE
-------	------	-------

NODE# Node number where the initial displacement or acoustic field is specified (integer).
DOF# Degree of freedom local number where the initial displacement or acoustic field is specified (integer).
VALUE Value of the specified initial displacement or acoustic field (real).
ZERO This initializes the entire intital displacement or acoustic field to zero. No other **IDISPLACEMENTS** data has to be entered when this option is selected (characters).

When working in the basis of generalized coordinates (or reduced basis), the format of this command is as follows.

IDISPLACEMENTS	
MODAL	
MODE#	VALUE

MODAL This keyword, completely spelled out, can be used either (1) to input the initial conditions in the generalized coordinates (or reduced) system, whether a modal or direct time-integration analysis is requested in the **DYNAMICS** command, or (2) to superpose to the initial conditions specified in the finite element coordinates system additional initial conditions formulated in the generalized coordinates (or reduced) system (characters). In the second case, **IDISPLACEMENTS** and/or **IVELOCITIES** should be specified only once and the keyword **MODAL** can be inserted either before or after the set of data associated with the initial conditions specified in the finite element coordinates. This option can also be used to perform a "Ping-Pong" analysis (**AERO**). In both cases, the **READMODE** command statement must also be present in the input file to provide the eigenmodes to be used.
MODE# Mode id number for which the initial condition is specified (integer).
VALUE Value of the specified initial condition (real).

Next: [IDISP6PITA](#), Previous: [IDISPLACEMENTS](#)

57 INITIAL DISPLACEMENT 6 COLUMNS (IDISP6 completely spelled out) *S*

Command Statement:	IDISP6 [<input type="text"/>]
--------------------	--

The **IDISP6** command statement is used to specify a nodal displacement type of initial conditions in a different format than that of the [IDISPLACEMENTS](#) command. This format is specified below.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of

freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: If both this command and the command [LMPC](#) are specified in the input file, the content of [IDISP6](#) must satisfy the linear multipoint constraints described in [LMPC](#).

Note 3: If the [GEPS](#) command is not specified, and both [IDISPLACEMENTS](#) and [IDISP6](#) commands are present in the input file, **AERO-S** selects the [IDISP6](#) command to initialize the displacement field.

Note 4: If the [GEPS](#) command is present in the input file, then **AERO-S** uses the content of the command [IDISP6](#) to construct the geometric stiffness, and the content of [IDISPLACEMENTS](#) to initialize the displacement field. If [IDISPLACEMENTS](#) is not specified in the input file, then **AERO-S** uses the command [IDISP6](#) to perform both tasks of constructing the geometric stiffness and initializing the displacement field.

Note 5: Modal initial condition is not supported under [IDISP6](#) because it is not relevant.

Note 6: In the context of a linearized (perturbation) analysis, the initial displacements specified under this command are interpreted as initial displacement perturbations.



NODE#	VAL_DOF1	VAL_DOF2	VAL_DOF3	VAL_DOF4	VAL_DOF5	VAL_DOF6
-------	----------	----------	----------	----------	----------	----------



Amplification factor that multiplies each VAL_DOF*i* value for each node (real).

NODE#

Node number where the initial displacement is specified (integer).

VAL_DOF*i*#

Value of the specified initial displacement for the *i*-th degree of freedom (real).

Next: [IVEL6PITA](#), Previous: [IDISP6](#)

58 INITIAL OR SEED DISPLACEMENT FOR PITA (IDISP6PITA completely spelled out)

Command Statement:	IDISP6PITA
--------------------	-------------------

The [IDISP6PITA](#) command statement can be used to specify the time-slices' initial or seed displacements for the PITA methodology (see [PITA](#)), thus overriding the alternative approach consisting of generating this information by applying the basic time-integrator on the coarse time-grid. It is applicable in both linear and nonlinear settings. Its syntax is similar to that of the

[IDISP6](#) (see [IDISP6](#)) command except that it can introduce up to as many sets of initial displacements as there are time-slices, one set per time-slice. Each set of initial displacements must be input after the previous one, starting with that corresponding to the first time-slice and continuing in consecutive order. All those time-slices for which a set of seed displacement values is not specified are initialized by the aforementioned alternative approach.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

IDISP6PITA	NUMSLICES
------------	-----------

NODE#	VAL_DOF1	VAL_DOF2	VAL_DOF3	VAL_DOF4	VAL_DOF5	VAL_DOF6
-------	----------	----------	----------	----------	----------	----------

NUMSLICES	Number of consecutive time-slices for which seed displacements are specified (integer).
NODE#	Node number where the seed displacement is specified (integer).
VAL_DOFi#	Value of the seed displacement for the i-th degree of freedom at the beginning of a time-slice (float).

Next: [ITEMPERATURES](#), Previous: [IDISP6PITA](#)

59 INITIAL OR SEED VELOCITY FOR PITA (IVEL6PITA completely spelled out)

Command Statement:	IVEL6PITA
--------------------	------------------

The IVEL6PITA command statement can be used to specify the time-slices' initial or seed velocities for the PITA methodology (see [PITA](#)), thus overriding the alternative approach consisting of generating this information by applying the basic time-integrator on the coarse time-grid. It is applicable in both linear and nonlinear settings. Its syntax is similar to that of the IDISP6 (see [IDISP6](#)) command except that it can introduce up to as many sets of initial velocities as there are time-slices, one set per time-slice. Each set of initial velocities must be input after the previous one, starting with that corresponding to the first time-slice and continuing in consecutive order. All those time-slices for which a set of seed velocity values is not specified are initialized by the aforementioned alternative approach.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

IVEL6PITA	NUMSLICES
-----------	-----------

NODE#	VAL_DOF1	VAL_DOF2	VAL_DOF3	VAL_DOF4	VAL_DOF5	VAL_DOF6
-------	----------	----------	----------	----------	----------	----------

NUMSLICES Number of consecutive time-slices for which seed velocities are specified (integer).
 NODE# Node number where the seed velocity is specified (integer).
 VAL_DOFi# Value of the seed velocity for the i-th degree of freedom at the beginning of a time-slice (float).

Next: [IVELOCITIES](#), Previous: [IVEL6PITA](#)

60 INITIAL TEMPERATURES *S*

Command Statement:	ITEMPERATURES
--------------------	----------------------

The ITEMPPERATURES command statement is used to specify a nodal displacement type of initial conditions. Each node can have one degree of freedom.

ITEMPPERATURES

NODE	VALUE
------	-------

NODE# Node number where the initial temperature is specified (integer).
 VALUE Value of the specified initial temperature (float).

Next: [INPC](#), Previous: [ITEMPPERATURES](#)

61 INITIAL VELOCITIES *S*

Command Statement:	IVELOCITIES
--------------------	--------------------

The IVELOCITIES command statement is used to specify a nodal velocity type of initial conditions. Each node can have up to six degrees of freedom. For a time-domain acoustic analysis, this command statement can be used to specify nodal values of the initial time-derivative of the solution by setting DOF# to 8 (see below).

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

IVELOCITIES

NODE#	DOF#	VALUE
-------	------	-------

NODE# Node number where the initial velocity is specified (integer).
 DOF# Degree of freedom local number where the initial velocity is specified (integer).
 VALUE Value of the specified initial temperature (float).

When working in the basis of generalized coordinates (or reduced basis), the format of this command is as follows.

IVELOCITIES	
MODAL	
MODE#	VALUE

MODAL This keyword, completely spelled out, can be used either (1) to input the initial solution time-derivatives in the generalized coordinates (or reduced) system, whether a modal or direct time-integration analysis is requested in the [DYNAMICS](#) command, or (2) to superpose to the initial velocity conditions specified in the chosen coordinates system additional initial velocity conditions formulated in the generalized coordinates (or reduced) system (characters). In the second case, IVELOCITIES should be specified only once and the keyword MODAL can be inserted either before or after the set of data associated with the initial velocity conditions specified in the chosen coordinates system. The [READMODE](#) command statement must also be present in the input file to provide the eigenmodes to be used.
 MODE# Mode id number for which the initial condition is specified (integer).
 VALUE Value of the specified initial condition (real).

Next: [MATERIAL](#), Previous: [IVELOCITIES](#)

62 INTRUSIVE POLYNOMIAL CHAOS

Command Statement:	INPC
--------------------	------

The INPC command signals to the **AERO-S** code to perform an intrusive non deterministic analysis using the Polynomial Chaos representation of the solution. It requires that the input file also contains the [GROUP](#), [RANDOM](#), and [STATICS](#) commands and their respective data.

Note 1: Currently, this command is limited to linear, static, structural analysis.

Note 2: Currently, this command is supported only by the PCG, BCG, and CR iterative solvers (see [STATICS](#)).

The input format of this command is given below.

INPC

DEG_OUT

DEG_OUT Degree of the polynomial chaos representation of the solution (integer).

Next: [LMPC](#), Previous: [INPC](#)

63 LINEAR MATERIAL PROPERTIES *S*

Command Statement:	MATERIAL
--------------------	----------

The MATERIAL command statement is used to signal that the following data lines correspond **and/or point to**: (1) definitions of materials and/or geometric properties for rigid elements and linear elastic or acoustic elements, and/or geometric properties for materially nonlinear elements, and/or (2) requests for specific methods to enforce the constraints associated with joint and rigid elements (see [CONSTRAINTS](#)). Several input formats are possible for this command and are described below.

Note 1: Even for a given input format, the interpretation of a data entry can change from one element type to another. For example for the first input format, the main interpretation of some data entries changes when the element type is that of a lumped torsional spring, a torsional spring connector, a linear spring connector, a Timoshenko beam, a thermal element, a rigid translational link, or a rigid rotational link.

Note 2: For composite elements, this command is not required unless stress and/or strain output is requested. Even in this case, only the total thickness of the element must be specified under this command. Dummy or zero values can be entered for the other properties as their real values should be specified under the COMPOSITE command.

Note 3: The Perfectly Matched Layer (PML) method is supported only by the elements type 95, 96, 98, and 99.

Note 4: Specifying under this command a method for enforcing the constraints associated with joint and/or rigid elements sharing a certain material ID number overrides for these elements the specification under [CONSTRAINTS](#) of a default method for enforcing constraints.

MATERIAL

MID	A	E	ν	ρ	h	k	t	P	Ta	c_p	w	Ixx	Iyy	Izz	ymin
ymin	zmin	zmax	DAMPING_TYPE		a	b									

MID	The material id number.
A	Cross sectional area for trusses, beams, and lineal thermal elements (float).
E	Young's Modulus for all element types (float). However, this field can also be filled by a negative integer in which case it means that Young's Modulus is a function of temperature and is to be determined from the YMTT and TEMPERATURES commands. In the latter case, the integer specified here points to the id of the Young's modulus-temperature table in the YMTT command (negative integer).
ν	Poisson's ratio for all element types except trusses (float).
ρ	Mass density per unit volume except for element type 20 (composite or orthotropic shell element) when defined using the COEF option, in which case it is mass density per unit area (float).
h	Heat convection coefficient (only for boundary convection elements: type 47, 48, and 49 (float)).
k	Heat conduction coefficient (float).
t	Element thickness (float).
P	Perimeter/circumference area for thermal elements (float), or the depth of the boundary where convection occurs when element type 47 is used to model non-lateral boundary convection (float).
Ta	Reference temperature (Absolute) of the element (float). WARNING: if this value is different from the nodal temperature of that element or this value is different from zero and no nodal temperature is specified, it creates a thermal loading even in a pure structural analysis.
c_p	Specific heat coefficient in a thermal analysis (float).
w	Coefficient of thermal expansion in either a thermal analysis, or a structural analysis if the specified ambient temperature τ_a is different from the specified nodal temperatures (see TEMPERATURES and THERMOE) (float). In a thermal analysis, this field can also be filled by a negative integer meaning that the coefficient of thermal expansion is a function of temperature and is to be determined from the TETT and TEMPERATURES commands. In this case, the integer specified here points to the id of the coefficient of thermal expansion-temperature table in the TETT command (negative integer).
Ixx/ss	For a mechanical analysis (MECH under the DYNAMICS command), this is the cross-sectional moment of inertia about the local and centroidal x-axis. For an acoustic analysis in the time-domain (ACOU under the DYNAMICS command), this is the speed of sound in the medium represented by the attribute number.
Iyy	Cross-sectional moment of inertia about the local and centroidal <i>principal</i> y-axis.
Izz	Cross-sectional moment of inertia about the local and centroidal <i>principal</i> z-axis.
ymin	Negative local y-coordinate of the bottom fiber of a beam cross section.
ymax	Positive local y-coordinate of the top fiber of a beam cross section.
zmin	Negative local z-coordinate of the top fiber of a beam cross section.
zmax	Positive local z-coordinate of the top fiber of a beam cross section.

DAMPING_TYPE Optional sub-command keyword to specify a type of damping (characters). Two options are currently available:

RAYDAMP This option specifies a Rayleigh proportional type of damping for structural dynamic (see [DYNAMICS](#)) or frequency response (see [IMPEDANCE](#)) analysis. In this case, damping is introduced at the element level in the form of a finite element damping matrix C^e that is constructed as

$$D^e = aK^e + bM^e$$

where M^e and K^e denote the element-level mass and stiffness matrices, respectively, and a and b are two real coefficients that are specified below.

STRDAMP This option specifies a structural type of damping for frequency response analysis (see [IMPEDANCE](#)). In this case, damping is introduced in the form of the following modified local (element-level) Young modulus

$$E^e^* = E^e(1 - i\eta)$$

where E^e denotes the element-level Young modulus, the star superscript designates the modified Young modulus, i denotes the pure imaginary number satisfying $i^2 = -1$, the minus sign in $-i$ is due to the convention $u_s e^{-i\omega_j t}$ (see [IMPEDANCE](#)), and η is the *loss factor*. In general, η is a function of the forced frequency $f = \omega/2\pi$. If in the frequency range of interest η can be

represented as a linear function of f — that is,

$$\eta(f) = \eta_1 + \eta_2 f$$

this linear representation can be specified in the [MATERIAL](#) command by inputting $a = \eta_1$ and $b = \eta_2$ below and specifying f in [IMPEDANCE](#). Otherwise, an

arbitrary variation of the loss factor with the frequency can be inputted in the form of one or multiple lookup tables (or curves) using [SDETAFT](#), in which case given a frequency f specified in [IMPEDANCE](#), η is determined by local interpolation.

a If DAMPING_TYPE is set to RAYDAMP, a specifies the Rayleigh damping stiffness coefficient (real) — that is, the stiffness coefficient in the Rayleigh proportional damping matrix D^e for the material identified by [MID](#). In the nonlinear case, this coefficient is assigned at each iteration to the linearized stiffness matrix and the product of these two quantities, which varies with the nonlinear iterations, constitutes the stiffness contribution to the damping term. In a structural dynamic analysis, this value overrides for the material identified by [MID](#) any value of the Rayleigh damping stiffness coefficient specified under [DYNAMICS](#). In any analysis performed using the [IMPEDANCE](#) command, this value overrides for the material identified by [MID](#) any value of the Rayleigh damping stiffness coefficient specified under [IMPEDANCE](#).

On the other hand, if DAMPING_TYPE is set to STRDAMP, a has a different meaning that pertains to the loss factor η briefly discussed above. In this case, if the loss factor of the material identified by [MID](#) is to be represented as a linear function

of the frequency f of the form $\eta(f) = \eta_1 + \eta_2 f$ (see above), then $a = \eta_1$ and $b = \eta_2$ should be specified as such. However, if the loss factor is to be interpolated using an arbitrary lookup table defined in [SDETAFT](#), a should be set to the *negative integer* value $-curve_id$ (integer), where $curve_id$ is the "id number" of the *relevant* lookup table defined in [SDETAFT](#) using the sub-command keyword [CURVE](#). In that case, b (see below) should be simply ignored.

b If [DAMPING_TYPE](#) is set to [RAYDAMP](#), b specifies the Rayleigh damping mass coefficient (real) — that is, the mass coefficient in the Rayleigh proportional damping matrix D^e (real) for the material identified by [MID](#) (see above). In a structural dynamic analysis, this value overrides for the material identified by [MID](#) any value of the Rayleigh damping mass coefficient specified under [DYNAMICS](#). In any analysis performed using the [IMPEDANCE](#) command, this value overrides for the material identified by [MID](#) any value of the Rayleigh damping mass coefficient specified under [IMPEDANCE](#).

On the other hand, if [DAMPING_TYPE](#) is set to [STRDAMP](#), b has a different meaning that pertains to the loss factor η briefly discussed above. In this case, if the loss factor of the material identified by [MID](#) is to be represented as a linear function of the frequency f of the form $\eta(f) = \eta_1 + \eta_2 f$ (see above), then $b = \eta_2$ and $a = \eta_1$ should be specified as such. However, if the loss factor is to be interpolated using an arbitrary lookup table defined in [SDETAFT](#), a (see above) should be set to the *negative integer* value $-curve_id$ (integer), where $curve_id$ is the "id number" of the *relevant* lookup table defined in [SDETAFT](#) using the sub-command keyword [CURVE](#), and b should be simply ignored.

If the element is a lumped torsional spring ([eltyp](#) = 11), a linear spring connector ([eltyp](#) = 21), or a torsional spring connector ([eltyp](#) = 22), the material properties are defined as follows.

MID	Kx	Ky	Kz	lx1	ly1	lz1	lx2	ly2	lz2	lx3	ly3	lz3	l3z
DAMPING	c												

MID	The material id number from element attribute table.		
Kx	Torsional/Translational spring constant along local x-axis (float).		
Ky	Torsional/Translational spring constant along local y-axis (float).		
Kz	Torsional/Translational spring constant along local z-axis (float).		
lx1	ly1	lz1	The first axis of the local frame expressed in the global frame (floats).
lx2	ly2	lz2	The second axis of the local frame expressed in the global frame (floats).
lx3	ly3	lz3	The third axis of the local frame expressed in the global frame (floats).
DAMPING	Sub-command keyword for specifying a damping coefficient (characters).		
c	Damping coefficient (real).		

If the element is a Timoshenko beam ([eltyp](#) = 7), the material properties are defined as follows.

MID	A	E	ν	ρ	α_Y	α_Z	C1	P	Ta	c_p	w	Ixx	Iyy	Izz
ymin	ymax		zmin	zmax	DAMPING_TYPE		a	b						

MID	The material id number from element attribute table.
A	Cross sectional area (float).
E	Young's Modulus (float).
ν	Poisson's ratio (float).
ρ	Mass density per unit volume (float).
α_Y	Shear deflection constant associated with Iyy (float).
α_Z	Shear deflection constant associated with Izz (float).
C1	Non-uniform torsion constant (float).
P	Not Applicable
Ta	Not Applicable
c_p	Not Applicable
w	Not Applicable
Ixx	Cross-sectional moment of inertia about the local and centroidal x-axis
Iyy	Cross-sectional moment of inertia about the local and centroidal <i>principal</i> y-axis.
Izz	Cross-sectional moment of inertia about the local and centroidal <i>principal</i> z-axis.
ymin	Negative local y-coordinate of the bottom fiber of a beam cross section.
ymax	Positive local y-coordinate of the top fiber of a beam cross section.
zmin	Negative local z-coordinate of the top fiber of a beam cross section.
zmax	Positive local z-coordinate of the top fiber of a beam cross section.
DAMPING_TYPE	Optional sub-command keyword to specify a type of damping (characters). Two options are currently available:
RAYDAMP	This option specifies a Rayleigh proportional type of damping for structural dynamic (see DYNAMICS) or frequency response (see IMPEDANCE) analysis (see above for further details).
STRDAMP	This option specifies a structural type of damping for frequency response analysis (see IMPEDANCE) (see above for further details).
a	If DAMPING_TYPE is set to RAYDAMP, a specifies the Rayleigh damping stiffness coefficient (real). On the other hand, if DAMPING_TYPE is set to STRDAMP, then $a = \eta_1$, where η_1 is the first component of the loss factor $\eta(f) = \eta_1 + \eta_2 f$ (see above for further details).
b	If DAMPING_TYPE is set to RAYDAMP, b specifies the Rayleigh damping mass coefficient (real). On the other hand, if DAMPING_TYPE is set to STRDAMP, then $b = \eta_2$, where η_2 is the second component of the loss factor $\eta(f) = \eta_1 + \eta_2 f$ (see above for further details).

If the element is a rigid translational or rotational link (eltyp = 68 or 69), the material properties are defined as follows.

MID	x	y	z	dum									
dum	dum	dum	dum										

- MID The material id number from element attribute table.
- x Any nonzero value implies a rigid motion in this direction (float).
- y Any nonzero value implies a rigid motion in this direction (float).
- z Any nonzero value implies a rigid motion in this direction (float).
- dum Any dummy value (float).

If the element is a joint (eltyp = 119, 120, 121, 122, 123, 124, 125, or 127), rigid (eltyp = 65, 66, 67, 68, 69, 70, 71, 73, 74, or 76), or constraint function element (eltyp = 77, 78, 79, 177, 178, 179), the method for enforcing the associated constraints (see [CONSTRAINTS](#)) and any applicable mass property can be specified as follows.

MID	CONMAT	[CONSTRAINT_METHOD]	[MASS	DENSITY	[GEOPARA]]
-----	--------	---------------------	-------	---------	------------

- MID The material id number from element attribute table.
- CONMAT Keyword indicating that the following data entries specify the method chosen for enforcing the constraints associated with the joint, rigid, or constraint function elements sharing the material id number **MID** (characters).
- CONSTRAINT_METHOD Optional parameter which specifies the method for enforcing the constraints associated with the joint, rigid, or constraint function elements sharing the material id number **MID** (characters or characters and real). The default method is set in [CONSTRAINTS](#) and used whenever this entry is omitted.
- multipliers The Lagrange multiplier method.
- elimination The elimination method.
- penalty beta The penalty method. The real-valued parameter **beta** should be a large positive number, typically of the order of 10^8 (no default value is provided).
- augmented beta The augmented Lagrangian method. The real-valued parameter **beta** should be a large positive number, typically of the order of 10^8 (no default value is provided).
- MASS Optional keyword indicating that the rigid elements with eltyp = 65, 66, 70, 73, or 76 sharing the material id number **MID** are to be attributed an element level (lumped or consistent) mass matrix computed as in the case of flexible elements, using the following data entries (characters).
- DENSITY Optional parameter which specifies the density for the rigid elements with eltyp = 65, 66, 70, 73, or 76 sharing the material id number **MID** (real).

GEOPARA	Optional parameter which specifies the cross sectional area for those rigid truss (eltyp = 65) and beam (eltyp = 66) elements, or the thickness for those rigid shell (eltyp = 73 or 76) elements sharing the material id number MID (real).
---------	---

If the element is a nonlinear spring (eltyp = 201 or 202), the definition of its material properties can be specified as follows.

MID	SPRINGMAT	k
-----	-----------	---

MID	The material id number from element attribute table.
SPRINGMAT	Keyword indicating that the following data entry specifies the stiffness constant of the nonlinear spring elements sharing the material id number MID (characters).
k	Stiffness constant of the nonlinear spring elements sharing the material id number MID (real).

If the element is a joint spring combination (eltyp = 220, 221, 222, 223, 225 or 227), the definition of its material properties and the method for enforcing the associated constraints (see [CONSTRAINTS](#)) can be specified as follows.

MID	CONMAT	[CONSTRAINT_METHOD]	SPRINGMAT	k1	k2	k3
-----	--------	---------------------	-----------	----	----	----

MID	The material id number from element attribute table.
CONMAT	Keyword indicating that the following data entries specify the method chosen for enforcing the constraints associated with the joint spring combination elements sharing the material id number MID (characters).
CONSTRAINT_METHOD	Optional parameter which specifies the method for enforcing the constraints associated with the joint spring combination elements sharing the material id number MID (characters or characters and real). The default method is set in CONSTRAINTS and used whenever this entry is omitted.
multipliers	The Lagrange multiplier method.
elimination	The elimination method.
penalty beta	The penalty method. The real-valued parameter beta should be a large positive number, typically of the order of 10^8 (no default value is provided).
augmented beta	The augmented Lagrangian method. The real-valued parameter beta should be a large positive number, typically of the order of 10^8 (no default value is provided).
SPRINGMAT	Keyword indicating that the following data entry specifies the stiffness constant of the joint spring combination elements sharing the material id number MID (characters).

k3 k1 k2 Stiffness constants of the joint spring combination elements sharing the material id number **MID** (real). More specifically k1 is the stiffness of the first embedded spring, k2 that of the second embedded spring when applicable, and k3 that of the third embedded spring when applicable.

If the element is a revolute joint-with-driver element (**eltyp** = 126), or a revolute joint-with-actuator element (**eltyp** = 226), or a prismatic joint-with-driver element (**eltyp** = 134), or a prismatic joint-with-actuator element (**eltyp** = 234), its description includes the prescription of the *relative* rotation (**eltyp** = 126), applied moment (**eltyp** = 226), *relative* translation (**eltyp** = 134), applied force (**eltyp** = 234), in the form $g(t) = \text{SCALE_FACTOR} * f(t) + \text{SHIFT}$ (**eltyp** = 126, 134, 226, 234), where **SCALE_FACTOR** is an amplification factor and $f(t)$ is a time-dependent function governed by **FUNCTION_TYPE** and up to four parameters a , b , c , and d . The properties of these four elements can be specified as follows.

MID	CONMAT	[CONSTRAINT_METHOD]	FUNCTION_TYPE	SCALE_FACTOR SHIFT	[a]	[b]	c	d
SPRINGMAT		[k]						

MID	The material id number from element attribute table.
CONMAT	Keyword indicating that the following data entries specify the method chosen for enforcing items below associated with the revolute or prismatic joint-with-driver or joint-with-actuator material id number MID (characters).
CONSTRAINT_METHOD	Optional parameter which specifies the method for enforcing the constraints associated with joint-with-driver or joint-with-actuator elements sharing the material id number MID (character). The default method is set in CONSTRAINTS and used whenever this entry is omitted.
multipliers	The Lagrange multiplier method.
elimination	The elimination method.
penalty	The penalty method. The real-valued parameter beta should be a large positive number (no default value is provided).
augmented beta	The augmented Lagrangian method. The real-valued parameter beta should be a large order of 10⁸ (no default value is provided).
FUNCTION_TYPE	Type of the relative rotation/applied moment/relative translation/applied force prescribed with-driver/revolute joint-with-actuator/prismatic joint-with-driver/prismatic joint-with material id number MID (characters). There are five types to choose from:
SINE	Sets $\theta(t) / M(t) / u(t) / F(t)$ to a sinusoidal relative rotation/applied moment/relative translation/applied force partially described by $f(t) = \sin(at + b)$, where [a] and [b] are two parameters specified below.
RAMP	Sets $\theta(t) / M(t) / u(t) / F(t)$ to a time-dependent relative rotation/applied moment/relative translation/applied force partially described by a bounded ramp function of the form $f(t) = 0$ for $t \leq a$, $f(t) = \frac{t-a}{b-a}$ for $a \leq t \leq b$, and $f(t) = 1$ for $t \geq b$, where [a] and [b] are two parameters specified below.

TRIA Sets $\theta(t)$ / $M(t)$ / $u(t)$ / $F(t)$ to a time-dependent relative rotation/applied moment/r

partially described by a triangular (hat) function of the form

$$f(t) = 0 \text{ for } t \leq a, \quad f(t) = \frac{t-a}{b-a} \text{ for } a \leq t \leq b, \quad f(t) = \frac{c-t}{c-b} \text{ for } b \leq t \leq c, \quad f(t) = 0 \text{ for } t \geq c$$

where a , b and c are three parameters specified below.

TRAP Sets $\theta(t)/M(t)$ / $u(t)$ / $F(t)$ to a time-dependent relative rotation/applied moment/re

partially described by a trapezoidal function of the form

$$f(t) = 0 \text{ for } t \leq a, \quad f(t) = \frac{t-a}{b-a} \text{ for } a \leq t \leq b, \quad f(t) = 1 \text{ for } b \leq t \leq c, \quad f(t) = \frac{d-t}{d-c} \text{ for } c \leq t \leq d, \quad f(t) = 0 \text{ for } t \geq d$$

where a , b , c and d are four parameters specified below.

SSHA Sets $\theta(t) / M(t)$ / $u(t)$ / $F(t)$ to a time-dependent relative rotation/applied moment/r

partially described by an S-shaped function of the form

$$f(t) = 0 \text{ for } t \leq a, \quad f(t) = \frac{(t-a)^2}{2(b-a)^2} \text{ for } a \leq t \leq b, \quad f(t) = 1 - \frac{(c-t)^2}{2(c-b)^2} \text{ for } b \leq t \leq c, \quad f(t) = 0 \text{ for } t \geq c$$

where a , b and c are three parameters specified below.

Constant amplification factor (real).

SCALE_FACTOR

SHIFT

a

Constant shift (real).

b

Time parameter (real).

c

Time parameter (real).

d

Time parameter (real).

SPRINGMAT

k

Keyword indicating that the following data entry specifies the stiffness constant of the with-actuator elements sharing the material id number **MID** (characters).

Stiffness constant of the revolute or prismatic joint-with-actuator elements sharing the

If the element is a thermal element (**eltyp** = 3, 10, 46-51, 53, 56-58, 81-86, 4646), the material properties are defined as follows.

MID	THRMAT	A	ρ	c_p	h/ ϵ	σ	k	t	P	Tr
-----	--------	---	--------	-------	---------------	----------	---	---	---	----

MID

The material id number from element attribute table.

THRMAT

keyword that specifies that the following data entries are associated with thermal elements.

A

Cross sectional area for lineal thermal elements (float).

ρ

Mass density per unit volume (float).

c_p

Specific heat coefficient (float).

h/ ϵ

Heat convection coefficient for a heat convection or bulk fluid thermal element, or transfer factor for a heat radiation element (float). When radiation is exchanged between two bodies Ω_1 (identified here by this value of **MID**) and Ω_2

(any other body), the transfer factor \mathcal{F}_{1-2} depends on the emittances of both bodies as well as the geometrical view. In the special case of a gray object in a large environment — that is, when Ω_1 represents a smaller body and Ω_2 a larger isothermal environment (for example, the atmosphere at some temperature) — \mathcal{F}_{1-2} becomes the emissivity ϵ_1 of the first body. In this case, $\mathcal{F}_{1-2} = \epsilon_1 = 1$ if Ω_1 is furthermore a black body.

 σ

Stefan's constant, also known as the Stefan-Boltzmann constant (in SI units, $\sigma = 5.670400 \times 10^{-8} \text{ Js}^{-1} \text{ m}^{-2} \text{ K}^{-4}$) (float).

k

Heat conduction coefficient (float).

t

Thickness of a (two-dimensional) thermal element (float).

P

Perimeter/circumference area for thermal elements (float), or depth of the boundary where convection (radiation) occurs when element type 47 (56) is used to model non-lateral boundary convection (radiation) (float).

Tr

For a heat radiation element, reference temperature of the enclosure receiving the radiation (float).

If the element is an acoustic element (eltyp = 31–45, 63, 90, 93–96, 98–108), the material properties may be defined as follows.

MID	AMAT	c	ρ
-----	------	---	--------

or

MID	AMAT	c_R	c_I	ρ
-----	------	-------	-------	--------

or

MID	AMAT	c	ρ	pml_type	γ	pmlx1	pmlx2	pmly1	pmly2	pmlz1	pmlz2
-----	------	---	--------	----------	----------	-------	-------	-------	-------	-------	-------

or

MID	AMAT	c_R	c_I	ρ	pml_type	γ	pmlx1	pmlx2	pmly1	pmly2	pmlz1
pmlz2											

MID The material id number from element attribute table.

AMAT Keyword that specifies that the following data entries are associated with acoustic fluid elements.

c Speed of sound in the material identified by MID (float).

c_R Real part of the speed of sound in the material identified by MID (float).

c_I Imaginary part of the speed of sound in the material identified by MID (float).

ρ Density (mass per unit volume) of the material identified by MID (float).

pml_type

Type of PML (Perfectly Matched Layer) (integer).

pml_type = 1 designates a box PML. In this case the PML elements are assumed to be in the region defined by $pmlx1 < x < pmlx2$, $-pmlx2 < x < -pmlx1$, $pmly1 < y < pmly2$, $-pmly2 < y < -pmly1$, $pmlz1 < z < pmlz2$, and $-pmlz2 < z < -pmlz1$.

pml_type = 2 designates a spherical PML. In this case the PML elements are assumed to be in the region defined by $pmlx1 < r < pmlx2$, where $r = \sqrt{x^2 + y^2 + z^2}$.

pml_type = 3 designates a cylindrical PML. In this case the PML elements are assumed to be in the region defined by $pmlx1 < r < pmlx2$, where $r = \sqrt{x^2 + y^2}$, $pmlz1 < z < pmlz2$, and $-pmlz2 < z < -pmlz1$.

γ

PML attenuation parameter (real). Given the speed of sound c specified above, the circular frequency $\omega = 2\pi f$ where f is the frequency specified in

IMPEDANCE — and therefore the wavenumber $k = \frac{\omega}{c}$ — the thickness of the PML layer t , and this attenuation parameter **γ**, the PML damps the outgoing waves by the factor $e^{-ktC(\gamma)}$ where $C(\gamma) = (1 - \frac{1}{\gamma}) \frac{\epsilon\gamma}{\gamma} + \frac{1}{\gamma^2} - \frac{1}{2}$. The

recommended value for **γ** is $1.5 \leq \gamma \leq 4.0$. Note that the mesh within the

PML must be designed so that it properly resolves the decaying solution. To this effect, also note that in practice, using two cubic elements through the thickness of the PML is sufficient to resolve a decaying solution characterized by a damping factor $e^{-ktC(\gamma)} = 0.001$ — that is, a solution that decays exponentially within the PML, from a relative amplitude of 1 to a relative amplitude of 0.001.

pmlx1

PML geometrical parameter (see above) (real).

pmlx2

PML geometrical parameter (see above) (real).

pmly1

PML geometrical parameter (see above) (real).

pmly2

PML geometrical parameter (see above) (real).

pmlz1

PML geometrical parameter (see above) (real).

pmlz2

PML geometrical parameter (see above) (real).

If the element is a fabric truss element (**eltyp** = 111), the material properties are defined as follows (see Powell, D.A. and Zohdi, T.I. Attachment mode performance of network-modeled ballistic fabric shielding. *Composites: Part B* 2009; 40: 451-460).

MID	FABMAT	type	E	ρ	A	U_c	U_f	λ	h	d	$\delta\lambda$	n_p	N_f
-----	--------	------	---	----------	---	-------	-------	----------	---	---	-----------------	-------	-------

MID

The material id number from element attribute table (integer).

FABMAT

Keyword specifying that the following data entries are associated with fabric elements (characters).

type

Type of fabric material (integer). If **type** = 1, the fabric properties are automatically determined by **AERO-S** using a micro-scale computation. If **type** = 2, they are determined using Gaussian distribution.

E	If type = 1, this is Young's modulus for the fibrils that make up the yarn (float). If type = 2, it is the mean value of Young's modulus for the yarn.
ρ	Mass density per unit volume of the fibril/yarn (float).
A	Cross sectional area of the entire yarn element (float).
U_c	If type = 1, this is the breaking stretch (l/l_0) of the fibrils (float). If type = 2, it is the stretch level at which the yarn begins to damage.
U_f	Stretch level at which the entire yarn has failed (i.e. the stress response is approximately zero) (float).
λ	If type = 1, this is the initial guess for the parameter controlling the damage rate of the yarn; its actual value is positive and determined by Newton's method (float). If type = 2, it is the slope of the assumed linear variation of the damage parameter with Young's modulus.
h	If type = 1, this is the length of an unstretched yarn (initial length of the truss element) (float). If type = 2, it is the value of the damage parameter for a zero Young's modulus.
d	If type = 1, this is the standard deviation for the inclination distance of the fibrils (misalignment of the fibrils) (float). If type = 2, it is the standard deviation associated with the assumed Gaussian distribution of Young's modulus.
$\delta\lambda$	If type = 1, this parameter is set to 0 and ignored (float). If type = 2, it is the standard deviation associated with the assumed Gaussian distribution of the damage parameter.
n_p	If type = 1, this is the maximum number of Newton iterations for fitting the damage parameter λ to micro-scale data (integer). If type = 2, this parameter is ignored.
N_f	If type = 1, this is the number of fibers in a typical yarn (integer). If type = 2, this parameter is ignored.

Next: [LOAD](#), Previous: [MATERIAL](#)

64 LINEAR MULTIPONT CONSTRAINTS

Command Statement:	LMPC
--------------------	-------------

The LMPC command statement is used to specify a set of linear multipoint constraint equations of the form

$$\sum_{j=1}^{j=n} c_{ij} u_j = r_i, \quad i = 1, 2, \dots$$

in the structural model. There is no limitation on the number of multipoint constraints, or number of degrees of freedom related by the same constraint equation.

The format of this command statement is as follows.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: Whereas the constraints specified under this command must be linear, the degrees of freedom u_j can be governed by nonlinear equations of equilibrium. Therefore, this command is also supported for nonlinear analysis using the command [NONLINEAR](#).

Note 3: The Lagrange multiplier method for enforcing the constraints associated with this command is supported only by the FETI-DP family of solvers, the GMRES solver, and the SPOOLES and MUMPS direct sparse solvers with pivoting enabled, and in all but explicit dynamic analyses (see [STATICS](#)).

Note 4: For time-dependent problems, the specified initial conditions must verify the specified linear multipoint constraints.

LMPC

```
CONSTRAINT# RHS  CONSTRAINT_METHOD
NODE#      DOF# COEFF
.
.
NODE#      DOF# COEFF
```

This corresponds to the constraint equation number i (integer).

CONSTRAINT#

RHS

This is the right-hand side r_i of the i -th constraint equation (float).

CONSTRAINT_METHOD

This is the method for enforcing the constraint (characters). The default method is set in [CONSTRAINTS](#) and used whenever this entry is omitted.

The Lagrange multiplier method.

multipliers

The elimination method.

elimination

penalty
[beta]

The penalty method. The parameter `beta` should be a large positive number, typically of the order of 10^8 (no default value is provided).

NODE#

This is the number of the node contributing the coefficient c_{ij} of the i -th constraint equation (integer).

DOF#

This is the local number of the degree of freedom at the node specified above contributing the coefficient c_{ij} of the i -th constraint equation (integer).

COEFF

This is the coefficient c_{ij} of the i -th constraint equation (float).

Next: [LOADCASE](#), Previous: [LMPC](#)

65 LOAD

Command Statement:	LOAD
--------------------	-------------

The **LOAD** command statement is used to inform **AERO-S** where the user defined subroutines for user defined forces and/or displacements and/or control are located. An example input file using the **LOAD** command can be found in FEM.d/fem\examples/Control.d

LOAD pathandfilename

pathandfilename Specifies **between quotes " "** the path and filename of the **LOAD** file. The extension of this file must be ".so".

Next: [LUMPED](#), Previous: [LOAD](#)

66 LOADCASE DEFINITION

Command Statement:	LOADCASE [LOADCASE_ID]
--------------------	--

The **LOADCASE** command can be used to define one or more "load" cases. Each "load" case is defined as a linear combination of "load" sets. Defining no "load" case is equivalent to defining a load case 0 which combines the "load" set 0 and all "loads" generated by the commands which do not support the "load" set construct (**LOADSET_ID**). Note also that any "load" generated by a command which does not support the "load" set construct (**LOADSET_ID**) is automatically added to every "load" case defined herein.

The input format of this command is given below.

LOADCASE [LOADCASE_ID]
--

LOADSET_ID_1 COEFF_1	
.	.
.	.
.	.
LOADSET_ID_n COEFF_n	

LOADCASE_ID

Optional non-negative integer which uniquely identifies a "load" case. The default value is 0. For static analysis, multiple "load" cases can be defined by repeating this command in the same input file using different values for **LOADCASE_ID** and different data. For dynamic analysis, only one "load" case can be defined and must be attributed the identifier 0.

LOADSET_ID_1

Integer identifying a "load" set.

COEFF_1

For static and dynamic analyses, this parameter is a real coefficient that can be used to amplify the "load" set identified by LOADSET_ID_1 (real). For dynamic analysis, this parameter can also consist of the keyword MFTT or HFTT followed by an integer identifying a TABLE_ID defined in [MFTT](#) or [HFTT](#). In this case, the amplification factor for the "load" set identified by LOADSET_ID_1 is the time-dependent function defined by MFTT[TABLE_ID] or HFTT[TABLE_ID].

Next: [MASS](#), Previous: [LOADCASE](#)

67 LUMPED

Command Statement:	LUMPED
--------------------	---------------

By default, **AERO-S** computes all element mass matrices and gravity (see [GRAVITY](#)) loads by a consistent approach, **except for explicit dynamic computations** (see [DYNAMICS](#)), in which case **AERO-S** always uses a lumped approach for this purpose. If a consistent mass matrix is not available for a particular element (see [TOPOLOGY](#)), then **AERO-S** uses in all cases a lumped mass matrix and gravity load for that element.

Alternatively, this command can be used to instruct **AERO-S** to compute all element mass matrices and gravity loads by a lumping method.

LUMPED

Next: [MATLAW](#), Previous: [LUMPED](#)

68 MASS EVALUATION

Command Statement:	MASS
--------------------	-------------

The **MASS** command statement is used to signal that the user would like that the total mass of the structural system be computed. The result is output on the screen by FEM. In addition to the total mass of the structure, the center of gravity (cg), the center of volume, and the closest node to the cg are also computed and printed on the screen.

Next: [MATUSAGE](#), Previous: [MASS](#)

69 MATERIAL LAW

Command Statement:	MATLAW
--------------------	---------------

The command **MATLAW** can be used to define one or several material laws, whether they are linear or nonlinear. However, because the command [MATERIAL](#) can be used for inputting the material properties of linear elastic elements, **MATLAW** (together with [MATUSAGE](#)) is expected to be used primarily for specifying nonlinear material laws. Typically, the user will write his/her own software describing a material law and package it in a library with a ".so" extension. The command **MATLAW** is then used to label this material law and pass to its library the parameters it expects. The user should

also grab the special makefile and include files for this command which are located within the AERO-S.d/Matlaw.d directory.

The command `MATLAW` can also be used to specify material laws that have been predefined and implemented in **AERO-S**.

Note 1: In the absence of the command [NONLINEAR](#) in the input file, a linear elastic material is attributed to all elements even when otherwise specified under this command, and its properties are set to those specified under the command [MATERIAL](#).

Note 2: In the presence of the command [NONLINEAR](#) in the input file, the material laws and properties specified under `MATLAW` and [MATUSAGE](#) take precedence over those inputted under [MATERIAL](#) if a conflict arises.

Note 3: In the presence of the command [NONLINEAR](#) but absence of the command `MATLAW` in the input file, linear elasticity with infinitesimal strains is assigned as the default material law to beam and shell elements as the nonlinear geometric effects of these elements are accounted for using the corotational method, and the Saint Venant-Kirchhoff hyperelastic law is assigned as the default material law to 3D solid elements as their nonlinear geometric effects are accounted for using the total Lagrangian method.

MATLAW

READ	material_name	pathandfilename						
MATERIAL_LABEL	MATERIAL_NAME		PARA#1	...	PARA#N			
MATERIAL_LABEL	Linear	ρ	E	ν				
MATERIAL_LABEL	StVenantKirchhoff	ρ	E	ν				
MATERIAL_LABEL	HenckyElastic	ρ	E	ν				
MATERIAL_LABEL	BilinearPlastic	ρ	E_1	ν	E_2	σ_{yield}	β	
MATERIAL_LABEL	FiniteStrainPlastic	ρ	E_1	ν	E_2	σ_{yield}	β	
MATERIAL_LABEL	LogStrainPlastic	ρ	E_1	ν	E_2	σ_{yield}	β	
MATERIAL_LABEL	NeoHookean	ρ	E	ν				
MATERIAL_LABEL	MooneyRivlin	ρ	c_1	c_2	c			
MATERIAL_LABEL	HypoElastic	E	ν	ρ				

MATERIAL_LABEL	J2Plasticity	E_1	ν	ρ	σ_{yield}	K	H	ϵ
MATERIAL_LABEL	KK1	E_1	ν	ρ	σ_{yield}	K	H	ϵ
MATERIAL_LABEL	KK2	E_1	ν	ρ	ϵ			

READ

Sub-command keyword used to introduce a material name (characters) and associate it with a library to be found at "pathandfilename.so" (characters) and linked with **AERO-S**. The path and filename of the material law library file must be specified between quotes (""). The extension of this file must be .so.

MATERIAL_LABEL

The material law id number (integer).

MATERIAL_NAME

Name of the material law (characters).

PARA#1

First parameter expected by the library pathandfilename.so.

PARA#N

Last parameter expected by the library pathandfilename.so.

Linear

Name of the predefined usual linear material law.

StVenantKirchhoff

Name of a predefined finite strain hyperelastic material law (appropriate for metals and moderate strains).

HenckyElastic

Name of another predefined finite strain hyperelastic material law (appropriate for larger strains).

BilinearPlastic

Name of the predefined infinitesimal strain bilinear plastic material law featuring kinematic hardening.

FiniteStrainPlastic

Name of a predefined finite strain plastic material law featuring kinematic hardening.

LogStrainPlastic

Name of another predefined finite strain plastic material law featuring logarithmic strains and kinematic hardening.

NeoHookean

Name of the predefined compressible version of the finite strain hyperelastic material law.

MooneyRivlin

Name of the predefined compressible version of the finite strain Mooney-Rivlin hyperelastic material law.

HypoElastic

Name of the predefined plane stress version of the hypoelastic material law.

J2Plasticity

Name of the predefined plane stress version of the von Mises plasticity material law based on the second stress invariant featuring kinematic and/or isotropic hardening.

KK1

Name of the predefined plane stress plasticity material law due to Korkolis and Kydiakides featuring kinematic and/or isotropic hardening and an advanced yield function including deformation-induced anisotropy.

KK2

Name of the predefined plane stress plasticity material law for aluminum Al-6260-T4 due to Korkolis and Kydiakides featuring an advanced yield function including deformation-induced anisotropy and an experimental-based stress-strain curve.

 ρ Mass density per unit volume (real). E

Young's Modulus (real).

 ν

Poisson's ratio (real).

 E_1

Young's Modulus for the bilinear and finite strain plastic material laws (real).

E_2	Second slope for the bilinear and finite strain plastic material laws (real).
σ_{yield}	Yield stress for the bilinear, finite strain, J2, and KK1 plastic material laws (real).
β	Free-parameter of the hardening model (real): $\beta = 0$ corresponds to pure kinematic hardening, $\beta = 1$ to pure isotropic hardening, and $0 \leq \beta \leq 1$ to a combination of kinematic and isotropic hardening.
c_1, c_2, c	Coefficients of the compressible Mooney-Rivlin model recalled below (real). $\Psi(J, I_1, I_2) = c(J - 1)^2 - 2(c_1 + 2c_2)\ln J + c_1(I_1 - 3) + c_2(I_2 - 3)$
K	Isotropic hardening modulus for the J2 plasticity and KK1 material laws (real).
H	Kinematic hardening modulus for the J2 plasticity and KK1 material laws (real).
ϵ	Relative tolerance for the convergence of the nonlinear material solver at each material point (real). The default value is 10^{-6} .

Next: [MODE](#), Previous: [MATLAW](#)

70 MATERIAL LAW USAGE

Command Statement:	MATUSAGE
--------------------	-----------------

The MATUSAGE command statement can be used to assign a material law specified in the MATLAW command to one or several elements. The input format of this command is

MATUSAGE

ELEMENT#	MATLAW_LABEL#
----------	---------------

or

STARTING_ELEMENT#	ENDING_ELEMENT#	MATLAW_LABEL#
-------------------	-----------------	---------------

ELEMENT#	Element number whose material law number is to be specified (integer).
MATLAW_LABEL	Material law identification number (integer).
MATERIAL_LABEL	First element of a sequence of elements that have the same MATLAW_LABEL (integer).

MATERIAL_NAME

Last element of a sequence of elements that have the same MATLAW_LABEL (integer).

Next: [NODALCONTACT](#) Previous: [MATUSAGE](#)

71 SAVING EIGENMODES OR PROJECTING ONTO EIGENVECTORS

Command Statement:	MODE
--------------------	-------------

The MODE command is used to signal that: (a) when the [EIGEN](#) command is used, the eigen solutions are to be saved in a binary file named EIGENMODES (or binary files EIGENMODES# when using **AERO-S** in distributed memory mode with one file per MPI process) and located in the execution path of **AERO-S**, (b) when the [DYNAMICS](#) command is used, the binary file named EIGENMODES (or binary files EIGENMODES# when using **AERO-S** in distributed memory mode with one file per MPI process) located in the execution path of **AERO-S** is read, and the projections of the transient solution onto the eigen modes found in that file are computed (and outputted if requested in the [OUTPUT](#) command), and (c) when the [IMPEDANCE](#) command is used with FREQSWEEP, PADEPOLES and ALG = PadeLanczos, the binary file named EIGENMODES (or binary files EIGENMODES# when using **AERO-S** in distributed memory mode with one file per MPI process) located in the execution path of **AERO-S** is read, and the eigen modes stored in this (or these) file(s) are exploited to identify within a range of interest the eigenvalues missed by a previous eigen computation and their corresponding eigenvectors.

Note 1: The EIGENMODES file is written using the “internal” numbering of **AERO-S** for the degrees of freedom. This internal numbering depends on the equation solver and renumbering scheme that were specified under the [STATICS](#) and [RENUMBERING](#) commands, respectively, when the eigen solutions were computed. Hence, when reusing the EIGENMODES file for computing the projections of the transient solution of a problem onto the vectors contained in this file, the same equation solver and renumbering scheme must be specified under the [STATICS](#) and [RENUMBERING](#) commands, respectively.

Next: [NFRAMES](#), Previous: [MODE](#)

72 NODAL CONTACT

Command Statement:	NODALCONTACT
--------------------	---------------------

The NODALCONTACT command can be used to specify node-to-node contact of the form

$$(u_2 - u_1) \cdot n_{12} = 0$$

or

$$(u_2 - u_1) \cdot n_{12} \geq gap$$

where u denotes the displacement field, n_{12} denotes the normal to the contact surface oriented from node 1 to node 2, and gap denotes the initial gap. The input format of this command can be as follows.

Note 1: For `mode_v = 1` and `mode_v = 3`, the enforcement of the nodal contact constraints by the Lagrange multiplier method is supported only by the FETI-DP family of solvers, and for static and implicit dynamic analyses only.

Note 2: For `mode_v = 1` and `mode_v = 3`, the enforcement of the nodal contact constraints by the elimination method is not supported.

Note 3: For `mode_v = 0` and `mode_v = 2`, the enforcement of the nodal tied contact constraints by the Lagrange multiplier method is supported only by the FETI-DP family of solvers, the GMRES solver, and the SPOOLES and MUMPS direct sparse solvers with pivoting enabled, and in all but explicit dynamic analyses.

NODALCONTACT

or

NODALCONTACT	MODE	default_mode_v
--------------	------	----------------

followed by one or more lines of the form

NODE_1	NODE_2	N_X	N_Y	N_Z
--------	--------	-----	-----	-----

or

NODE_1	NODE_2	N_X	N_Y	N_Z	GAP	gap_v
--------	--------	-----	-----	-----	-----	-------

NODE_1	NODE_2	N_X	N_Y	N_Z	MODE	mode_v
--------	--------	-----	-----	-----	------	--------

NODE_1	NODE_2	N_X	N_Y	N_Z	MODE	mode_v	GAP	gap_v
--------	--------	-----	-----	-----	------	--------	-----	-------

NODE_1	NODE_2	N_X	N_Y	N_Z	MODE	mode_v	GAP	gap_v
CONSTRAINT_METHOD								

NODE_1	Id number of the first node (integer).
NODE_2	Id number of the second node (integer).
N_X	x-component of the normal to the contact surface oriented from NODE_1 to NODE_2 (float).
N_Y	y-component of the normal to the contact surface oriented from NODE_1 to NODE_2 (float).
N_Z	z-component of the normal to the contact surface oriented from NODE_1 to NODE_2 (float).
GAP	Keyword to be spelled out (characters).
gap_v	Initial gap between NODE_1 and NODE_2. A negative gap means an initial penetration (float).
MODE	Keyword to be spelled out (characters).
mode_v	Specifying mode_v = 0 designates a tie-of-the-normal-component-of-the-displacement constraint. Specifying mode_v = 1 designates a normal contact constraint. Specifying mode_v = 2 designates a tie-of-the-normal-and-tangential-components-of-displacement constraint. Specifying mode_v = 3 designates both normal contact and tie-of-the-tangential-components-of-displacement constraints (integer).
default_mode_v	Default mode_v for all constraints. If not specified, the default mode is set to 1, i.e. normal contact (integer).
CONSTRAINT_METHOD	This is the method for enforcing the associated constraints (characters). The default method is set in CONSTRAINTS and used whenever this entry is omitted.
multipliers	The Lagrange multiplier method.
elimination	The elimination method.
penalty [beta]	The penalty method. The parameter beta should be a large positive number, typically of the order of 10^8 (no default value is provided).

Next: [NODES](#), Previous: [NODALCONTACT](#)

73 NODAL FRAMES

Command Statement:	NFRAMES
--------------------	----------------

The NFRAMES command statement can be used to specify nodal reference frames to which any node defined in [NODES](#) can refer to either for the purpose of specifying in which frame the nodal coordinates are inputted, or in which frame the nodal degrees of freedom are defined, or both.

Note 1: All nodal degrees of freedom referred to in the [ACTUATORS](#), [DISPLACEMENTS](#), [DIMASS](#), [FORCES](#), [IDISPLACEMENTS](#), [IDISP6](#), [IDISP6PITA](#), [IVELOCITIES](#), [IVEL6PITA](#), [LMPC](#), [SENSORS](#), [USDD](#) and [USDF](#) commands are defined in the corresponding nodal degree of freedom frames (see [NODES](#)) which by default are the same as the global reference frame. However, all computed directional results (for example, the x, y and z components of a displacement field or the xy component of a stress field) can be outputted in either the nodal reference frames, or the global reference frame (see [OUTPUT](#)).

Note 2: Currently, nodal degree of freedom frames are not supported for [NONLINEAR](#) analysis.

Two input formats are available for this command: one for the case of a position frame, and one for the case of a degree of freedom frame. The total number of lines below this command statement should be equal to the total number of different nodal frames referenced in [NODES](#), whether they are position frames or degree of freedom frames.

NFRAMES

NFRAME#	x_O	y_O	z_O	$S_{1x} \ S_{1y} \ S_{1z}$	$S_{2x} \ S_{2y} \ S_{2z}$
	$S_{3x} \ S_{3y} \ S_{3z}$				

or

NFRAMES

NFRAME#	$S_{1x} \ S_{1y} \ S_{1z}$	$S_{2x} \ S_{2y} \ S_{2z}$	$S_{3x} \ S_{3y} \ S_{3z}$

NFRAME# ID number of a nodal position (if 13 entries are inputted on the same line) or degree of freedom (if 10 entries are inputted on the same line) reference frame (integer).

x_O x coordinate of the origin of the nodal position frame identified by the ID number specified above (float).

y_O y coordinate of the origin of the nodal position frame identified by the ID number specified above (float).

z_O z coordinate of the origin of the nodal position frame identified by the ID number specified above (float).

$S_{1x} \ S_{1y} \ S_{1z}$ First axis of the nodal reference frame identified by the ID number specified above, **the nodal x-axis**, expressed in the global reference frame (floats).

$S_{2x} \ S_{2y} \ S_{2z}$ Second axis of the nodal reference frame identified by the ID number specified above, **the nodal y-axis**, expressed in the global reference frame (floats).

$S_{3x} \ S_{3y} \ S_{3z}$ Third axis of the nodal reference frame identified above, **the nodal z-axis**, expressed in the global reference frame (floats).

Next: [NONINPC](#), Previous: [NFRAMES](#)

74 NODES *S*

Command Statement:	NODES
--------------------	--------------

The **NODES** command statement is used to signal that the following data lines correspond to the coordinates, position frame, and degree of freedom frame of each node. The input data format is given below. There should be as many lines as the number of nodes in the mesh.

Note 1: All nodal degrees of freedom referred to in the [ACTUATORS](#), [DISPLACEMENTS](#), [DIMASS](#), [FORCES](#), [IDISPLACEMENTS](#), [IDISP6](#), [IDISP6PITA](#), [IVELOCITIES](#), [IVEL6PITA](#), [LMPC](#), [SENSORS](#), [USDD](#), and [USDF](#) commands are defined in the corresponding nodal degree of freedom frames which by default are the same as the global reference frame. However, all computed directional results (for example, the x, y and z components of a displacement field or the xy component of a stress field) can be outputted in either the nodal reference frames, or the global reference frame (see [OUTPUT](#)).

NODES

NODE#	X-ORDINATE	Y-ORDINATE	Z-ORDINATE	POS_FRM	DOF_FRM
-------	------------	------------	------------	---------	---------

NODE#	ID number of a node (integer).
X-ORDINATE	x-ordinate of the node identified by the specified node ID number (float).
Y-ORDINATE	y-ordinate of the node identified by the specified node ID number (float).
Z-ORDINATE	z-ordinate of the node identified by the specified node ID number (float).
POS_FRM	ID number of the nodal position frame (see NFRAMES) in which the coordinates of the node identified by the node ID number specified above are defined (integer). The value 0, which is the default value, specifies that the nodal position frame is the global frame. (See NFRAMES).
DOF_FRM	ID number of the nodal degree of freedom frame (see NFRAMES) in which the degrees of freedom of the node identified by the node ID number specified above are defined (integer). The value 0, which is the default value, specifies that the nodal degree of freedom frame is the global frame. (See NFRAMES).

Next: [NONLINEAR](#), Previous: [NODES](#)

75 NON INTRUSIVE POLYNOMIAL CHAOS

Command Statement:	NONINPC
--------------------	----------------

The **NONINPC** command signals to the **AERO-S** code to perform a non intrusive uncertainty quantification analysis using the Polynomial Chaos (PC) representation. It requires that the input file also contains the [GROUPS](#), [RANDOM](#), and [STATICS](#) commands as well as their respective data. Essentially, **AERO-S** computes in this case the mean and standard deviation of a computed primal field v as follows

$$\bar{v} = \int_{-\infty}^{+\infty} vp(v)dv = v_0 = \frac{\sum_{i=1}^N v^{(i)}}{N}$$

$$\sigma_v = \sqrt{\int_{-\infty}^{+\infty} (v - \bar{v})^2 p(v)dv} = \sqrt{\sum_{i=1}^{P-1} \left(\int_{-\infty}^{+\infty} \psi_i^2 p(\psi_i(\{\xi_j\}))d\psi_i \right) v_i^2}$$

$$= \sqrt{\sum_{i=1}^{P-1} \left(\int_{-\infty}^{+\infty} \psi_i^2 p(\psi_i(\{\xi_j\}))d\psi_i \right) \left(\frac{\sum_{j=1}^N v^{(j)} \psi_i^{(j)}}{N \int_{-\infty}^{+\infty} \psi_i^2 p(\psi_i)d\psi_i} \right)^2}$$

where v_0 denotes the constant of the PC expansion of the non deterministic v field ("the output"), N denotes the number of realizations, $v^{(i)}$ denotes the deterministic solution of the field v associated with the i-th realization of the random system, p denotes the probability density function, ξ_j denotes the j-th "basic" random variable which is assumed to be Gaussian, v_i denotes the deterministic coefficient of the i-th PC shape function ψ_i , P denotes the total number of PC shape functions ψ_i representing the non deterministic output, and $\psi_i^{(j)}$ denotes the evaluation of ψ_i at the j-th realization of the random system.

Note 1: Currently, this command is limited to the post-processing of the results of a linear, static, structural analysis.

The input format of this command is given below.

NONINPC

DEG_OUT	NUM_RLZ
---------	---------

DEG_OUT Degree of the PC representation of the output which, together with the data of the [RANDOM](#) command, determines P .

NUM_RLZ Number of realizations N of the random system (which implies the number of deterministic solutions) (integer).

Next: [OUTPUT](#) Previous: [NONINPC](#)

76 NON LINEAR ANALYSIS

Command Statement:	NONLINEAR
--------------------	------------------

The command **NONLINEAR** signals to **AERO-S** that the current static or dynamic analysis to be performed is a nonlinear one. For structural problems, if this command is present in the input file and a nonlinear material law is not specified for an element using **MATUSAGE** and **MATLAW**, a geometrically nonlinear formulation of the internal force vector and stiffness matrix is activated using a default material law. The geometrically nonlinear formulation and the default material law are element-dependent. For beam and shell elements, the geometric nonlinearities are accounted for using the corotational formulation and the default material law is linear elasticity with infinitesimal strains. For 3D solid elements, the geometric nonlinearities are accounted for using the total Lagrangian method and the default material law is the Saint Venant-Kirchhoff hyperelastic law. Also for structural problems, if the command **NONLINEAR** is present in the input file and a nonlinear material law is specified for an element using **MATUSAGE** and **MATLAW**, this law overrides the default material law and the geometric nonlinearities are accounted for as in the previous case.

Note 1: In the absence of this command in the input file, all elements are assigned linear elasticity as the material law and the kinematics are assumed to be linear, even when otherwise specified.

Note 2: Currently, all 3D solid, beam, shell and rigid elements can be used for geometrically nonlinear explicit dynamic analyses.

Note 3: Currently, all 3D solid, beam, shell and rigid elements except the BT shell element (type 16) can be used for geometrically nonlinear static and implicit dynamic analyses.

Note 4: Currently, all 3D solid elements, the andes shell element (type 15) and the BT shell element (type 16) can be used for materially nonlinear explicit dynamic analyses.

Note 5: Currently, all 3D solid elements and the andes shell element (type 15) can be used for materially nonlinear static and implicit dynamic analyses.

The input format is given below.

NONLINEAR

METHOD	(keyword)
PARAMETERS	(keyword or keyword and values)

METHOD

arclength

This sub-command keyword selects the arclength method for solving the nonlinear equations associated with a nonlinear static computation. If this option is not selected, then the default algorithm for solving the nonlinear equations is a standard Newton-Raphson iteration method (characters).

PARAMETERS

MAXITR maxitr_v

For a nonlinear static or an implicit dynamic computation, this sub-command keyword (characters) followed by its integer value specifies the maximum number of Newton-like iterations to be performed (per time-step). The default value is 100.

NLTOL nltol1_v
nltol2_v nltol3_v nltol4_v

For a nonlinear static or implicit dynamic computation, this sub-command keyword (characters) and the four real values following it specify the various tolerances needed for assessing the convergence of the Newton-Raphson method. More specifically, the first two parameters `nltol1_v` and `nltol2_v` specify the tolerances for the convergence of the relative residual and solution increment, respectively (real, real). Their default values are 10^{-6} and infinity, respectively. The third and fourth parameters `nltol3_v` and `nltol4_v` specify the tolerances for the convergence of the absolute residual and solution increment, respectively (real, real). Their default values are zero and infinity, respectively. The convergence of the Newton-Raphson method is declared when *either* of the following occurs (a) both of the relative residual and relative increment are below their specified respective tolerance levels, or (b) both of the absolute residual and absolute increment are below their specified respective tolerance levels. Note that a tolerance can be set to infinity using the characters "INF", and to machine epsilon 10^{-16} using the characters "EPS".

DLAMBDA real_1
real_2

This sub-command keyword (characters) followed by two real numbers specifies the external "loading" parameters for a nonlinear *static* analysis only. The first real number, `real_1`, specifies the load fraction increment, and the second real number, `real_2`, specifies the total load factor. For example, "dlambda 0.25 1.0" specifies applying the external load in 4 steps and using for this purpose a load increment equal to 0.25*(Original Inputted Source Terms) at each load-step. On the other hand, "dlambda 0.25 2.0" specifies magnifying the external load by a factor , then applying this external load in 8 steps with a load increment equal to 0.25*(Original Inputted Source Term). This incremental loading strategy can help the convergence of the Newton-Raphson method.

FITALG fitalg_v

For structural problems, this sub-command keyword (characters) followed by an integer value specifies the fitting algorithm for the corotational formulation. For `fitalg = 1` (default value) the tangent stiffness matrix is not consistent but the corotational framework is most robust. For `fitalg = 2`, the tangent stiffness matrix is consistent (integer).

FAILSAFE real_1

For a nonlinear implicit dynamic computation, this keyword activates a fail-safe computational strategy when the Newton solver fails to converge *and* the absolute residual is less than the tolerance specified in `real_1` (real). The fail-safe strategy consists in repeating a time-step computation during which the aforementioned events occur using half the value of the time-step, until the Newton-Raphson method converges. Upon convergence, the value of the time-step is doubled in each subsequent dynamic step computation until the original value of the time-step is restored. The default value of `real_1` is machine precision.

unsymmetric

The corotational formulation adopted by **AERO-S** generates an unsymmetric geometric tangent stiffness matrix. By default, this matrix is symmetrized. If this default mode of operation is not desirable, this sub-command keyword can be used to specify that the geometric tangent stiffness matrix of the cororational formulation is to remain unsymmetric. In that case, an unsymmetric equation solver must be specified in [STATICS](#) (characters).

PENALTY int_1
real_1 real_2

This sub-command keyword (characters) followed by one integer and two real numbers specifies the parameters of the penalty or augmented Lagrangian method for enforcing [CONSTRAINTS](#) in a nonlinear analysis. The integer `int_1` is the maximum number of outer iterations on the penalty parameter in the case of the penalty method, or on the Lagrange multipliers (and optionally the penalty parameter) in the case of the augmented Lagrangian method; its default value is 1. The first real number, `real_1`, is the absolute tolerance for assessing the convergence of the outer iteration loop based on the infinite norm of the constraint violations; its default value is 1e-8. The second real number, `real_2`, is the factor by which the penalty parameter in either method is increased at every outer iteration; its default value is 10.

REBUILD int_1
[int_2]

This sub-command keyword (characters) followed by two integers manages two different intervals for deciding when to rebuild the tangent "stiffness" matrix during the solution of a nonlinear problem by the Newton-Raphson method. The integer `int_1` specifies the iteration interval within a time-step of a dynamic analysis, or a load-step of a static analysis where `DLAMBDA` is used to apply the external load incrementally. For example, `int_1 = 1` specifies rebuilding the tangent stiffness matrix at every iteration, and `int_1 = 2` specifies rebuilding it at every second iteration. The default value for `int_1` is 1. Usage of the integer `int_2` is optional, and its default value is 1. Otherwise, `int_2` specifies the time-step or load-step interval. For example, `int_2 = 1` specifies every time-step or load-step, and `int_1 = 2` specifies every other time-step or load-step. The default value of `int_2` is also 1. Hence, setting `int_1 = 1` and `int_2 = 1` specifies rebuilding the tangent stiffness matrix at every Newton iteration of every time-step or load-step. Setting `int_1 = 2` and `int_2 = 3` specifies rebuilding the tangent stiffness matrix at every second iteration of every third time-step or load-step. The characters "MAX" can be used to set either or both of `int_1` and `int_2` to a very large interval to limit or effectively deactivate rebuilding the tangent stiffness matrix, respectively. An interesting intermediate case is given by the combination `int_1 = "MAX"` and `int_2 = 1` which specifies rebuilding the tangent stiffness matrix only at the start of every time-step or load-step.

Next: [OUTPUT6](#), Previous: [NONLINEAR](#)

77 OUTPUT OF RESULTS (OUTPUT completely spelled out)

Command Statement:	OUTPUT	[KEYLETTER]
--------------------	---------------	-------------

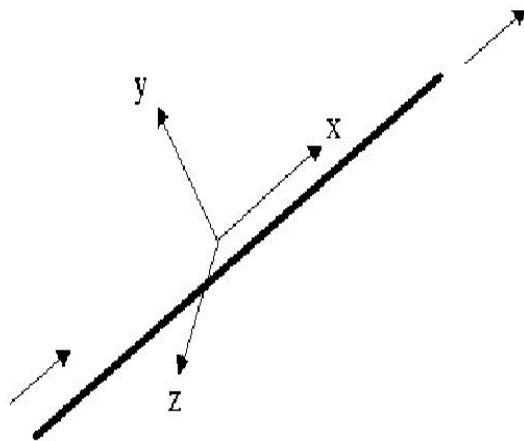
The `OUTPUT` command statement is used to signal that the following lines of data correspond to specifying which results are to be outputted. For vector results such as displacement fields, this command forces **AERO-S** to output only the three translational components.

The shared memory version of **AERO-S** generates ASCII output files. Most of these are in a format that is suitable for postprocessing by the **XPost** software, but some are in a format that is suitable for postprocessing by **gnuplot**. The distributed memory version of **AERO-S** generates binary output files except for the case of selective nodal output which it generates in ASCII format. The **SOWER** software can be used to convert the binary output files into ASCII output files, in a format suitable for postprocessing by **XPost**. All "nodal" output files are generated in the **gnuplot** format. Examples using the various `OUTPUT` commands can be found in `FEM.d/fem\examples/Output.d/`.

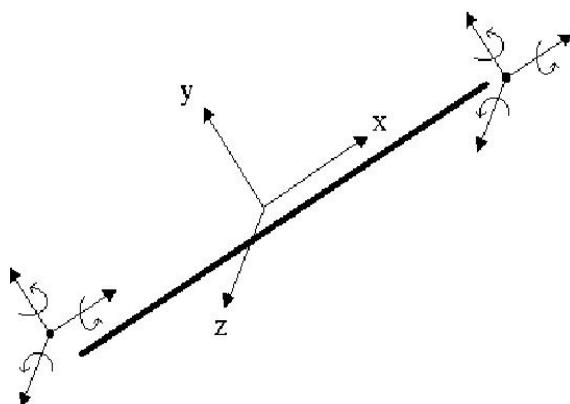
Note 1: In addition to the time-instances implied by the specified value of the parameter INCREMENT, any result specified for output is also outputted at the beginning of a simulation if INCREMENT is non zero, and at the end of a simulation if **AERO-S** exits gracefully.

Note 2: The strain and stress outputs are not available for the bar and beam elements, except for STRAINXX and STRESSXX

Note 3: The internal forces and moments and STRAINXX and STRESSXX for the bar and beam elements are outputted in the element frames using the sign conventions graphically depicted below.



Sign conventions for the output of the internal forces in a bar element



Sign conventions for the output of the internal forces in a beam element

Note 3: For nonlinear structural analysis, the stresses and strains for the corotational elements are outputted on the deformed configuration and in the corotated frames.

Note 4: For nonlinear structural analysis, the stresses and strains for the total Lagrangian elements are outputted on the undeformed configuration.

Note 5: For nonlinear structural analysis, the internal forces and moments and **STRAINXX** and **STRESSXX** for the bar and beam corotational elements are outputted in the element frames associated with the deformed configuration.

Note 6: When multiple outputs are expected, they are printed one set after the other in the same output file.

Note 7: To visualize the directions of principal strains and stresses, the auxiliary code PDIREC should be used to generate the node set and element set used by **XPost** for this purpose. When running PDIREC, the user is asked to enter the name of the mesh **XPost** file (for example, the one obtained from executing **AERO-S** with the option **-t**), a **AERO-S** result file, the type of that result for labeling purposes (eg. **SP1DIREC**), and a characteristic length to be used for scaling the length of the elements representing the directions to be visualized. PDIREC can also be used for visualizing any other vector output (eg. **GDISPLAC** or **GVELOCIT**).

Note 8: The vocabulary used for the request of stresses and strains that are associated with the elements listed below differs from the standard continuum mechanics notation. See below for specific notations.

Note 9: When both **OUTPUT** and **OUTPUT6** commands are used in the same input file, the **KEYLETTER** option specified for one of these two commands applies to all output files requested under both of them; if this option is specified twice, once for each of these two commands, and the two specified settings differ, the one which is specified last in the input file prevails.

The input format of this command can be as follows.

OUTPUT	[KEYLETTER]
---------------	--------------------

RESULT [...]	[FORMAT] [OPTION]	PATHANDFILENAME	INCREMENT	[NODE_NUM BER]	[OPTION]
------------------------	------------------------------------	------------------------	------------------	-----------------------	-----------------

or

RESULT [...]	[FORMAT] [OPTION]	PATHANDFILENAME	INCREMENT	[N NODE_NUMBER]	[OPTION]
------------------------	------------------------------------	------------------------	------------------	------------------------	-----------------

or

RESULT [...]	[FORMAT] [OPTION]	PATHANDFILENAME	INCREMENT	[NG GROUP_NUMBER]	[OPTION]
-----------------	----------------------	-----------------	-----------	-------------------	----------

KEYLETTER Optional keyletter specifying how to handle gaps in the numbering of nodes *as far as output is concerned*.

e In this case, the output is expanded — that is, padded with zero(es) at each line corresponding to a gap in the numbering of the nodes. This is the default case. If **XPost** is to be used for visualizing the results, the geometry of the **AERO-S** computational model should be converted in this case to the **XPost** format using the option **-t** or **-m** to perform the same node padding (see [INTRODUCTION](#)).

c In this case, no padding is performed, and the output file is ordered according to the ascending order of the freely numbered nodes. If **XPost** is to be used for visualizing the results, the geometry of the **AERO-S** computational model should be converted in this case to the **XPost** format using the option **-T** or **-M** to avoid any node padding (see [INTRODUCTION](#)).

In summary, the user should note that **AERO-S** accepts gaps in the number of elements and nodes (free numbering) and performs all computations by accounting for such gaps efficiently. The visualizer **XPost** can also handle freely numbered nodes and elements; however, for speed purposes, **XPost** interprets a result as a node-gap-free result. For this reason, if a **AERO-S** computational model contains gaps in the numbering of the nodes, using **XPost** for visualizing the computed results requires performing one of the following:

- Setting KEYLETTER to e and converting the geometry of the **AERO-S** computational model to the **XPost** format using the option **-t** or **-m** (see [INTRODUCTION](#)) — in which case the output files will be larger than needed.
- Setting KEYLETTER to c and converting the geometry of the **AERO-S** computational model to the **XPost** format using the option **-T** or **-M** (see [INTRODUCTION](#)) — in which case the output files will not be larger than needed but a different node numbering (more specifically, a compressed version) will be used in the **XPost** model.

RESULT	The following results can be outputted:
GDISPLAC	Generalized nodal displacements.
DISPLACX	Nodal displacements in the x direction.
DISPLACY	Nodal displacements in the y direction.
DISPLACZ	Nodal displacements in the z direction.
ROTATIOX	Nodal rotations in the x direction.
ROTATIOY	Nodal rotations in the y direction.
ROTATIOZ	Nodal rotations in the z direction.
DISPLMOD	Euclidean norms of the nodal displacements.
ROTATMOD	Euclidean norms of the nodal rotations.
GDISPMOD	Euclidean norms of the generalized nodal displacements.
GVELOCIT	Nodal velocities.
GACCELER	Nodal accelerations.
ROMATRIX	Rotation matrix representation of nodal rotations (9 components).

QUATERNI	Positive unit quaternion representation of nodal rotations (4 components).
GTEMPERA	Nodal temperatures.
GTEMPVEL	Nodal first time-derivatives of the temperature.
GEIGENPA	Eigenvalues and Eigenvectors. If singular modes are found during an eigen analysis, they are saved in the file FNAME associated with GEIGENPA <i>before</i> the non-singular eigen results.
GEIGSLSH	Sloshing Eigenvalues and Eigenvectors (displacement potential). If a singular mode is found during the analysis, it is saved in the file FNAME associated with GEIGSLSH <i>before</i> the non-singular eigen results.
SLSHDISP	True fluid nodal displacements associated with the computed sloshing modes.
STRAINXX	\underline{xx} component of the (total) strain tensor.
STRAINYY	\underline{yy} component of the (total) strain tensor.
STRAINZZ	\underline{zz} component of the (total) strain tensor.
STRAINXY	\underline{xy} component of the (total) strain tensor.
STRAINYZ	\underline{yz} component of the (total) strain tensor.
STRAINZX	\underline{zx} component of the (total) strain tensor.
STRAINVM	von Mises strain.
EFFPSTRN	Effective plastic strain.
STRAINP1	First principal strain.
STRAINP2	Second principal strain.
STRAINP3	Third principal strain.
PLSTRNXX	$\underline{\underline{xx}}$ component of the plastic strain tensor.
PLSTRNYY	$\underline{\underline{yy}}$ component of the plastic strain tensor.
PLSTRNZZ	$\underline{\underline{zz}}$ component of the plastic strain tensor.
PLSTRNXY	$\underline{\underline{xy}}$ component of the plastic strain tensor.
PLSTRNYZ	$\underline{\underline{yz}}$ component of the plastic strain tensor.
PLSTRNZX	$\underline{\underline{zx}}$ component of the plastic strain tensor.
EP1DIREC	Direction of first principal strain.
EP2DIREC	Direction of second principal strain.
EP3DIREC	Direction of third principal strain.
STRESSXX	$\underline{\underline{xx}}$ component of the second-Piola stress tensor.
STRESSYY	$\underline{\underline{yy}}$ component of the second-Piola stress tensor.
STRESSZZ	$\underline{\underline{zz}}$ component of the second Piola stress tensor.
STRESSXY	$\underline{\underline{xy}}$ component of the second Piola stress tensor.
STRESSYZ	$\underline{\underline{yz}}$ component of the second Piola stress tensor.
STRESSZX	$\underline{\underline{zx}}$ component of the second Piola stress tensor.
STRESSVM	(Second Piola) von Mises stress.
STRESSP1	(Second Piola) first principal stress.
STRESSP2	(Second Piola) second principal stress.
STRESSP3	(Second Piola) third principal stress.
SP1DIREC	Direction of first principal stress.

SP2DIRC	Direction of second principal stress.
SP3DIRC	Direction of third principal stress.
BKSTRSXZ	\underline{xx} component of the back stress tensor.
BKSTRSYZ	\underline{yy} component of the back stress tensor.
BKSTRSZZ	\underline{zz} component of the back stress tensor.
BKSTRSXY	\underline{xy} component of the back stress tensor.
BKSTRSYZ	\underline{yz} component of the back stress tensor.
BKSTRSZX	\underline{xz} component of the back stress tensor.
HEATFLXX	Temperature flux along the x direction.
HEATFLXY	Temperature flux along the y direction.
HEATFLXZ	Temperature flux along the z direction.
GRDTEMPX	Temperature gradient along the x direction.
GRDTEMPY	Temperature gradient along the y direction.
GRDTEMPZ	Temperature gradient along the z direction.
INXFORCE	Internal forces along the element x -axis.
INYFORCE	Internal forces along the element y -axis.
INZFORCE	Internal forces along the element z -axis.
AXMOMENT	Applied (internal) moments along the element x -axis.
AYMOMENT	Applied (internal) moments along the element y -axis.
AZMOMENT	Applied (internal) moments along the element z -axis.
ENERGIES	External, Aeroelastic, Elastic, Kinetic, Damping, and Numerical Production energies. The Numerical Production energy corresponds to the energy conservation error for a conservative system. When positive, it suggests a numerically unstable scheme. When negative, it suggests the presence of numerical damping in the scheme. Using the standard notation and that of the DYNAMICS section, these energies are computed as follows:

External :

$$\sum_{k=0}^{n-1} [(1 - \gamma) F^{\text{ext}^k} + \gamma F^{\text{ext}^{k+1}}]^T (u^{k+1} - u^k)$$

Aeroelastic :

$$\sum_{k=0}^{n-1} [(1 - \gamma) F^{\text{aer}^k} + \gamma F^{\text{aer}^{k+1}}]^T (u^{k+1} - u^k)$$

Elastic :

$$u^n^T K u^n$$

Kinetic :

$$\dot{u}^n^T M \dot{u}^n$$

Damping :

$$\sum_{k=0}^{n-1} [(1 - \gamma) C \dot{u}^k + \gamma C \dot{u}^{k+1}]^T (u^{k+1} - u^k)$$

Numerical Production : Kinetic - Previous_Kinetic + Elastic - Previous_Elastic + Increment_Damping_Between_Previous_and_Current - Increment_Exterior_Between_Previous_and_Current

AEROFORX	Aeroelastic nodal forces along the x direction at $[t^{n+1}]$, after application of the boundary conditions (see DYNAMICS).
AEROFORY	Aeroelastic nodal forces along the y direction at $[t^{n+1}]$, after application of the boundary conditions (see DYNAMICS).
AEROFORZ	Aeroelastic nodal forces along the z direction at $[t^{n+1}]$, after application of the boundary conditions (see DYNAMICS).
AEROMOMX	Aeroelastic nodal moments along the x direction at $[t^{n+1}]$, after application of the boundary conditions (see DYNAMICS).
AEROMOMY	Aeroelastic nodal moments along the y direction at $[t^{n+1}]$, after application of the boundary conditions (see DYNAMICS).
AEROMOMZ	Aeroelastic nodal moments along the z direction at $[t^{n+1}]$, after application of the boundary conditions (see DYNAMICS).
RAEROFOR	Aeroelastic resultant forces along the x , y , and z directions (integrated over the wet surface of the structure) at $[t^{n+1}]$, before application of the boundary conditions (see DYNAMICS).
RAEROTFL	Fluid heat fluxes transferred to the structure at $[t^{n+1}]$.
GHELMHOL	Real or complex acoustic pressure scattered field computed by an acoustic analysis in the frequency domain. The real part is always output first, then, if it is non uniformly zero, the imaginary part is output as a separate frame but in the same output file.
GACOUPRE	Nodal values of the acoustic pressure computed by an acoustic analysis in the time domain.
GACOUVEL	Nodal values of the first time-derivative of the acoustic pressure computed by an acoustic analysis in the time domain.
GACOUACC	Nodal values of the second time-derivative of the acoustic pressure computed by an acoustic analysis in the time domain.
KIRCHHOFF	Acoustic pressure field at locations specified using the KIRLOC command and evaluated using the Kirchhoff integral. This result can be outputted only if the surface of the scatterer is defined using the command HSCB . The corresponding output file contains on each line five words of the form: X-ORDINATE_I Y-ORDINATE_I Z-ORDINATE_I REAL(U_I) IMAG(U_I). X, Y, and Z-ORDINATE_I are the X, Y, and Z coordinates of the I-th point specified using the command KIRLOC . REAL(U_I) and IMAG(U_I) are the real and imaginary parts of the acoustic pressure field computed at the I-th specified point using the Kirchhoff integral method.
FARFIELD	Farfield pattern of the acoustic pressure field. This field can be output only if the HSCB command is also specified. In this case, the output file contains on each line four words of the form " α , β , real-part(ffd), imaginary-part(ffd)". The first two quantities are the spherical angles (expressed in radians) that determine the direction $d = [\cos \beta * \cos \alpha, \cos \beta * \sin \alpha, \sin \beta]$. The number of output lines depends on the parameter INCREMENT (see below).
GENCOORD	Projections using a mass-based metric of a transient solution on the eigen modes saved by the MODE command in the file EIGENMODES. This field can be output only if the MODE command is also specified.

MODERROR	Relative truncation error using the two-norm of the representation of a transient solution by its projection on the eigen modes saved by the MODE command in the file EIGENMODES. This field can be output only if the MODE command is also specified.
MODALDSP	Modal coefficients of the displacements, when dynamics computations are done in a modal basis.
MODALEXF	Modal coefficients of the external forces, when dynamics computations are done in a modal basis.
CONPRESS	Contact pressure forces (Lagrange multipliers in the normal directions).
CONFACE	Denotes the status of interactions at a node. A value of 0.5 indicates the node is not in contact. A value of 1, 2, or 3 denotes the number of interactions at that node.
NORMAL_FORCE_MAG	Normal force magnitude at a node. If multiple constraints exist at a node, the value is for the last constraint.
NORMAL_TRACTION_MAG	Normal traction magnitude at a node. If multiple constraints exist at a node, the value is for the last constraint.
TANGENTIAL_FORCE_MAG	Tangential force magnitude at a node. If multiple constraints exist at a node, the value is for the last constraint.
TANGENTIAL_TRACTION_MAG	Tangential traction magnitude at a node. If multiple constraints exist at a node, the value is for the last constraint.
CDIRNORX	x component of the normal direction for the constraint at a node. If multiple constraints exist at a node, the value is for the last constraint.
CDIRNORY	y component of the normal direction for the constraint at a node. If multiple constraints exist at a node, the value is for the last constraint.
CDIRNORZ	z component of the normal direction for the constraint at a node. If multiple constraints exist at a node, the value is for the last constraint.
CDIRTANX	x component of the tangential direction for the constraint at a node. If multiple constraints exist at a node, the value is for the last constraint.
CDIRTANY	y component of the tangential direction for the constraint at a node. If multiple constraints exist at a node, the value is for the last constraint.
CDIRTANZ	z component of the tangential direction for the constraint at a node. If multiple constraints exist at a node, the value is for the last constraint.
SLIP_MAG	Incremental slip at a node. If multiple constraints exist at a node, the value is for the last constraint.
NODAL_DISSIPATION	Frictional energy dissipated at a node.
CONTACT_AREA	Contact area for a node.
GAP_CUR	Normal gap for the current time. If multiple constraints exist at a node, the value is for the last constraint.
GAP_OLD	Normal gap from the previous time. If multiple constraints exist at a node, the value is for the last constraint.
REACTION	Reaction forces and moments along all three orthogonal directions.
HEATREAC	Thermal reaction sources.
STATEVCT	Snapshots of the state vector (displacements).
VELOVCT	Snapshots of the velocity vector (velocities).
ACCELVCT	Snapshots of the vector of accelerations.
FRCNGVCT	Snapshots of the vector of external forces.
FDIFFVCT	Snapshots of the difference between the vector of internal forces and that of external forces.
ROBDAF	File containing the reduced-order basis generated by the command ROBC .

SAMPLMSH	File containing the sampled mesh needed for performing a hyper reduction, in the form of a list of elements and their corresponding hyper reduction coefficients (see ATTRIBUTES). The format of this file is designed to allow its inclusion in the AERO-S input file using the INCLUDE command (see INTRODUCTION).
FORMAT	This optional field can be used to override the default format for outputting a numerical value by one which consists of 2 integer numbers F and P separated by a blank (integers). F specifies the field width — that is, the total number of digits to be printed including the decimal point, the "E" of the exponent and the sign of the exponent, but excluding the sign of the output value. P is the number of digits after the decimal and before the exponent. For example, 8 2 results in printing the output value 1523 as 0.15E+03.
FNAME	User specified filename associated with result (characters).
INCREMENT	Output increment for transient and eigenvalue results (integer). For a FARFIELD output result, this field specifies, in the three-dimensional case, the number of longitudinal directions where the farfield is to be evaluated. In this case, $(INCREMENT/2)+1$ latitudinal directions are also considered and therefore the farfield is evaluated at $((INCREMENT/2)+1)*INCREMENT$ points uniformly distributed in spherical coordinates. In two-dimensions, the farfield is evaluated at INCREMENT points that are evenly distributed on a circle. For transient and quasi-static analyses, the final time-instance is always outputted for the requested fields — except for the aeroelastic forces and moments — independently of the value of INCREMENT.
N	Sub-command keyword (character) signaling that the next entry is a node identifying number.
NODE_NUMBER	When this optional field is used, with or without the sub-command keyword N, only the computed results associated with the node NODE_NUMBER are outputted in the same format as otherwise, except that the header of the XPost file is omitted for easier use of a visualization software such as gnuplot (integer).
NG	Sub-command keyword (characters) signaling that the next entry is a group identifying number (see GROUPS).
GROUP_NUMBER	When this optional field is used together with the sub-command keyword NG, only the computed results associated with the nodes belonging to the group of nodes GROUP_NUMBER (see GROUPS) are outputted in the following format which is suitable for a visualization software such as gnuplot (integer).
FLOAT-1	
	NODE_NUMBER-1 X Y Z RESULT-1 RESULT-2 ... RESULT-n
	NODE_NUMBER-2 X Y Z RESULT-1 RESULT-2 ... RESULT-n
	...
FLOAT-2	
	NODE_NUMBER-1 X Y Z RESULT-1 RESULT-2 ... RESULT-n
	NODE_NUMBER-2 X Y Z RESULT-1 RESULT-2 ... RESULT-n
	...

where FLOAT-i is an *i*-th time-instance in the case of a dynamic analysis or

a circular frequency value (number of cycles per second) in the case of a frequency sweep analysis (see [IMPEDANCE](#)), NODE_NUMBER-j is the j -th

node number of group GROUP_NUMBER, X, Y and Z are the x , y , and z

coordinates of NODE_NUMBER-j, and RESULT-1 RESULT-2 ... RESULT-n are all the computed results pertaining to NODE_NUMBER-j and the chosen instance of RESULT.

OPTION
Any of the following optional fields can be specified after the previous optional and/or non-optional entries documented above:

NDTYPE [NUM_OUT_RLZ]
NDTYPE is an optional parameter that can take one of the following values: MEAN, STDV, or PDF. It should be used only when an intrusive or non intrusive non deterministic analysis is performed. When MEAN is selected and a non intrusive non deterministic analysis is performed, the mean value of the RESULT is outputted using the number of realizations NUM_RLZ specified in the NONINPC command. When STDV is chosen and a non intrusive non deterministic analysis is performed, the standard deviation of the RESULT is outputted using the number of realizations NUM_RLZ specified in the NONINPC command. When PDF is specified, the probability density function is outputted using a number of realizations equal to the integer set in the suboptional field NUM_OUT_RLZ (integer). First, the shape functions are evaluated at NUM_OUT_RLZ realizations of the random system. Then, NUM_OUT_RLZ realizations of the non deterministic RESULT are reconstructed from these evaluations using

$$v^{(j)} = \sum_{i=0}^{P-1} v_i^{(j)} \psi_i^{(j)}$$
 in the case of a displacement or $\sigma^{(j)} = \sum_{i=0}^{P-1} \sigma_i^{(j)} \psi_i^{(j)}$ in

the case of a stress field σ and written in the output file (note that the case of a von Mises stress is treated slightly differently because it is a nonlinear function of the stress tensor components). A histogram can be constructed from these results.

STR_AVG_OPTION
If the output of a stress or strain field is requested, the user can set this optional field to one of the following values NODALFULL, NODALPARTIAL, ELEMENTAL, or GAUSS (characters) to specify a type of stress/strain averaging. If NODALFULL is specified, the stress or strain is averaged at each node using all elements attached to this node. In this case, the contributions of the beam and bar elements are first transformed to the global frame. If NODALPARTIAL is specified, the averaging is performed using only the elements that are neither of a bar or a beam type. If ELEMENTAL is specified, the stress or strain is output without any averaging at each node of each element. Note however that currently, the ELEMENTAL option is not supported when a FETI solver is invoked. If GAUSS is specified, the stress or strain is outputted without any averaging at each Gauss point of each element.

STR_SHELL_OPTION
This optional field is applicable only when the output of a stress/strain field of a shell element is requested. It can take any of the following values UPPER, MEDIAN, LOWER (characters). If UPPER is specified, the stresses are computed on the upper surface of the element. If MEDIAN is specified (default value), they are computed on the median surface of the element. If LOWER is specified, they are computed on the lower surface of the element.

STR_BEAM_OPTION
This optional field is applicable only when the output of STRAINXX or STRESSXX of a beam element is requested. It specifies two real numbers (real). The first (second) real number represents the positive or negative y (z) coordinate in percentage of the maximum positive (top fiber) or negative (bottom fiber) y (z) coordinate of the beam cross section, in the local frame

COMPLEX_OUTPUT_OPTION
[NSTEPS]

attached to the beam, of the fiber at which ϵ_{xx} or σ_{xx} is requested. The default value for both real numbers is zero.

These optional and suboptional fields are applicable only when the output is complex-valued. **COMPLEX_OUTPUT_OPTION** can take any of the following values **REALIMAG**, **MOPHASE**, **ANIM** (characters). When set to **REALIMAG**, the real parts of the complex-valued output are printed first and the imaginary parts are printed next. When set to **MOPHASE**, the moduli ρ of the output values are first

printed and then the phases θ are printed. When set to **ANIM**, the values of $\rho \cos(\theta - \phi_1)$ are printed first, followed by the values of $\rho \cos(\theta - \phi_2)$, etc, concluding with the values of $\rho \cos(\theta - \phi_N)$, where ϕ_i is defined as $\frac{2\pi}{N}$ and

N is specified in the suboptional field **NSTEPS** (integer).

FRM_OPTION

This optional parameter can be used to request outputting directional results (for example, the x , y and z components of a computed displacement field or the xy component of a stress field) in the nodal frame defined at each node of the finite element model, or in the global frame. In the former case, this parameter should be set to **NODFRA**, and in the latter case, to **GLOFRA**.

ROTVEC_OPTION

This optional parameter can be used to select the specific type of rotation vector output for **GDISPLAC** (only under [OUTPUT6](#)), **ROTATIOX**, **ROTATIOY**, **ROTATIOZ**, **ROTATMOD**, and **GDISPMOD**. It can take values **Euler** (default value), **Complement**, **Linear**, **ReducedEulerRodrigues**, **CayleyGibbsRodrigues**, **WienerMilenkovic**, or **BauchauTrainelli**. The default **Euler** rotation vector, also known as the axis-angle representation, is a vectorial parameterization of the rotation group whose magnitude is the (scalar) rotation angle θ and rotation axis is the unit 3-vector \hat{e} — that is, $\Psi = \theta\hat{e}$. This is not to be confused with non-vectorial Euler angles for the parameterization of rotation. The complement rotation vector is specifically the complement of the Euler rotation vector and corresponds to a rotation angle of $(2\pi - \theta)$ about the

axis $-\hat{e}$. For details about the other options, refer to *The vectorial parameterization of rotation*, by Bauchau and Trainelli, in *Nonlinear Dynamics*, 32, No 1, pp. 71–92, 2003. Regardless of the value of **ROTVEC_OPTION**, the rotation vector is a representation of the "total rotation" — that is, it is measured with respect to the reference orientation (which corresponds in **AERO-S** to the case where the rotation matrix is the identity matrix).

RESCALING

FLAG Optional sub-command keyword followed by an **on** or **off** flag which can be used to define whether or not to rescale the rotation vector — which is a one-to-many mapping from the rotation matrix. It applies to **GDISPLAC**, **GVELOCIT**, and **GACCELER** under [OUTPUT6](#), and to **ROTATIOX**, **ROTATIOY**, **ROTATIOZ**, **ROTATMOD**, and **GDISPMOD**. The default value of **FLAG** is **On**. The specific interpretation of this option depends on the setting of **ROTVEC_OPTION**. For example, in the case of an Euler rotation vector, setting **FLAG** to **on** means that the outputted rotation vector is that with a rotation angle $\theta \leq \pi$.

When the rotation vector is rescaled, its components become discontinuous at the point of rescaling. Otherwise, its components may be discontinuous at certain singularity points ($\theta = 2\pi$ in the case of the Euler rotation

vector).

ANGULAR_OPTION

This optional parameter can be used to choose a definition of the angular velocity or acceleration for GVELOCIT or GACCELER under [OUTPUT6](#). It can take the following values: `Convected` (default), `Spatial`, and `Total`. The `Convected` definition refers to a moving frame that is computed by **AERO-S** and attached to each node. The definition `Spatial` refers to the fixed reference frame, and the definition `Total` refers to the time derivatives of the (Euler) rotation vector. In the case of `Total`, the `RESCALING` option may also be used to specify whether the angular velocity (acceleration) is the first (second) time derivative of the rescaled or non-rescaled Euler rotation vector. Note that angular velocities and accelerations inputted using `IVEL` or [USDD](#) should be `Convected`.

Next: [PITA](#), Previous: [OUTPUT](#)

78 OUTPUT OF RESULTS 6 COLUMNS (**OUTPUT6** completely spelled out)

Command Statement:	OUTPUT6	[KEYLETTER]
--------------------	----------------	-------------

The `OUTPUT6` command has the same options and format of the `OUTPUT` command. Its outcome differs from that of `OUTPUT` only when vector results such as generalized displacements fields are to be outputted. In that case, the `OUTPUT6` command forces **AERO-S** to output all six components of the generalized displacement field in a format slightly different from that of the file created by `OUTPUT`, in that the node number is included at the beginning of each line.

Note 1: When multiple outputs are expected, they are printed one set after the other in the same output file.

Note 2: When both `OUTPUT` and `OUTPUT6` commands are used in the same input file, the KEYLETTER option specified for one of these two commands applies to all output files requested under both of them; if this option is specified twice, once for each of these two commands, and the two specified settings differ, the one which is specified last in the input file prevails.

Next: [PRELOAD](#), Previous: [OUTPUT6](#)

79 PARALLEL-IN-TIME ALGORITHM (PITA)

Command Statement:	PITA
--------------------	-------------

The `PITA` command statement is used to request the parallelization in time of the time-integration algorithm specified under the `DYNAMIC` command. This option is supported only when executing the distributed version of the **AERO-S** executable. The simultaneous presence of the command statement `NONLINEAR` in the input file automatically triggers the nonlinear version of the `PITA` algorithm. By default, the initialization of the time-slices is performed by running the chosen time-integrator on the coarse time-grid. Alternatively, the `IDISP6PITA` (see [IDISP6PITA](#)) and `IVEL6PITA` (see [IVEL6PITA](#)) commands can be used to specify the seed displacements and velocities. The input format is given below.

Note 1: The current implementation allows only geometric nonlinearities.

Note 2: The foundations of the PITA methodology are described in: J. Cortial, *Time-Parallel Methods for Accelerating the Solution of Structural Dynamics Problem*, PhD Thesis, Stanford University, 2011.

[PITA]

J_RATIO	MAX_ITER
PITA_OPTIONS	

or

J_RATIO	MAX_ITER	MAX_SLICES_ON_CPU
PITA_OPTIONS		

J_RATIO Ratio between the time-steps on the fine and coarse time-grids. Also, number of time-steps in each time-slice (integer).

MAX_ITER Maximum number of outer PITA iterations (integer).

MAX_SLICES_ON_CPU Maximum number of time-slices assigned to a processor (integer). If the number of time-slices is larger than the number of available CPUs, the supernumerary time-slices are ignored. For optimal performance, there should be as many available CPUs as time-slices. The default value is 1.

PITA_OPTIONS None, one or several of the following optional statements

TIMEREVERSIBLE Instructs **AERO-S** to exploit the time-reversibility of the problem. In this case, each time-slice is divided into 2 halves, one treated by usual forward-in-time integration, and the other by backward-in-time integration. For optimal performance, 2 CPUs should be assigned to each time-slice.

ORTHOPROJ0L
orthoprojtol Sets the relative tolerance used to construct the orthogonal projectors for the correction step. The default value is 10^{-6} .

READINITSEED Instructs **AERO-S** to use the initial seed information provided via the commands [IDISP6PITA](#) and [IVEL6PITA](#). If additional seed values are required, they are obtained in this case via time-integration on the coarse time-grid.

REMOTE_COARSE Instructs **AERO-S** to perform the coarse time-grid integration on a dedicated CPU, which will not be available during the parallel integration on the fine-time grid (linear case only). For optimal performance in the linear case, it is recommended to use this option and provide one additional CPU. This command is ignored when only one CPU is available.

NOFORCE Informs **AERO-S** about the absence of any external force so that a special version of the PITA can be executed for optimal performance (linear case only).

JUMPCVG	jumpcvg	Instructs AERO-S to enable the jump-reduction-based convergence criterion with the specified tolerance jumpcvg (linear case only). The default value is J_RATIO ² .
GLOBALBUSES	integer	Determines the scheme to use for updating the global bases using the following convention: 1 = all seed vectors, 2 = all seed and propagated vectors (nonlinear case only). The default value is 1.
LOCALBUSES		Instructs AERO-S to use the local basis updating scheme instead of the global one (nonlinear case only). This option may improve performance but at the expense of time-accuracy. It is not implemented (and therefore ignored) for time-reversible simulations.
JUMPOUTPUT		Instructs AERO-S to output the magnitudes of the jumps.

Next: [PRESSURE](#), Previous: [PITA](#)

80 PRELOAD

Command Statement:	PRELOAD
--------------------	----------------

The PRELOAD command statement can be used to specify a preload for a truss or a membrane element. For a truss element, the preload is an axial force that affects its stiffness matrix. For a membrane element, the preload is a three-dimensional generalized force (or force per unit directional length) that also affects its element stiffness matrix. In both cases, the default value(s) of the the preload is (are) zero. It is assumed that the nodal coordinates of the model specified under [NODES](#) include the deformations due to preload.

Preload affects all of linear, nonlinear, static, transient, and eigen analyses. For a truss element, the outputted internal axial force includes the amount of preload. For a membrane element, the outputted stresses also include the amount of preload.

The following two formats are supported and can be mixed.

PRELOAD

ELEMENT#	PRELOAD_VALUE1	PRELOAD_VALUE2	PRELOAD_VALUE3
----------	----------------	----------------	----------------

STARTING_ELEMENT#	THRU	ENDING_ELEMENT#	PRELOAD_VALUE1	PRELOAD_VALUE2
PRELOAD_VALUE3				

ELEMENT#	Element number where a preload is to be specified (integer).
PRELOAD_VALUE1	For a truss element, value of the axial preloading force. For a membrane element, value of the preload generalized force (force per unit length) in the global <i>x</i> direction (float).
PRELOAD_VALUE2	For a membrane element, value of the preload generalized force (force per unit length) in the global <i>y</i> direction (float).

PRELOAD_VALUE3	For a membrane element, value of the preload generalized force (force per unit length) in the global z direction (float).
STARTING_ELEMENT#	First element of a sequence of elements where a specified preload value or set of values apply (integer).
THRU	Keyword indicating that the following entry is the last element of a sequence of elements (characters).
ENDING_ELEMENT#	Last element of a sequence of elements where a preload value or set of values apply (integer).

Next: [QSTATICs](#), Previous: [PRELOAD](#)

81 PRESSURE

Command Statement:	PRESSURE [LOADSET_ID]
--------------------	------------------------------

The PRESSURE command can be used to specify a piece-wise uniform pressure field, and/or a pressure field resulting from an air blast in the free field and whose parameters are specified under the [CONWEP](#) command. The specified pressure field can be applied to beam elements, membrane elements, shell elements, and surface elements defined in [SURFACETOPO](#). For a beam element (type 6 or 7), the pressure is applied in the Y-direction of the **element frame** specified in the [EFFRAMES](#) command or implied by the third node technique for beam elements (see [TOPOLOGY](#)).

For a membrane element (type 128, or 129), a shell element (type 8, 88, 15, 1515, 16, 20, 2020, 73, or 76), or a surface element defined in [SURFACETOPO](#) for the sole purpose of defining a surface computation that may be independent from any specific element type, the pressure field is applied normal to the surface of the element using the usual convention of a positive outward normal — that is, if the three nodes of a shell element are denoted by a , b , and c and numbered in this order, the normal is computed as $\vec{ab} \wedge \vec{ac}$. Furthermore, a positive value of PRESSURE_VALUE implies a force in the direction of the normal.

In a nonlinear analysis, the generated pressure forces are of the **follower** type for shell elements and surface elements defined in [SURFACETOPO](#); for beam elements they can either be of the follower type (default) or non-follower type, depending on a compilation switch that must be set for this purpose.

Note 1: AERO-S always generates pressure loads using a consistent method.

Note 2: The piece-wise uniform pressure field represented by the pressure value PRESSURE_VALUE can be varied in time using the command [MFTT](#).

Note 3: If the surface of interest is already discretized with beam or shell elements, using the ELEMENT# method of pressure input is more consistent with the remainder of the finite element analysis than using the SURFACE counterpart, because in this case the pressure loads are computed using the shape functions of the corresponding beam or shell elements.

Note 4: The blast pressure generated by CONWEP is currently not applicable to beam or membrane elements.

The following three formats are available for this command and can be mixed.

PRESSURE [LOADSET_ID]

ELEMENT#	PRESSURE_VALUE	[CONWEPSWITCH]
----------	----------------	----------------

STARTING_ELEMENT#	ENDING_ELEMENT#	PRESSURE_VALUE	[CONWEPSWITCH]
-------------------	-----------------	----------------	----------------

SURFACE	SURFACE#	PRESSURE_VALUE	[CONWEPSWITCH]
---------	----------	----------------	----------------

LOADSET_ID Optional non-negative integer which identifies explicitly the "load" set to which the source term generated by this command belongs to (integer). The default value is 0. Hence, the PRESSURE command can be repeated as many times as desired within the same input file using each time a different value for LOADSET_ID and different data. The [LOADCASE](#) command can refer to LOADSET_ID to define one or multiple "load" cases for static analysis (see the [STATICS](#) command and the explanation of its sub-command keyword CASES), and/or the "load" case for dynamic analysis.

ELEMENT# Element number where a pressure field is to be specified (integer).

PRESSURE_VALUE# Value of the piece-wise uniform pressure field in that element (float). If CONWEPSWITCH is set to On, this pressure value is added to the blast pressure generated by the CONWEP software module.

CONWEPSWITCH On/Off switch for the pressure load due to an air blast in the free field whose parameters are set in the [CONWEP](#) command (characters). If this switch is set to On, the pressure field computed by the software module CONWEP is added to that specified in PRESSURE_VALUE. Otherwise, only the pressure value specifies in PRESSURE_VALUE is used to generate the pressure load to be applied on the specified element or set of elements. The default value of this switch is On.

STARTING_ELEMENT# First element of a sequence of elements where the piece-wise uniform pressure field has the same value (integer).

ENDING_ELEMENT# Last element of a sequence of elements where the piece-wise uniform pressure field has the same value (integer).

SURFACE Keyword indicating that a surface defined in [SURFACETOPO](#) is to be identified next by its integer identification number (characters).

SURFACE# Integer identification of the surface defined in [SURFACETOPO](#) where the uniform pressure value PRESSURE_VALUE is to be applied (integer).

Next: [RANDOM](#), Previous: [PRESSURE](#)

82 QUASISTATICS ANALYSIS

Command Statement:	QSTATICS
--------------------	-----------------

The QSTATICS command statement is used to signal that the subsequent data lines correspond to the following quasistatics *iterative* algorithm for solving $Ax=b$, where A can represent a diagonal block of a coupled system and can be singular

$$\begin{aligned} A\tilde{x} &= b^{n+1} \\ \alpha^{n+1} &= \alpha^n + \beta R^T b^{n+1} \\ \bar{x}^{n+1} &= \bar{x}^n + \theta(\tilde{x} - \bar{x}^n) \\ x^{n+1} &= \bar{x}^{n+1} + R\alpha^{n+1} \end{aligned}$$

This command is currently implemented only for linear analysis, and is useful only for a multidisciplinary analysis (aerothermal, aeroelastic, aerothermoelastic).

For single discipline analysis, it is at best ($\theta = 1$) a direct solution method.

The second and fourth steps of the iterative above solution algorithm are currently enabled only for an aerothermal (thermostructure-thermofluid) analysis in which the thermostructure problem is singular. In that case, A represents the matrix of the thermostructure subsystem and R represents a basis of its null space.

Note 1: An equation solver must be specified under the STATICS command.

The input format of this command can be as follows.

QSTATICS

MECH	θ	tolqs	maxqs	delta
HEAT	θ	β	tolqs	maxqs

θ Underrelaxation factor $0 < \theta \leq 1$.
 β Underrelaxation factor $0 < \beta \leq 1$.

tolqs Convergence tolerance.
 maxqs Maximum number of iterations.

delta Time-step equivalent-value of the increment between any two iterations. This parameter is taken into account only in: (a) the presence of a user-defined "control.C" file (see [ACTUATORS](#), see [SENSORS](#), see [USDD](#), and see [USDF](#)), and/or (b) the time-stamp in the output format for quasi-static analysis. It is the equivalent

of a time-step and is used to convert the current iteration into current time. The default value is 0. When `delta` is specified to a non-zero value, θ is automatically set to $\theta = 1$ and convergence is not checked because in that case, convergence is reached in one iteration and it is understood that the purpose of the quasistatics analysis is to solve a series of consecutive static problems whose right-hand sides are set by a "control.C" file.

Next: [READMODE](#), Previous: [QSTATIC](#)

83 RANDOM

Command Statement:	RANDOM
--------------------	---------------

The `RANDOM` command statement is used to signal that some materials have properties with random values. All random material properties are assumed to have a Gaussian probability distribution.

Note 1 : Currently, an input file can specify only one random material property per group.

Note 2 : To ensure the positivity of the material properties at every realization, the following non-Gaussian model is used

$$\text{MATPROP} = \text{MEAN} + \frac{\text{STDV}}{\sqrt{2}}(\xi^2 - 1)$$

with the constraint $\frac{\text{STDV}}{\sqrt{2}} < \text{MEAN}$. Here ξ denotes a standard normal random variable.

The input format of this command is given below.

RANDOM

GROUP#	MATPROP	MEAN	STDV
--------	---------	------	------

GROUP# Id of the element group for which the random properties are to be specified (integer).

MATPROP# Material property for which the random properties are to be specified (string). Currently, this keyword can take any of the following values: `E` (Young's modulus), `A` (cross section of a bar or beam element), and `kx`, `ky`, and `kz` (linear spring coefficients (see [MATERIAL](#))).

MEAN Mean value of the non deterministic `MATPROP` (real). This value overwrites the deterministic value of `MATPROP` specified under the `MATERIAL` command for the elements in the corresponding `GROUP#`.

STDV Standard deviation of the non deterministic `MATPROP` (real).

Next: [RENUMBERING](#), Previous: [RANDOM](#)

84 READING EIGEN MODES

Command Statement:	READMODE
--------------------	-----------------

The READMODE command can be used for inputting a modal basis and modal initial conditions (see [IDISPLACEMENTS](#), [IVELOCITIES](#)) for dynamic computations (see [DYNAMICS](#)), or inputting a reduced-order basis (see [ROBC](#)), or performing a “Multi-Ping-Pong” analysis (see [AERO](#)).

READMODE pathandfilename NUBV

pathandfilename Name of the file containing the modes or reduced-order basis vectors to be read (characters). The format of this file is as follows. The first line contains the number of modes included in this file. The second line specifies the number of nodes of the underlying mesh. The modes follow afterwards. Each mode can be specified by a frequency (number of cycles per second) on a separate line and a number of lines equal to the number of nodes, each containing the 3 or 6 values of the mode for the 3 translational or all 6 degrees of freedom of that node. Alternatively, each mode can also be specified by a frequency (number of cycles per second) on a separate line and a number of lines equal to the number of nodes, each containing the node number followed by 3 or 6 values of the mode for the 3 translational or all 6 degrees of freedom of that node.

NUBV Number of consecutive modes or reduced-order basis vectors to be read in the above file (integer).

Next: [RESTART](#), Previous: [READMODE](#)

85 RENUMBERING

Command Statement:	RENUMBERING
--------------------	--------------------

The RENUMBERING command statement is used to specify the type of node renumbering to be performed on the mesh for either or both of the skyline and sparse solvers. The input format is given below.

RENUMBERING

TYPE

TYPE For FEM, the “rcm” and “sloan” schemes are available for the skyline solvers, and the “esmond” (minimal degree ordering) and “metis” (recursive spectral bisection ordering) are available for the (esmond) sparse solver. When a FETI solver is used, two renumbering schemes can be specified if a skyline as well as a sparse solver are to be used by this FETI solver. In this case, the two renumbering schemes can be specified in any order, each on a separate line (string).

Next: [TRBM](#), Previous: [RENUMBERING](#)

86 RESTART

Command Statement:	RESTART
--------------------	----------------

The RESTART command is used to request saving computational data in a RESTART file in order to enable later the restart of a dynamics simulation, and/or initializing a dynamics computation using computational results previously saved in a RESTART file (restart data). The processes of creating and reading from a RESTART file are specified on separate lines using different syntaxes.

Note 1: In addition to the time-instances implied by the specified value of the parameter INCREMENT, the restart data is also saved at the end of a simulation if **AERO-S** exits gracefully.

The syntax for invoking this command is given below.

PATHANDFILENAME1 INCREMENT
PATHANDFILENAME2 EXTENSION

INCREMENT	An integer number that specifies at every how many time-integration steps the “rcfem.restart” file will be updated. For an updating restart, this number specifies at every how many iterations the file “restart.upd” will be updated.
PATHANDFILENAME1 INCREMENT	This command, which can be combined with the following one, instructs AERO-S to save/overwrite the restart data into the specified file PATHANDFILENAME1 (string) every INCREMENT (integer) time-steps. An example using RESTART can be found in FEM.d/fem_examples/Restart.d/
PATHANDFILENAME2 EXTENSION	This command, which can be combined with the previous one, instructs AERO-S to read the restart data from the specified file PATHANDFILENAME2 (string) and append the extension EXTENSION (string) (e.g. “.2”) to all output filenames specified under the command OUTPUT. An example using RESTART can be found in FEM.d/fem_examples/Restart.d/

Next: [RBMFILTER](#), Previous: [RESTART](#)

87 RIGID BODY (AND OTHER ZERO ENERGY) MODES

Command Statement:	TRBM
--------------------	-------------

The TRBM command is used to specify a tolerance for monitoring small pivots during the factorization of a matrix. “Small” pivots are deemed to be zero pivots. Therefore, they are associated with rigid body modes (or zero energy modes in general). When this command is specified in the **AERO-S** input file and the sparse, skyline, blockskyline, or mumps pivot solver is specified under the [STATICS](#) command, the generalized inverse of the stiffness, conductivity, mass (when an initial acceleration is to be computed to satisfy the governing equation (see [DYNAMICS](#))), or other relevant matrix is computed in factored form by eliminating the equation associated with a deemed zero pivot and setting the corresponding unknown to zero. This command can be used together with the [GRBM](#) command (see below).

Note 1: See [GRBM](#) for an alternative option for analyzing singular systems.

Note 2: If both the [GRBM](#) and [TRBM](#) commands are specified in the same **AERO-S** input file, the one specified last is chosen. However, if that one is [TRBM](#), it does not de-activate the [GRBM](#) command but simply overwrites the value of its second tolerance, `VALUE_2`, by the value of the tolerance specified under [TRBM](#).

TRBM

VALUE

`VALUE`

Tolerance for monitoring the zero pivots of a matrix during its factorization. The default value is 1.0e-16 (real).

Next: [ROBC](#), Previous: [TRBM](#)

88 RIGID BODY MODES FILTER

Command Statement: **RBMFILTER**

This command is suitable for structure, aeroelastic, and aerothermoelastic simulations involving an unrestrained (or partially restrained) object when: 1) the effect of the rigid body modes is not sought-after, 2) prescribing Dirichlet (displacement) boundary conditions to remove the rigid body modes is not desirable. When this command is specified in the input file and the structural problem contains rigid body modes, a non-trivial projector is constructed. In a static or quasistatic analysis, the transpose of this projector is applied to the (possibly variable) right hand-side to make sure that the external load is self-equilibrated and therefore admissible. In a dynamic analysis: a) this projector is applied to the initial solution (initial displacement *and* velocity fields), and b) at each time-step, the transpose of this projector is applied to the (time-dependent) right-hand side of the equation being solved. In all cases, the net effect is to work with (and output) a solution where the structural rigid body modes have been filtered out.

The constructed projector can be written as
$$P = I - R(R^T Q R)^{-1} R^T Q$$
 where I is the identity

matrix, R is the matrix of the rigid body modes of the structure (always computed with the GRBM method), and Q is either the identity matrix for static analysis, or the mass matrix for quasistatic and dynamic analyses. Hence, in a quasistatic analysis, this command requires the density to be present in the material properties specified under the **MATERIAL** command. An example input file using this command can be found in `FEM.d/fem\examples/Rbm\Filter.d`.

Note 1: This command works with all solvers except the FETI type solvers. It is implemented for linear static, linear quasistatic, linear dynamic, and nonlinear dynamic analyses. It is not currently implemented for nonlinear static analysis.

Note 2: This command is not active when a modal dynamic analysis is performed (see [DYNAMICS](#)).

The reason is that one can achieve the same objective simply by not including the rigid body modes in the input for the **READMODE** command.

The input format of this command is specified below.

RBMFILTER

level Leaving this blank is equivalent to choosing the value of 1.

1 This is the default value. In this case, the projector is used as described above, in both linear and nonlinear analyses.

2 This value is applicable only for linear structural dynamics problems. In this case, the projector is also applied at each time-step to the outcome of the solution of the system of dynamic equations. As a result, the rigid body modes are also filtered directly from the displacement and velocity fields. This is a safety measure in case the previous option does not work properly, for example, because of some ill-conditioning.

Next: [RMSHC](#), Previous: [RBMFILTER](#)

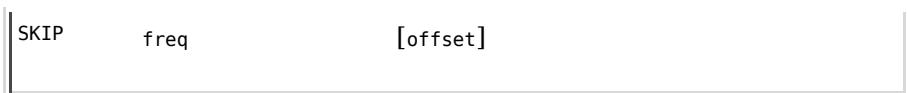
89 CONSTRUCTING A REDUCED-ORDER BASIS

Command Statement: **ROBC**

The command **ROBC** can be used to construct a Reduced-Order Basis (ROB) by applying SVD to one or multiple specified sets of solution state (displacement and velocity) vectors, and/or one or multiple specified sets of previously computed ROB vectors. The inputted solution state snapshots can be scaled by inputted weights before the SVD is performed. The inputted ROB vectors are always scaled by their associated singular values, which are also inputted, before the SVD is performed. The computed ROB can be outputted using the result keyword **ROBDAF** under [OUTPUT](#).

ROBC

SNAPFI	pathandfilename1	pathandfilename2	pathandfilenameN
		...	
ROBFI	pathandfilename1	pathandfilename2	pathandfilenameN
		...	
DIMENS	dim		
MNORMA	flagmn		
VSCALI	flagvs		
SSCALI	flagss	α_1	α_N
	



SNAPFI	Sub-command keyword for specifying one or multiple files containing solution state snapshot vectors and associated weights (characters). If the snapshots were generated in the time domain, the weight of a snapshot is computed as the square root of the average of the time-steps separating this snapshot from the previous and next ones, except for the first and last outputted snapshots which are computed as follows: the weight of the first snapshot is computed as the square root of the time-step separating this snapshot and the next one, and that of the last snapshot is computed as the square root of the time-step separating this snapshot and the previous one.
pathandfilename	Path and name of each file of snapshot vectors (and associated weights) to be considered for constructing the desired ROB (characters). If more than one file is specified, all files are read. The union of all inputted snapshot vectors (and ROB vectors inputted after the sub-command keyword ROBFI and scaled by their associated weights) are compressed by SVD to construct the desired ROB.
ROBFI	Sub-command keyword for specifying one or multiple files containing ROB vectors and associated weights (characters). Usually, these weights are the singular values corresponding to the specified ROB vectors.
pathandfilename	Path and name of each file of ROB vectors (and associated weights) to be considered for constructing the desired ROB (characters). If more than one file is specified, all files are read. All inputted ROB vectors are scaled by their associated weights which are usually the corresponding singular values. The union of these scaled ROB vectors (and snapshot vectors inputted after the SNAPFI sub-command keyword and possibly scaled by their associated weights) is compressed by the SVD to construct the desired ROB.
DIMENS	Sub-command keyword for specifying the desired dimension of the ROB to be constructed (characters).
dim	Dimension of the ROB to be constructed (integer).
MNORMA	Sub-command keyword for specifying whether to construct the desired ROB using the SVD metric associated with the mass matrix of the system or that associated with the identity matrix (characters).
flagmn	If this flag is set to YES, the desired ROB is computed using the SVD metric associated with the mass matrix of the system (characters). If it is set to NO (which is the default setting), the desired ROB is computed using the metric associated with the identity matrix.
VSCALI	Sub-command keyword for specifying whether or not to scale the velocity components of the inputted solution state snapshot vectors by their associated weights before constructing the desired ROB (characters).
flagvs	If this flag is set to YES, the velocity components of the inputted solution state snapshot vectors are scaled by their associated weights before constructing the desired ROB (characters). If it is set to NO (which is the default setting), they are left as is.
SSCALI	Sub-command keyword for specifying whether or not to scale the inputted snapshots by user-specified coefficients before constructing the desired ROB, and acquiring these coefficients in the first case (characters).
flagss	If this flag is set to YES, the inputted solution state snapshot vectors are scaled by the user-specified coefficients before constructing the desired ROB (characters). If it is set to NO (which is the default setting), they are left as is.

α_1	Scaling coefficient to be applied to the solution state snapshot vectors contained in the file <code>pathandfilename1</code> , if flagss is set to YES (real).
α_N	Scaling coefficient to be applied to the solution state snapshot vectors contained in the file <code>pathandfilenameN</code> , if flagss is set to YES (real).
SKIP	Sub-command keyword for specifying whether or not to skip some of the solution state snapshot vectors inputted using the sub-command keyword SNAPFI for the construction of the desired ROB (characters).
freq	Frequency (every so many) at which to skip a solution state snapshot vector in any specified solution state snapshot file (integer). The default value is 1.
offset	Offset in terms of the solution state snapshot number to apply before starting the skipping process (integer). The default value is 0.

Next: [SENSORS](#), Previous: [ROBC](#)

90 CONSTRUCTING A REDUCED MESH

Command Statement:	RMSHC
--------------------	--------------

The RMSHC command can be used to construct a (hyper) reduced mesh by sampling the original mesh using a non-negative least-squares algorithm, and a POD-based reduced-order basis and training solutions.

RMSHC

PODROB	pathandfilename1
TRNSOL	pathandfilename2
DIMENS	dimens
SAMFRQ	samfrq
TOLERA	tolera
EXTFOL	flag

PODROB	Sub-command keyword to specify a path and filename for the file containing the reduced-order basis (characters).
pathandfilename1	Path and filename of the file containing the reduced-order basis (characters).
TRNSOL	Sub-command keyword to specify a path and filename for the file containing the training solutions needed for building the training force vectors (characters).
pathandfilename2	Path and filename of the file containing the solution snapshots needed for building the training force vectors (characters).
DIMENS	Sub-command keyword to specify the dimension of the reduced-order basis (characters).
dimens	Dimension of the reduced-order basis (integer).
SAMFRQ	Sub-command keyword to specify a sampling frequency for the inputted solution snapshots (characters).
samfrq	Specifies using every <code>samfrq</code> -th solution snapshot for building the training force vectors (integer).
TOLERA	Sub-command keyword to specify a tolerance for controlling the error in energy conservation due to mesh sampling (characters).

tolera	Tolerance for controlling the error in energy conservation due to mesh sampling. For example, <code>tolera = 0.01</code> means that 99 % of the energy associated with the training force vectors and the reduced-order basis is conserved on the reduced mesh (real).
EXTFOL	By default, the training force vectors are the internal force vectors. However, this sub-command keyword can be used to define the training force vectors as the union of the internal and external follower (see FORCES) force vectors.

Next: [SLOSH](#), Previous: [RMSHC](#)

91 SENSORS *S*

Command Statement:	SENSORS
--------------------	----------------

The `SENSORS` command statement is used to specify to **AERO-S** the degrees of freedom to be observed and whose structural state is to be passed to the user defined control subroutine “control.C” (see [ACTUATORS](#)). An example input file using the `SENSORS` command can be found in `FEM.d/fem\examples/Control.d`

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: For nonlinear analyses, sensor information on the velocity and acceleration of a nodal degree of freedom is not currently available for use in the “control.C” file. These are currently passed as zero.

SENSORS

NODE#	DOF#
-------	------

NODE#	Node number where the sensor is to be placed (integer).
DOF#	Degree of freedom local number where the sensor is to be placed (integer).

Next: [SZEM](#), Previous: [SENSORS](#)

92 SLOSHING PROBLEMS

Command Statement:	SLOSH
--------------------	--------------

The command `SLOSH` is used to specify the computation of the sloshing modes of an incompressible and inviscid fluid whose free surface is orthogonal to a specified gravity field. The computational approach assumes linear kinematics and is based on a fluid displacement potential. In three

dimensions, it does not involve any fluid material property. In two dimensions, it requires inputting the thickness of the considered slice of the problem. In both cases, the **SLOSH** analysis associated with this command requires identifying a free surface using free-surface elements (element type 312 in three dimensions and element type 302 in two dimensions, see [TOPOLOGY](#)) but does not require any other boundary condition.

Hence, this command should be used in conjunction with the commands [EIGEN](#) and [STATICS](#) in order to solve the arising eigenvalue problem.

SLOSH

SLGRAV **slgrav**

slgrav Magnitude (scalar) of the gravitational acceleration whose direction must be orthogonal to the free surface of the problem. **AERO-S** uses this specified value to convert the sloshing eigenvalues to sloshing frequencies (number of cycles per second) for output (real).

Next: [HFTT](#), Previous: [SLOSH](#)

93 SLOSHING ZERO ENERGY MODE

Command Statement: **SZEM**

This command is effective only for a sloshing eigen computation involving a zero energy mode. It is used to request the computation of the constant potential mode using a physics-based algorithm (rather than the tolerance-based algorithm associated with the **TRBM** command). It should be used in conjunction with the **SLOSH** and related commands.

SZEM

Next: [STATICS](#), Previous: [SZEM](#)

94 SOURCE TIME TABLE-HEAT CONDUCTION

Command Statement: **HFTT[TABLE_ID]**

The **HFTT** command statement must be used to specify the time-dependent amplification of the boundary fluxes. For FEM, pairs of time and amplification values are input. Linear interpolation is also used for "in between" points.

HFTT [TABLE_ID]

TIME_1	AMP_1
.	.
.	.
.	.
TIME_n	AMP_n

TABLE_ID

Optional non-negative integer which uniquely identifies a source-time table so that it can be associated with a "load" set to define the "load" case for a dynamic analysis using the [LOADCASE](#) command. The default value is 0. Hence, the [HFTT](#) command can be repeated as many times as desired within the same input file using each time a different value for TABLE_ID and different data.

TIME_1

A specified time point (float).

AMP_1

A specified amplification value at time point TIME_1 (float). This amplification factor is automatically set to zero for all times prior to the earliest specified time point and all times later than the latest specified time point.

Next: [SDETAFT](#), Previous: [HFTT](#)

95 STATICS

Command Statement:

STATICS

The STATICS command statement can be used mainly for two independent purposes: select a static analysis, and/or select an equation solver needed for the static analysis or any other type of analysis specified in that case by an additional command such as [DYNAMICS](#), [EIGEN](#), or other. For linear static analysis, it can also be used for requesting the interpretation of concentrated follower (see [FORCES](#)) and pressure-induced (see [PRESSURE](#) and [CONWEP](#)) forces and moments as configuration-dependent external forces and moments and applying to them a piecewise constant treatment, thereby leading to a quasi-static analysis.

The input format of this command is given below.

Note 1: Except when otherwise specified, the solvers proposed below are available only for symmetrical systems of equations.

Note 2: Among all solvers proposed below, the following ones are suitable for the solution of singular (but consistent) systems: skyline, blockskyline, sparse, mumps pivot, superlu, and FETI DP. Furthermore among these solvers, only sparse, skyline, mumps pivot, and FETI DP compute and return for further usage the null space of the singular matrix. Singular systems arise: (1) if the problem is formulated without sufficient Dirichlet boundary conditions, or (2) the formulated problem contains redundant constraints and the Lagrange multiplier method is chosen for enforcing these constraints (see [CONSTRAINTS](#)).

Note 3: Among all solvers proposed below, the following ones are suitable for the solution of indefinite systems: spooles and mumps with pivoting turned on, superlu, gmres, and FETI DP. Indefinite systems arise: (1) if the analysis involves the [HELMHOLTZ](#), [IMPEDANCE](#), or [EIGEN](#) command with a positive shift (see SHIFT in [EIGEN](#)) or for buckling analysis (see ARPACK in [EIGEN](#)), or (2) the structural model includes rigid and/or joint elements (see [TOPOLOGY](#)), linear multi-point constraints (see [LMPC](#)), or tied surfaces (see [TIEDSURFACES](#)), and the Lagrange multiplier method is chosen

for enforcing the associated constraints (see [CONSTRAINTS](#)).

Note 4: Among all FETI methods, only `FETI DP` is maintained for nonlinear structural static and dynamic analyses.

Note 5: When a non deterministic analysis is performed using the intrusive version of the Polynomial Chaos method, only the `pcg`, `bfg`, and `cr` solvers can be selected for solving the resulting large system of equations.

Note 6: In all aforementioned cases, the following direct solvers can be executed in parallel on a distributed memory system using MPI or a shared memory system using OpenMP, in the context of a single domain, that of a coarse solver for a FETI-type method, or that of multiple subdomains (see [DECOMPOSE](#)):

SPOOLES	Can be executed in parallel on a shared memory system using OpenMP, in both contexts of a single domain and that of a coarse solver for a FETI-type method.
MUMPS	Can be executed in parallel on a distributed system using MPI, in all contexts of a single domain, that of a coarse solver for a FETI-type method, and that of multiple subdomains.
skyline	Can be executed in parallel on a shared memory system using OpenMP, in both contexts of a single domain and that of a coarse solver for a FETI-type method.
block-skyline	Can be executed in parallel on a shared memory system using OpenMP, in both contexts of a single domain and that of a coarse solver for a FETI-type method.

Note 7: In all aforementioned cases, the following iterative solvers can be executed in parallel on a distributed memory system using MPI and/or a shared memory using OpenMP, in the context of multiple subdomains (see [DECOMPOSE](#)):

All FETI-type solvers	Can be executed in parallel on a shared memory system using OpenMP, on a distributed memory system using MPI, and on a hybrid system using OpenMP on the shared memory subsystem and MPI across such subsystems.
-----------------------	--

STATICS

METHOD
PARAMETERS
PIECEWISE real_1 real_2
CASES case_id_1 case_id_2 ... case_id_n

METHOD (keyword)

direct	For a skyline direct solver (characters). This is also the default solver (except when the <code>INPC</code> command is used) and is available for frequency-domain acoustic (Helmholtz) problems.
skyline	For a skyline direct solver (characters). This is also the default solver and is available for frequency-domain acoustic (Helmholtz) problems.
sparse	For Esmond's sparse direct solver (characters). This solver is also available for frequency-domain acoustic (Helmholtz) problems.

<code>sgisparse</code> (completely spelled out)	For SGI's sparse direct solver (characters). Runs only on SGI machines.
<code>spooles</code> (completely spelled out)	For the SPOOLES sparse direct solver (characters). Cannot be used for: (a) a singular system, and (b) a system with less than 24 equations and unknowns.
<code>spooles pivot</code> (completely spelled out)	For the SPOOLES sparse direct solver with pivoting option turned on (characters).
<code>spooles unsymmetric</code> (completely spelled out)	For the SPOOLES unsymmetric sparse direct solver (characters).
<code>mumps</code> (completely spelled out)	For the MUMPS sparse symmetric positive definite direct solver (characters).
<code>mumps pivot</code> (completely spelled out)	For the MUMPS sparse general symmetric direct solver with pivoting option turned on (characters).
<code>mumps unsymmetric</code> (completely spelled out)	For the MUMPS unsymmetric sparse direct solver (characters).
<code>superlu</code> (completely spelled out)	For the SUPERLU unsymmetric sparse direct solver (characters).
<code>sgisky</code> (completely spelled out)	For SGI's skyline direct solver (characters). Runs only on SGI machines.
<code>frontal</code> (completely spelled out)	For a frontal direct solver (characters).
<code>pcg</code>	For a preconditioned conjugate gradient solver (characters).
<code>bcg</code>	For a bi-conjugate gradient solver (characters).
<code>cr</code>	For a conjugate residual solver (characters).
<code>gmres</code>	For a generalized minimum residual solver (characters). Applicable to symmetric and unsymmetric systems.
<code>FETI</code>	For a one-level FETI solver (default) (characters).
<code>FETI 1</code>	For a FETI-1 (one-level FETI method) solver (characters).
<code>FETI 2 OLD</code>	For a FETI-2 solver using a "full" coarse problem implementation (characters).
<code>FETI 2 NEW</code>	For a FETI-2 solver using a "sparse" coarse problem implementation (characters).
<code>FETI DP</code>	For a FETI-DP (FETI dual-primal method) solver (characters).
<code>FETI DPH</code>	For a FETI-DPH (FETI dual-primal method with augmentation basis) solver (characters). This solver is to be used only with the <code>IMPEDANCE</code> command. Depending on whether the system to be solved is damped or not, or contains a complex boundary condition or multipoint constraint or not, this solver performs in the real or complex domain.
<code>PARAMETERS</code>	(keywords or keywords and values) for pcg/bcg/cr/gmres solvers, in any order.
<code>precno</code>	Specifies the preconditioner (characters).
0	Unpreconditioned (integer).
1	Diagonal scaling (integer). This option should not be used when solving an indefinite system arising in conjunction with the usage of the LMPC or TIEDSURFACES command, or whenever a structural model includes rigid and/or joint elements (see TOPOLOGY), because in all these cases, the system matrix will contain zero diagonal entries.
2	Incomplete block diagonal scaling (integer). This option is available only for a system of equations resulting from the intrusive version of the Polynomial Chaos method. In this case, the subsystems of equations

associated with the diagonal blocks can be solved either using the `sparse` solver (default), or the `FETI DP` method with its acceleration for multiple right sides. If the `FETI DP` method is chosen as a block diagonal solver, then its keyword and the keywords associated with its options should be specified after all other parameters of the main (outer) solver have been specified.

<code>tolpcg</code>	Error tolerance for the convergence of the pcg solver (real).
<code>tolbcg</code>	Error tolerance for the convergence of the bcg solver (real).
<code>tolcr</code>	Error tolerance for the convergence of the cr solver (real).
<code>tolgmres</code>	Error tolerance for the convergence of the gmres solver (real).
<code>maxitr</code>	Maximum number of iterations to be performed (integer).

PARAMETERS (keywords or keywords and values) for spooles solver, in any order.

`spooles_renum` Specifies the renumbering (integer, default value is 0).

0 Best of nested dissection and multisection.

1 Multiple minimum degree.

2 Multisection.

3 Nested dissection.

`spooles_scale` Specifies the scaling (integer, default value is 0).

0 No scaling.

1 Symmetric scaling.

`spooles_tau` Upper bound on the magnitude of the largest element in L or U when pivoting enabled (real $\boxed{1.0}$). If this number is too small the results will be wrong. The default value of 100 is generally safe if scaling is used (see `spooles_scale`).

`spooles_mslvl` Message output level (integer ≥ 0 , default value is 0).

`spooles_maxdomainsize` n /`spooles_maxdomainsize` is the maximum subgraph size used by SPOOLES orderings, where n is the number of equations in the system to be solved. This parameter is used to control the incomplete nested dissection process. Any subgraph whose weight is less than `maxdomainsize` is not split further (integer ≥ 0 , default value is 24).

`spooles_maxzeros` n *`spooles_maxzeros` is the maximum number of zeros allowed in a supernode/front (real > 0 and ≤ 1.0 , default value is 0.04).

`spooles_maxsize` Maximum number of internal columns in supernode/front (integer > 0 , default value is 64).

PARAMETERS (keywords or keywords and values) for mumps solver, in any order

`mumps_icntl #` Value for ICNTL(#), see mumps documentation for specific details (integer).

`mumps_cntl #` Value for CNTL(#), see mumps documentation for specific details (real).

PARAMETERS (keywords or keywords and values) for FETI, FETI-DP, FETI-DPH solvers, in any order).

`spacedim` This parameter is needed only for the FETI-DPH method with augmentation. It should be set to 2 for two-dimensional problems, and to 3 for three-dimensional ones (integer).

sparse	Specifies Esmond's sparse direct method as the local (subdomain and Dirichlet preconditioner) solver (characters). The default is a skyline solver which can also be invoked by specifying <code>skyline</code> instead of <code>sparse</code> on this line. Note also that there is another mechanism for specifying this option (see below).
local_solver	
skyline	Selects the skyline direct method as the local solver (subdomain and Dirichlet preconditioner problems) (characters). This is also the default choice.
sparse	Selects Esmond's sparse direct method as the local solver (characters).
spooles	Selects the SPOOLES sparse direct method as the local solver (characters).
spooles pivot	Selects the SPOOLES sparse direct method as the local solver with pivoting option turned on (characters).
mumps	Selects the MUMPS sparse direct method as the local solver (characters).
mumps pivot	Selects the MUMPS sparse direct method as the local solver with pivoting option turned on (characters).
coarse_solver	
blockskyline	Selects the block-skyline direct method as the coarse solver. This is also the default choice (characters).
skyline	Selects the skyline direct method as the coarse solver (characters).
sparse	Selects Esmond's (sequential) sparse direct method as the coarse solver (characters).
psparse	Selects the parallel sparse direct method as the coarse solver (characters).
spooles	Selects the SPOOLES sparse direct method as the coarse solver (characters).
spooles pivot	Selects the SPOOLES sparse direct method as the local solver with pivoting option turned on (characters).
mumps	Selects the MUMPS sparse direct method as the coarse solver (characters).
mumps pivot	Selects the MUMPS sparse direct method as the local solver with pivoting option turned on (characters).
precno	Specifies the local preconditioner (integer or characters).
0 or nprec	Unpreconditioned (integer).
1 or lumped	Lumped preconditioner (integer). This is also the default preconditioner.
2 or dirichlet	Dirichlet preconditioner (integer).
projector	
1	Identity based projector (integer). This is also the default projector.
2	Preconditioner based projector (integer). Uses the selected preconditioner for building the so-called Q matrix.
4	Superlumped projector (integer).
scaling	
1 or stiffness	Stiffness based scaling (integer). This is also the default scaling procedure.
2 or topology	Topology (subdomain connectivity) based scaling (integer).
version	
1	The FETI (also known as FETI-1) method (integer).
2	The two-level FETI (also known as FETI-2) method (integer).

<code>nocoarse</code>	This option is only for dynamics. When specified, the FETI algorithm is executed without any “coarse grid”.
<code>corners</code>	This keyword is useful only for the FETI-2, FETI-DP, and FETI-DPH methods. It specifies the treatments of the corners and corner dofs for the construction by these iterative methods of their respective coarse problems. For the FETI-2 method, the user can define both the corner and corner dof selection algorithms. For the FETI-DP and FETI-DPH methods, the corners are automatically chosen by AERO-S but the user can still control the number of dofs at these corners.
<code>cp3</code>	In that case, a corner is defined by FETI-2 as a crosspoint, and only the three active translational dof attached at each corner node are included in the construction of the FETI-2 or FETI-DP and FETI-DPH coarse problems. This is the default value for nodes with 3 dofs. Here, a crosspoint is defined as a point that belongs to more than four subdomains.
<code>cp6</code>	In that case, a corner is defined by FETI-2 as a crosspoint, and all six active dof attached at each corner node are included in the construction of the FETI-2 or FETI-DP and FETI-DPH coarse problems. This is the default value for nodes with 6 dofs. Here, a crosspoint is defined as a point that belongs to more than two subdomains.
<code>be3</code>	In that case, a corner is defined by FETI-2 as the beginning or end of an edge, and only the three active translational dof attached at each corner node are included in the construction of the FETI-2 or FETI-DP and FETI-DPH coarse problems. Note that a crosspoint is also the beginning or end of an edge.
<code>be6</code>	In that case, a corner is defined by FETI-2 either as the beginning or end of an edge, and all six active dof attached at each corner node are included in the construction of the FETI-2 or FETI-DP and FETI-DPH coarse problems. Note that a crosspoint is also the beginning or end of an edge.
<code>fsi_corners</code>	This keyword is useful only for the FETI-DPH method when applied to the solution of a fluid-structure interaction problem (see FSINTERFACE and HWIB) in the frequency domain. It manages the selection of additional, non_essential corner nodes except for numerical scalability. Note that if the decomposition is performed using DECOMPOSE and a separate run of AERO-S , the setting of this parameter must be the same when later running AERO-S using the generated mesh partition to solve a fluid-structure interaction problem using FETI-DPH.
<code>0</code>	In that case, no node of the fluid/structure interface is chosen as a corner node.
<code>1</code>	In that case, every fluid node at the intersection of a subdomain boundary interface and the fluid/structure interface is chosen as an additional corner node.
<code>2</code>	In that case, every node — whether it is a structure or fluid node — on the intersection of a subdomain boundary interface and the fluid/structure interface is chosen as an additional corner node.
<code>augment</code>	This keyword is useful only for the FETI-DP and FETI-DPH methods. It specifies the augmentation of the “coarse grid” by various methods defined by the following commands.
<code>trans/all</code> <code>EdgeGs</code>	In that case, the “coarse grid” is augmented using extra equations generated by the rigid body modes (rbms) of the subdomain interfaces, and organized edge-by-edge. Here an edge refers to an interface between two subdomains and not to the usual geometric edge. The “trans/all”

	option denotes the equation type where “trans” refers to the translational rbms, and “all” refers to both the translational and rotational rbms applied per edge.
Gs trans/all	In that case, the “coarse grid” is augmented using extra equations generated by the traces of the subdomain rigid body modes (rbms) on the subdomain interface boundaries, and organized subdomain-by-subdomain. The “trans/all” option denotes the equation type where “trans” refers to the translational rbms, and “all” refers to both the translational and rotational rbms.
trans/all WeightedEdgeGs	In that case, the “coarse grid” is augmented using extra equations generated by the weighted rigid body modes (rbms) of the subdomain interfaces, and organized edge-by-edge. Here an edge refers to an interface between two subdomains and not to the usual geometric edge. The “trans/all” option denotes the equation type where “trans” refers to the translational rbms, and “all” refers to both the translational and rotational rbms applied per edge. The weights are similar to those used for scaling the residuals. Hence, they are based on stiffness considerations if the scaling option is set to <code>stiffness</code> , or on topological considerations if the scaling option is set to <code>topology</code> .
numdir EdgeWs [type]	<p>This option is exclusive to the FETI-DPH solver and can be combined with the option <code>EdgeGs</code>. It augments the corner-based coarse problem of the FETI-DPH algorithm with extra equations generated by the free-space solutions of the frequency-domain acoustic or elastodynamic (or modelled dynamic shell) equation — these are real cosine and sine waves of arbitrary directions — and organized edge-by-edge. Here an edge refers to an interface between two subdomains and not to the usual geometric edge. There are as many free-space solutions to be considered as there are wave directions to be considered. Setting the optional parameter <code>type</code> to <code>solid</code>, <code>shell</code>, <code>fluid</code>, or <code>any</code>, depending on the type of the elements used in the mesh delivers the best performance. If the mesh contains several types of elements, the option <code>any</code> is recommended (default value). The parameter <code>numdir</code> specifies the number of desired directions and therefore controls the total number of such wavy augmentation modes (integer). Its default value is 0. The exact number of augmentation modes per interface edge is equal to $2 \times \text{numdir}$ for two- and three-dimensional frequency-domain acoustic problems, $4 \times \text{numdir}$ for two-dimensional elastodynamic (or modelled dynamic shell) problems, and $6 \times \text{numdir}$ for three-dimensional elastodynamic (or modelled dynamic shell) problems. In the latter case, the factor 6 comes from the fact that for each direction, there are 2 shear waves and 1 pressure wave, and each of these three waves has a cosine mode as well as a sine mode. The implemented directions are chosen according to the following scheme. In two dimensions, a sector is discretized into n sectors, with n an even integer. A direction is defined by connecting the center of the circle to a point on the circle delimiting a sector. Since both the cosine and sine modes are included, only one direction for each pair of opposite directions needs be retained, which results in a total of $n/2$ directions. Hence, n is chosen to be $2 * \text{numdir}$. In three dimensions, a cube is discretized into $n \times n \times n$ points. A direction is defined by connecting the center of the cube to a point lying on a face of</p>

the cube. Since both the cosine and sine modes are included, only one direction for each pair of opposite directions needs be retained, which results in a total of $n^2 + 4(n - 1)(n/2 - 1) + 2(n - 1)\text{mod}(n, 2)$

directions. Hence, n is chosen so that

$n^2 + 4(n - 1)(n/2 - 1) + 2(n - 1)\text{mod}(n, 2)$ is as close as possible to

`numdir`, with $n^2 + 4(n - 1)(n/2 - 1) + 2(n - 1)\text{mod}(n, 2) \geq \text{numdir}$.

<code>interf_solver</code>	This option is currently available only for the FETI-DP and FETI-DPH solvers. It specifies the Krylov method to be used with these algorithms for solving the interface problem (characters).
<code>CG</code>	Turns on the CG algorithm as an interface problem solver. This is the default value of <code>interf_solver</code> .
<code>CGAL</code>	Turns on Dostal's Augmented Lagrangian CG algorithm with adaptive precision control as an interface problem solver.
<code>GMRES</code>	Turns on the GMRES algorithm as an interface problem solver.
<code>GCR</code>	Turns on the GCR algorithm as an interface problem solver. For frequency sweep problems (see IMPEDANCE), this option is more computationally efficient than GMRES because it can be optimized for systems with multiple right sides (see <code>maxorth</code> below).
<code>orthotol</code>	This option is currently available only for the FETI-DPH solver. It specifies the tolerance to be used for filtering out "small" vectors during the local Gram-Schmidt-like orthogonalization of the augmentation vectors. The default value is 1.0e-02.
<code>mpc_type</code>	This keyword is useful only for the FETI-DP and FETI-DPH methods. It specifies the algorithm to be used for handling potential multipoint constraints (MPCs) (characters). The default is <code>primal</code> when the number of MPCs is less or equal to 1000, and <code>dual</code> otherwise.
<code>dual</code>	In this case, the MPCs are enforced by Lagrange multipliers and are satisfied only at convergence (characters).
<code>primal</code>	In this case, the MPCs are put in the coarse problem and are satisfied at every iteration (characters).
<code>mpc_precn</code>	This keyword is useful only for the FETI-DP and FETI-DPH methods, in the presence of MPCs, and when <code>mpc\type</code> is set to <code>dual</code> (characters). The default is <code>tblock</code> when running a single thread process, and <code>full</code> otherwise.
<code>full</code>	In this case, the $ CC^T $ matrix, where C denotes the constraint matrix, is treated as a single block matrix (characters).
<code>tblock</code>	In this case, the potential algebraic block-structure of the $ CC^T $ matrix, is exploited (characters).
<code>sblock</code>	In this case, the subdomain-structure of the $ CC^T $ matrix, is exploited (characters).
<code>mblock</code>	In this case, the mortar-interface-structure of the $ CC^T $ matrix, is exploited (characters).
<code>diag</code>	In this case, the $ CC^T $ matrix is approximated by its diagonal (characters).

<code>mpc_scaling</code>	
1 or stiffness	Stiffness based scaling (integer).
2 or topology	Topology (subdomain connectivity) based scaling (integer). This is also the default scaling procedure.
<code>cct_solver</code>	
skyline	Selects the skyline direct method as the coarse solver. This is also the default choice (characters).
sparse	Selects Esmond's (sequential) sparse direct method as the coarse solver (characters).
<code>cct_tol</code>	Specifies the tolerance to be used for detecting singularities in the solution of the CCT^T problems of the FETI-DP and FETI-DPH methods in the presence of MPCs (real). The default is 1.0E-12.
<code>kryprec</code>	This option is only for nonlinear problems. It turns on the Krylov preconditioner enrichment for the solution of nearby problems in a Newton method (characters).
1	Turns on the Krylov preconditioner and gives to the <code>maxorth</code> parameter (see below) the scope of the entire nonlinear analysis.
2	Turns on the Krylov preconditioner and gives to the <code>maxorth</code> parameter (see below) the scope of a load-step in a nonlinear analysis.
<code>global_cor_rbm_tol</code>	Specifies the tolerance to be used for detecting singularities in the solution of the corner-based coarse problems of the FETI-DP and FETI-DPH methods (real). The default is 1.0E-6.
<code>global_rbm_tol</code>	Specifies the tolerance to be used for detecting the global rigid body modes when using a FETI method (real). The default is 1.0E-6.
<code>maxorth</code>	Specifies the maximum total number of reorthogonalization vectors used for accelerating a FETI algorithm and/or enriching its chosen preconditioner during a simulation (integer). For linear problems or while the tangent operator of a nonlinear problem is frozen, the acceleration is performed using a multiple right side technique based on reorthogonalization, but only if <code>interf-solver</code> (see below) is set to CG or GCR. For nonlinear problems, the acceleration is performed using a multiple left side technique based on the enrichment of the preconditioner, but only if: (a) <code>interf_solv</code> (see below) is set to CG, and (b) <code>kryprec</code> (see above) is activated. If the <code>kryprec</code> option is not activated, <code>maxorth</code> is to be understood as the maximum total number of reorthogonalization vectors per Newton iteration, as these vectors are flushed when the tangent operator is rebuilt. For nonlinear problems where the tangent operator is periodically frozen, if the <code>kryprec</code> option is used, priority in the accumulation of vectors is given to enriching the preconditioner. The default value is <code>maxitr</code> (see below). To turn-off this reorthogonalization option, set <code>maxorth</code> to 1.
<code>tolfeti</code>	Error tolerance for the convergence of the feti solver (real). The default value is 1.0e-06.
<code>maxitr</code>	Maximum number of iterations to be performed (integer). The default value is 500.
<code>PIECEWISE real_1</code>	This sub-command keyword (characters) followed by two real numbers is applicable to <i>linear static analysis</i> only. It requests the interpretation of concentrated follower (see FORCES), pressure-induced (see PRESSURE and CONWEP), and temperature induced (see TEMPERATURES and
<code>real_2</code>	

[THERMOE](#)) forces and moments, and moments due to gravity (see [GRAVITY](#)) and discrete masses (see [DIMASS](#)) as configuration-dependent external forces and moments, and enables their piecewise constant treatment. In this treatment, the application of the aforementioned external loading is divided into steps and is performed at the beginning of each step using the updated deformed configuration. To this effect, the first entered real number, `real_1`, specifies the load fraction increment to apply at each step, and the second real number, `real_2`, specifies the total load factor to apply to all inputted external loads. For example, consider the input "PIECEWISE 0.25 1.0". The second entry "1.0" requests keeping all external loads inputted using various commands of **AERO-S** unchanged, and the first entry "0.25" requests their application to the system of interest in 4 steps — that is, the application at each step of 25% only of these external loads. On the other hand, inputting "PIECEWISE 0.25 2.0" requests multiplying first all external loads resulting from various external load input commands by the factor 2, then applying at each step a load increment equal to 0.25 times of the *original* external loads. This implies splitting the application of the magnified external loads in 8 steps and applying at each step an external load increment equal to 12.5% of the magnified external load.

CASES

This optional sub-command keyword (characters) can be used to select a "load" case among those defined in the [LOADCASE](#) command. In this capacity, it is relevant to all analyses. However for *linear* static analysis and single-frequency frequency response analysis, it can also be used to select multiple "load" cases among those defined in the [LOADCASE](#) command. Selecting no "load" case is equivalent to selecting the load case 0 which by default contains the "load" set 0 and any "load" generated by a command which does not support the "load" set construct (`LOADSET_ID`) (see the [FORCES](#), [PRESSURE](#), [HNEU](#), [FLUX](#), and/or [CONVECTION](#) command).

case_id_j

Non-negative integer identifying uniquely a j -th "load" case that is defined in the [LOADCASE](#) command. Specifying more than one case identifier on the same line results in a multiple "load" case analysis.

Next: [SURFACETOPO](#), Previous: [STATICS](#)

96 STRUCTURAL DAMPING LOSS FACTOR TABLE

Command Statement:	SDETAFT
--------------------	----------------

The command **SDETAFT** can be used to describe for a given material, the variation of the loss factor η (see the sub-command keyword **STRDAMP** in [MATERIAL](#)) with the frequency f . This evolution can be specified here in one or multiple curves (or tables) each defined by pairs of loss factor and frequency values. On each curve, linear interpolation is used for the "in between" points, and constant extrapolation (using the loss factor value at the closest frequency) is adopted for the "outside" points. When multiple curves are defined, they are inputted one after the other, and each is identified by an id number as described below.

SDETAFT

CURVE	curve_id
f_1	η_1
—	—
.	.
.	.
.	.
f_n	η_n
—	—

CURVE Sub-command keyword (characters) for defining a curve (or table) describing the variation of the loss factor η with the frequency f in the form of pairs (f_i, η_i) .

curve_id "Id number" for the defined curve (integer).

f_i A sampled frequency value (real).

η_i Value of the loss factor at the frequency f_i (real).

Next: [TETT](#), Previous: [SDETAFT](#)

97 SURFACE TOPOLOGY

Command Statement: **SURFACETOPO**

The SURFACETOPO command statement can be used to define a discrete surface on a body by describing its faceted connectivity. Each facet may, but does not necessarily have to, be an element defined in the command [TOPOLOGY](#). Such a surface can be paired with another similar surface using another command statement in order to define a surface-to-surface interaction (see [TIEDSURFACES](#), [CONTACTSURFACES](#), [FSINTERFACE](#), [AERO](#)). It can also be referred to by another command statement to define a surface boundary condition (see [PRESSURE](#), [FORCES](#), [DISPLACEMENTS](#), [FLUX](#) and [TEMPERATURES](#)) or a group of nodes (see [GROUPS](#)).

Each surface is inputted using two sections. The first one identifies the surface to be defined and assigns to it, if needed, a thickness. This section is followed by a second section which contains the description of all faces constituting the surface. Each face is specified on a separate line. The set of two sections can be repeated as many times as there are surfaces to be defined.

The format of this command is as follows.

SURFACETOPO	ID_NUMBER	SURFACE_THICKNESS	t
-------------	-----------	-------------------	---

ID_NUMBER

Surface id number (integer).

SURFACE_THICKNESS

Optional keyword indicating that the surface identified by ID_NUMBER defines the midplane of a physical surface (string).

t Real or virtual thickness of the surface used for detecting and enforcing contact conditions on both of its sides (real). The default value is 0.

FACE#	FACETYPE	CONNECTIVITY_NODES
-------	----------	--------------------

FACE#	Face id number whose type and connectivity are to be specified (integer).
FACETYPE	
1	4-node quadrilateral.
2	8-node quadrilateral.
3	3-node triangle.
4	6-node triangle.
5	9-node quadrilateral.
6	12-node quadrilateral.
7	10-node triangle.

Next: [THERMOE](#), Previous: [SURFACETOPO](#)

98 THERMAL EXPANSION TEMPERATURE TABLE

Command Statement:	TETT
--------------------	-------------

The **TETT** command statement can be used to describe the evolution of the coefficient of thermal expansion with temperature, for a given material. This evolution can be specified here in a curve (or table) defined by pairs of temperature and coefficient of thermal expansion values. Linear interpolation is used for "in between" points, and the extrema values are adopted for "outside" points. Several curves can be specified, one after the other. Each curve is identified by an "id number" as described below.

TETT

CURVE	curve_id
T_1	TE_1
.	.
.	.
.	.
T_n	TE_n

CURVE	curve_id	"Id number" for the following curve (or table) (integer).
T_1		A specified temperature value (float).

TE_1	A specified coefficient of thermal expansion value at temperature T_1 (float).
------	--

Next: [THERMOH](#), Previous: [TETT](#)

99 THERMOELASTICITY

Command Statement:	THERMOE
--------------------	----------------

The THERMOE command statement is used to indicate that **AERO-S** is to interact with itself to perform a *transient* thermoelastic (thermostructure-structure vibration) *one-way* coupled simulation.

The syntax for invoking this option is given below.

THERMOE

Next: [TIEDSURFACES](#), Previous: [THERMOE](#)

100 THERMOELASTICITY: FEM-HEAT (FEM-HEAT Only)

Command Statement:	THERMOH
--------------------	----------------

The THERMOH command statement is used to indicate that **AERO-S** is to interact with itself to perform a *transient* thermoelastic (thermostructure-structure vibration) *one-way* coupled simulation.

The syntax for invoking this option is given below.

THERMOH

Next: [TOPOLOGY](#), Previous: [THERMOH](#)

101 TIED SURFACES

Command Statement:	TIEDSURFACES
--------------------	---------------------

The TIEDSURFACES command statement can be used to tie — that is, enforce perfect contact between — pairs of surfaces defined using the [SURFACETOPO](#) command. Surface interactions are detected using ACME's search module.

Note 1: For an explicit dynamic analysis, the tie constraints are enforced using ACME. For an implicit dynamic analysis, they are enforced using **AERO-S**.

Note 2: For the enforcement of tied surface constraints by ACME's Lagrange multiplier method in explicit dynamic analysis, the discrete kinematic constraint equations are defined using a node-to-segment approach and the constraint forces are computed using ACME's enforcement module.

Note 3: For the enforcement of tied surface constraints by either the Lagrange multiplier method in all but explicit dynamic analyses, or the penalty or elimination method in all analyses, the discrete kinematic constraint equations are defined using a segment-to-segment (mortar) approach.

Note 4: The enforcement of tied surface constraints by the Lagrange multiplier method in all but explicit dynamic analyses is supported only by the FETI-DP family of solvers, the GMRES solver, and the SPOOLES and MUMPS direct sparse solvers with pivoting enabled (see [STATICS](#)).

TIEDSURFACES

SURF_PAIR_ID#	MASTER	SLAVE
---------------	--------	-------

or, for statics, frequency response, eigenvalue problems and dynamics with an implicit scheme

SURF_PAIR_ID#	MASTER	SLAVE	CONSTRAINT_METHOD	MORTAR_TYPE	NORMAL_TOL	TANGENTIAL_TOL
---------------	--------	-------	-------------------	-------------	------------	----------------

or, for dynamics with an explicit scheme

SURF_PAIR_ID#	MASTER	SLAVE	CONSTRAINT_METHOD	KPART_TYPE	NORMAL_TOL	TANGENTIAL_TOL
NUM_ITER	CONVERG_TOL					

SURF_PAIR_ID#	Id number of the surface pair to be described (integer).					
MASTER	Identification of the master (mortar method) surface (see SURFACETOPO) (integer).					
SLAVE	Identification of the slave (mortar method) surface (see SURFACETOPO) (integer).					
CONSTRAINT_METHOD	Method for enforcing the associated constraints (characters). The default method is set in CONSTRAINTS and used whenever this entry is omitted.					
multipliers	The Lagrange multiplier method.					
elimination	The elimination method.					
penalty	The penalty method. The parameter beta should be a large positive number, typically of the order of 10^8 (no default value is provided).					
[beta]						
MORTAR_TYPE	Mortar type: 0 = standard, 1 = dual, default value is 0 (integer).					
NORMAL_TOL	Normal search tolerance used by ACME to identify interactions, default value is 0.1 (float) (see Figs. 1.2 and 1.3 in Section 1.3 of ACME's User Reference Manual).					
TANGENTIAL_TOL	Tangential search tolerance used by ACME to identify interactions, default value is 0.001 (float) (see Figs. 1.2 and 1.3 in Section 1.3 of ACME's User Reference Manual).					
KPART_TYPE	Kinematic partitioning type for explicit dynamics: 0 = fixed, 1 = automatic, default value is 0 (integer).					
NUM_ITER	Maximum number of predictor-corrector iterations to be performed at each time step for explicit dynamics, default value is 5 (integer).					
CONVERG_TOL	Convergence tolerance of the predictor-corrector iteration loop for explicit dynamics, default value is 1.0e-10 (float).					

Next: [USDF](#), Previous: [TIEDSURFACES](#)

102 TOPOLOGY *S*

Command Statement:	TOPOLOGY
--------------------	-----------------

The **TOPOLOGY** command statement is used to signal that the following data lines correspond to the connectivity and type of each element. The input format is given below. There should be as many lines as there are elements in the system.

Note 1: **AERO-S** supports a run-time generation of frames for *flexible* beams (see [EFRAMES](#)) that is activated when a third node in the local x - z plane is found in the definition of a flexible beam element within this **TOPOLOGY** command. The only requirement for the third node is that it does not be colinear with the other two beam nodes. Using this third node option releaves the user from specifying the **EFRAMES** command. An example illustrating the third node option can be found in FEM.d/fem_examples/Third_Node.d.

Note 2: Thermal loads (see [TEMPERATURES](#)) are currently implemented only for the Euler beam, 4-noded plane stress/plane strain, 4-noded as well as 10-noded tetrahedron, 8-noded brick, 6-noded penta (prism with triangular cross section), and 3-noded and 4-noded shell elements.

Note 3: For thermal analysis, **AERO-S** constructs the contribution of convection effects to the right-hand side vector of the resulting system of equations using the [CONVECTION](#) command. However, it constructs the contribution of convection effects to the stiffness matrix using the "boundary convection" elements (type 47, 48 and 49) which must be superposed to the edges or faces of existing thermal elements. The data for these boundary convection elements (depth, area, convection coefficients and ambient temperature) must be specified in the [MATERIAL](#) command.

Note 4: Currently, only the SPOOLES and MUMPS sparse direct solvers and the FETI-DP(H) solvers can handle rigid elements.

Note 5: For DEM (Discontinuous Enrichment Method) and DGM (Discontinuous Galerkin Method) elements, Q, T, and H designate a quadrilateral, a triangle in two dimensions and a tetrahedron in three dimensions, and an hexahedron, respectively. In the notation Q-X-Y, T-X-Y, and H-X-Y, X denotes the number of enrichment functions in the element and Y denotes the number of Lagrange multiplier degrees of freedom per edge or face of the element. In the notation X1xX2-Y for elastodynamics, X1 denotes the number of directions of the plane wave enrichment functions, X2 denotes the number of enrichment functions per wave direction (X2 = 2 (one pressure and one shear wave) in two dimensions and X2 = 3 (one pressure and two shear waves) in three dimensions), and Y denotes the number of Lagrange multiplier degrees of freedom per edge or face of the element. The connectivity (geometry and local node numbering) of all Q, T, and H DEM and DGM elements is that of Q, T, and H higher-order isoparametric elements, respectively. Hence, for each DEM element, the degree of the polynomial field can be deduced from the number of nodes of that element.

Note 6: The fluid elements (type 301, 302, 311, 312, 321, and 331) are active only in the following cases: (1) a mass computation using the [MASS](#) command, (2) a sloshing eigen computation using the [SLOSH](#) and related commands, (3) a hydroelastic eigenvalue computation using the [EIGEN](#), [HEFRS](#), [HEFSB](#) and related commands.

Note 7: The heat radiation elements 56, 57, and 58 are primarily *nonlinear* elements. Hence, the presence of any of these elements in a thermal model usually implies a nonlinear analysis and therefore requires specifying the [NONLINEAR](#) command in the ASCII Input Command Data file. However, in the presence of such elements in the thermal model but absence of the [NONLINEAR](#) command in the aforementioned input file, **AERO-S** performs a linearized thermal (perturbation) analysis where the conductivity matrix is adjusted by the Jacobian of the finite element treatment of the radiation boundary condition evaluated at an equilibrium temperature field that must be specified in this case using the [ETEMP](#) command. In principle, the equilibrium temperature field is the solution of a related *nonlinear* steady thermal analysis problem with radiation boundary conditions; therefore, it captures the effect of the reference temperature of the enclosure receiving the radiation which is specified in the parameter τ_r of the [MATERIAL](#) command. However, the linearized thermal problem solved by **AERO-S** in the presence of a heat radiation element in the thermal model but absence of the [NONLINEAR](#) command is not affected by this reference temperature.

ELEMENT#	ELEMENT_TYPE	CONNECTIVITY_NODES
----------	--------------	--------------------

ELEMENT# Element identification number whose type and connectivity are to be specified (integer). A (*) indicates the element is not yet fully implemented and/or validated.

ELEMENT_TYPE: M=Mechanic - A=Acoustic (Helmholtz or Time-Domain) - H=Heat - C=Coupled Thermoelastic - F=Fluid

- 1 M: 3d truss (bar) element with 3 dof/node ([Figure 1](#)).
- 2 M: 2d 4-node plane stress/plane strain quad element with 2 dof/node ([Figure 2](#)). **Note that only a lumped mass matrix is available for this element.**
- 3 H: 3d 4-node quad element with 1 dof/node ([Figure 2](#)).
- 4 M: 2d 3-node triangular element with 2 dof/node ([Figure 3](#)). **Note that only a lumped mass matrix is available for this element.**
- 6 M: 3d Bernoulli beam element with 6 dof/node ([Figure 1](#)).
- 7 M: 3d Timoshenko beam element with 6 dof/node ([Figure 1](#)). **Note that only a lumped mass matrix is available for this element.**
- 8 M: 3d 3-node triangular AQR shell element with 6 dof/node ([Figure 3](#)). **Note that only a lumped mass matrix is available for this element.**
- 88 M: 3d 4-node shell element element with 6 dof/node that splits into 2 triangular shell elements of type 8 ([Figure 2](#)). **Note that only a lumped mass matrix is available for this element.**
- 9 H: 3d 2-node lineal element with 1 dof/node ([Figure 1](#)).
- 10 H: 2d 4-node quad element with 1 dof/node ([Figure 2](#)).
- 11 M: 3d torsion spring element with 3 dof/node for linear analysis ([Figure 4](#)).
- 12 C: 3d thermomelastic truss element with 4 dof/node ([Figure 1](#)).
- 13 C: 2d thermoelastic quad element with 3 dof/node ([Figure 2](#)).
- 14 C: 3d thermoelastic Bernoulli beam element with 7 dof/node ([Figure 1](#)).
- 15 M: 3d 3-node triangular AQR shell element with 6 dof/node and composite as well as nonlinear material capability ([Figure 3](#)). **Note that only a lumped mass matrix is available for this element.** Hence, this element effectively replaces the elements 8 and 20 even though both are still supported.

- 1515 M: 3d 4-node quadrilateral AQR shell element with 6 dof/node and composite as well as nonlinear material capability ([Figure 2](#)). **Note that only a lumped mass matrix is available for this element.** Hence, this element effectively replaces the elements 88 and 2020 even though both are still supported.
- 16 M: 3d Belytschko-Tsay 4-node quadrilateral or degenerated quadrilateral ([Figure 2](#)), or 3-node triangular shell element with 6 dof/node and 1-point quadrature rule ([Figure 3](#)), and nonlinear material capability. **Currently, this element is available only for explicit dynamic analyses.**
- 17 M: 3d 8-node brick element with 3 dof/node ([Figure 5](#)) and nonlinear material capability.
- 18 M: 3d 4-node shear panel element with 3 dof/node ([Figure 2](#)). **Note that only a lumped mass matrix is available for this element.**
- 19 M: 3d 3-node triangular membrane element with 6 dof/node ([Figure 3](#)). (Only in-plane and drilling stiffnesses). **Note that only a lumped mass matrix is available for this element.**
- 20 M: 3d 3-node triangular composite or orthotropic shell element with 6 dof/node ([Figure 3](#)). **Note that only a lumped mass matrix is available for this element.**
- 2020 M: 3d 4-node composite or orthotropic shell element with 6 dof/node ([Figure 2](#)). **Note that only a lumped mass matrix is available for this element.**
- 21 M: 3d translational spring-link element with 3 dof/node for linear analysis ([Figure 1](#)).
- 22 M: 3d torsion spring-link element with 3 dof/node for linear analysis ([Figure 1](#)).
- 23 M: 3d 4-node tetrahedral element with 3 dof/node ([Figure 6](#)) and nonlinear material capability.
- 24 M: 3d 6-node pentahedral element (prism with triangular cross section) with 3 dof/node ([Figure 7](#)) and nonlinear material capability.
- 25 M: 3d 10-node tetrahedral element with 3 dof/node ([Figure 8](#)) and nonlinear material capability.
- 30 A: 2d 4-node quad element with 1 dof/node ([Figure 2](#)). must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid.
- 31 A: 2d 4-node quadrilateral GLS element with 1 dof/node ([Figure 2](#)).
- 32* A: 2d 8-node quadrilateral element with 1 dof/node ([Figure 9](#)).
- 33* A: 2d 4-node quadrilateral bubble element with 1 dof/node ([Figure 2](#)).
- 34* A: 2d 4-node quadrilateral two-level bubble element with 1 dof/node ([Figure 2](#)).
- 35 A: 2d 3-node triangular element with 1 dof/node ([Figure 3](#)).
- 36 A: 2d 3-node triangular GLS element with 1 dof/node ([Figure 3](#)).
- 38* A: 2d 6-node triangular element with 1 dof/node ([Figure 10](#)).
- 40 A: 3d 4-node tetrahedron element with 1 dof/node ([Figure 6](#)).
- 41 A: 3d 4-node tetrahedron GLS element with 1 dof/node ([Figure 6](#)).
- 42 A: 3d 10-node tetrahedron element with 1 dof/node ([Figure 8](#)).
- 44 A: 3d 8-node brick GLS element with 1 dof/node ([Figure 5](#)).
- 45* A: 3d 8-node brick element with 1 dof/node ([Figure 5](#)).
- 46 H: 3d 3-node triangular thermal (heat conduction) element ([Figure 3](#)).
- 4646 H: 3d 4-node quadrilateral thermal (heat conduction) element that splits into 2 triangular elements of type 46 ([Figure 2](#)).
- 47 H: 3d 2-node line element with 1 dof/node for boundary convection ([Figure 1](#)).

- 48 H: 3d 4-node quadrilateral element with 1 dof/node for boundary convection ([Figure 2](#)).
49 H: 3d 3-node triangular element with 1 dof/node for boundary convection ([Figure 3](#)).
50 H: 3d 4-node, 10-node, 20-node, or 35-node tetrahedron element with 1 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 11](#)).
51 H: 3d 8-node brick element with 1 dof/node ([Figure 5](#)).
52* M: 3d 6-node triangular shell element ([Figure 10](#)).
53 H: 2d 3-node triangular thermal (heat conduction) element ([Figure 3](#)).
56 H: 3d 2-node heat radiation element ([Figure 1](#)).
57 H: 3d 3-node triangular heat radiation element ([Figure 3](#)).
58 H: 3d 4-node quadrilateral heat radiation element ([Figure 2](#)).
65 M: 3d 2-node rigid truss (bar) element ([Figure 1](#)); enforces constant length of the element.
66 M: 3d 2-node rigid beam element ([Figure 1](#)); enforces constant length of the element, equal rotations of the cross sections at its two nodes, and other constraints between its rotational and translational dofs to simulate a genuinely rigid beam.
67 M: 3d 2-node rigid link (translation and rotation, [Figure 1](#)); for each dof at one node, enforces equality to the corresponding dof at the other node. Therefore, this element enforces constant length like element type 66; however, it also enforces additional constraints and therefore is different from element type 66.
68 M: 3d 2-node rigid translation link ([Figure 1](#)); for each specified translational dof at one node, it enforces equality to the corresponding dof at the other node. Therefore, this element enforces constant length like element type 65; however, it also enforces additional constraints and therefore is different from element type 65.
69 M: 3d 2-node rigid rotation link ([Figure 1](#)); for each specified rotational dof at one node, it enforces equality to the corresponding dof at the other node.
70 M: 3d 8-node rigid brick element with 3 dof/node ([Figure 5](#)); enforces constant distance between each pair of its nodes (special case of element type 71).
71 M: 3d rigid line, plane, or solid element with 3 dof/node and anywhere from 3 to 32 nodes per element ([Figure 12](#)); enforces constant distance between each pair of its nodes.
72 M: 3d 20-node brick element with 3 dof/node ([Figure 12](#)) and nonlinear material capability.
73 M: 3d 3-node rigid shell element ([Figure 3](#)); equivalent to two rigid beam elements, each defined by an edge of the element (special case of element type 74).
74 M: 3d rigid line, plane, or solid element with 6 dof/node and anywhere from 3 to 32 nodes per element ([Figure 13](#)); enforces constant distance between each pair of its nodes, equal values of the rotational dofs at all nodes, and other constraints to simulate a genuinely rigid element.
76 M: 3d 4-node rigid shell element ([Figure 2](#)); equivalent to three rigid beam elements, each defined by an edge of the element (special case of element type 74).
77 M: 3d 1-node point-to-point constraint element with 3 dof/node.
78 M: 3d 1-node point-to-line constraint element with 3 dof/node.
79 M: 3d 1-node point-to-plane constraint element with 3 dof/node.

- 81 H: 2d 4-node contact resistance thermal element with 1 dof/node that can be inserted between two thermal elements of type 10 ([Figure 2](#)).
82 H: 3d 8-node contact resistance thermal element with 1 dof/node that can be inserted between two thermal elements of type 51 ([Figure 5](#)).
83 H: 3d 6-node contact resistance thermal element with 1 dof/node that can be inserted between two thermal elements of type 50 with 4 nodes each ([Figure 2](#)).
84 H: 2d 3-node triangular bulk fluid (thermal) element with 1 dof/node. The first node appearing in the connectivity list of this element must be inside the bulk fluid, and the other two nodes must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid ([Figure 14](#)).
85 H: 3d 4-node tetrahedron bulk fluid (thermal) element with 1 dof/node. The first node appearing in the connectivity list of this element must be inside the bulk fluid, and the other three nodes must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid ([Figure 15](#)).
86 H: 3d 5-node pyramidal bulk fluid (thermal) element with 1 dof/node. The first node appearing in the connectivity list of this element must be inside the bulk fluid, and the other four nodes must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid ([Figure 16](#)).
90 A: 3d 6-node wedge element with 1 dof/node ([Figure 7](#)).
91 M: 3d 32-node serendipity brick element with 3 dof/node ([Figure 13](#)) and nonlinear material capability.
92 M: 3d 26-node serendipity wedge element with 3 dof/node ([Figure 17](#)) and nonlinear material capability.
93 A: 3d 32-node serendipity brick element with 1 dof/node ([Figure 13](#)).
94 A: 3d 26-node serendipity wedge element with 1 dof/node ([Figure 17](#)).
95 A: 3d 8-node, 27-node, 64-node, or 125-node hexahedron element with 1 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 18](#)). They also support the PML (Perfectly Matching Layer) computational technology.
96 A: 3d 4-node, 10-node, 20-node, or 35-node tetrahedron element with 1 dof/node ([Figure 11](#)). **These elements use a special local node-numbering for connectivity which goes line-by-line** They also support the PML (Perfectly Matching Layer) computational technology.
97 M: 3d 15-node wedge element with 3 dof/node and nonlinear material capability.
98 A: 2d 4-node, 9-node, 16-node, or 25-node quadrilateral element with 1 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 19](#)). They also support the PML (Perfectly Matching Layer) computational technology.
99 A: 2d 3-node, 6-node, or 10-node triangular element with 1 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 20](#)). They also support the PML (Perfectly Matching Layer) computational technology.
100 M: 2d 4-node, 9-node, 16-node, or 25-node quadrilateral element with 3 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 19](#)).

- 101 M: 2d 3-node, 6-node, or 10-node triangular element with 3 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 20](#)).
- 102 M: 3d 8-node, 27-node, 64-node, or 125-node hexahedral element with 3 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 18](#)).
- 103 M: 3d 4-node, 10-node, 20-node, or 35-node tetrahedral element with 3 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 11](#)).
- 105 A: 3d 8-node, 27-node, 64-node, or 125-node spectral hexahedral element with 1 dof/node. **These spectral elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 18](#)).
- 108 A: 2d 4-node, 9-node, 16-node, or 25-node spectral quadrilateral element with 1 dof/node. **These spectral elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 19](#)).
- 109 H: 3d 8-node, 27-node, 64-node, or 125-node hexahedral element with 1 dof/node. **These elements use a special local node-numbering for connectivity which goes line-by-line** ([Figure 18](#)).
- 111 M: 3d fabric truss element with 3 dof/node ([Figure 1](#)). **Currently, a consistent mass matrix is not available for this element.**
- 118 M: 3d 2-node planar joint element with 6 dof/node ([Figure 27](#)). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its *x* and *y* axes define the orientation in the undeformed configuration of the *x* and *y* axes of the co-rotating reference frame at node 1. The position of node 2 is constrained to lie on the plane defined by the *x* and *y* axes of the co-rotating reference frame at node 1.
- 119 M: 3d 2-node welded joint element with 6 dof/node ([Figure 30](#)). This element constrains the relative translations and rotations between two nodes.
- 120 M: 3d 2-node spherical joint element with 3 dof/node. This element constrains the relative translations between two nodes ([Figure 21](#)).
- 121 M: 3d 2-node translational joint element with 3 rotational dof/node ([Figure 28](#)). This element constrains the relative rotations between two nodes.
- 122 M: 3d 2-node universal joint element with 6 dof/node ([Figure 22](#)). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its *y* axis defines the orientation in the undeformed configuration of the *y* axis of the co-rotating reference frame at node 2. Its *z* axis defines the orientation in the undeformed configuration of the *z* axis of the co-rotating reference frame at node 1. These two axes remain orthogonal during the deformations.

- 123 M: 3d 2-node revolute joint element with 6 dof/node ([Figure 23](#)). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x axis defines the orientation in the undeformed configuration of the axis of free relative rotation.
- 124 M: 3d 2-node cylindrical joint element with 6 dof/node ([Figure 24](#)). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x axis defines the orientation in the undeformed configuration of the axis of free relative translation and rotation.
- 125 M: 3d 2-node prismatic joint element with 6 dof/node ([Figure 25](#)). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x axis defines the orientation in the undeformed configuration of the axis of free relative translation.
- 126 M: 3d 2-node revolute joint-with-driver element with 6 dof/node ([Figure 23](#)) and a relative rotation (between node 2 and node 1) around the joint axis of rotation that can be prescribed using a time-dependent law specified in [MATERIAL](#) (for static analysis, the relative rotation is set to the initial value of the chosen law). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x axis defines the orientation in the undeformed configuration of the axis of forced relative rotation.
- 127 M: 3d 2-node pin-in-slot joint element with 6 dof/node ([Figure 29](#)). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x -axis defines the orientation in the undeformed configuration of the axis of free relative translation, and its y -axis defines the orientation in the undeformed configuration of the axis of free relative rotation.
- 128 M: 3d 4-node plane stress quadrilateral membrane element with 3 dof/node ([Figure 2](#)).
- 129 M: 3d 3-node plane stress triangular membrane element with 3 dof/node ([Figure 3](#)).
- 134 M: 3d 2-node prismatic joint-with-driver element with 6 dof/node ([Figure 25](#)) and a relative translation (between node 2 and node 1) along the joint axis of translation that is prescribed using a time-dependent law specified in [MATERIAL](#) (for static analysis, the relative translation is set to the initial value

of the chosen law). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x axis defines the orientation in the undeformed configuration of the axis of prescribed relative translation.

- 177 M: 3d 2-node point-to-moving-point constraint element with 3 dof/node.
178 M: 3d 3-node point-to-moving-line constraint element with 3 dof/node.
179 M: 3d 4-node point-to-moving-plane constraint element with 3 dof/node.
201 M: 3d 2-node uniaxial translational spring with 6 dof/node for linear or nonlinear analyses. The initial orientation of the spring axis is the local x -axis of the element frame. It must be specified under [EFRAMES](#), using the same format as for a beam element.
202 M: 3d 2-node uniaxial torsional spring with 3 rotational dof/node for linear or nonlinear analyses. The initial orientation of the spring axis is the local x -axis of the element frame. It must be specified under [EFRAMES](#), using the same format as for a beam element.
220 M: 3d 2-node spherical joint spring combination with 6 dof/node ([Figure 21](#)). This element is equivalent to one spherical joint element (type 120) and three embedded torsional spring elements (type 202). The initial orientation of the first embedded spring's axis is the local x -axis of the element frame. The initial orientation of the second embedded spring's axis is the local y -axis of the element frame. The initial orientation of the third embedded spring's axis is the local z -axis of the element frame. This information must be specified under [EFRAMES](#), using the same format as for a beam element.
221 M: 3d 2-node translational joint spring combination element with 6 dof/node ([Figure 28](#)). This element is equivalent to one translational joint element (type 121) and three embedded translational spring elements (type 201). The initial orientation of the first embedded spring's axis is the local x -axis of the element frame. The initial orientation of the second embedded spring's axis is the local y -axis of the element frame. The initial orientation of the third embedded spring's axis is the local z -axis of the element frame. This information must be specified under [EFRAMES](#), using the same format as for a beam element.
222 M: 3d 2-node universal joint spring combination element with 6 dof/node. This element is equivalent to one universal joint element (type 122) and two embedded torsional spring elements (type 202). The initial orientation of the first embedded spring's axis is the local y -axis of the element frame. The initial orientation of the second embedded spring's axis is the local z -axis of the element frame. This information must be specified under [EFRAMES](#), using the same format as for a beam element.
223 M: 3d 2-node revolute joint spring combination element with 6 dof/node. This element is equivalent to one revolute joint element (type 123) and one embedded torsional spring element (type 202). The initial orientation of the embedded spring's axis is the local x -axis of the element frame. This information must be specified under [EFRAMES](#), using the same format as for a beam element.
224 M: 3d 2-node cylindrical joint spring combination element with 6 dof/node. This element is equivalent to one cylindrical joint element (type 124), one embedded

- translational spring element (type 201), and one embedded torsional spring element (type 202). The initial orientation of the first (translational) embedded spring's axis is the local x -axis of the element frame. The initial orientation of the second (torsional) embedded spring's axis is also the local x -axis of the element frame. This information must be specified under [EFRAMES](#), using the same format as for a beam element.
- 225 M: 3d 2-node prismatic joint spring combination element with 6 dof/node. This element is equivalent to one prismatic joint element (type 125) and one embedded translational spring element (type 201). The initial orientation of the embedded spring's axis is the local x -axis of the element frame. This information must be specified under [EFRAMES](#), using the same format as for a beam element.
- 226 M: 3d 2-node revolute joint-with-actuator element with 6 dof/node. This element is equivalent to one revolute joint spring combination element (type 223) and two equal and opposite embedded follower (see [FORCES](#)) moments applied to the two nodes of the element. The magnitude of these moments is prescribed using a time-dependent law specified in [MATERIAL](#) (for static analysis, the magnitude of the moments is set to the initial value of the chosen law). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x -axis defines the orientation in the undeformed configuration of the axis about which the embedded moment applied to the second node of the element acts, and also the initial orientation of the embedded spring's axis.
- 227 M: 3d 2-node pin-in-slot joint spring combination element with 6 dof/node ([Figure 29](#)). This element is equivalent to one pin-in-slot joint element (type 127), one embedded translational spring element (type 201), and one embedded torsional spring element (type 202). The initial orientation of the first embedded (translational) spring's axis is the local x -axis of the element frame. The initial orientation of the second embedded (torsional) spring's axis is the local y -axis of the element frame.
- 234 M: 3d 2-node prismatic joint-with-actuator element with 6 dof/node. This element is equivalent to one prismatic joint spring combination element (type 225) and two equal and opposite embedded follower (see [FORCES](#)) forces applied to the two nodes of the element. The magnitude of these forces is prescribed using a time-dependent law specified in [MATERIAL](#) (for static analysis, the magnitude of the forces is set to the initial value of the chosen law). This element has a co-rotating reference frame attached to each of its two nodes. Both frames have the same orientation in the undeformed configuration and therefore can be viewed as far as input is concerned as the same frame. The orientation of this frame in the undeformed configuration must be specified under [EFRAMES](#), using the same format as for a beam element. Its x -axis defines the orientation in the undeformed configuration of the direction in which the embedded force applied to the second node of the element acts, and also the initial orientation of the embedded spring's axis.
- 301 F: 2d 4-node sloshing (fluid) quadrilateral element with 1 dof/node.
- 302 F: 2d 2-node free-surface (fluid) element with 1 dof/node for two-dimensional sloshing computations using element type = 301.

311 F: 3d 4-node sloshing (fluid) tetrahedron element with 1 dof/node.
312 F: 3d 3-node free-surface (fluid) triangular element with 1 dof/node for three-dimensional sloshing computations using element type = 311.
321 F: 2d 4-node hydroelastic vibration (fluid) quadrilateral element with 1 dof/node.
331 F: 3d 4-node hydroelastic vibration (fluid) tetrahedral element with 1 dof/node.
1100 A: 2d 4-node Helmholtz DGM element Q-4-1.
1101 A: 2d 4-node Helmholtz DGM element Q-8-2.
1102 A: 2d 4-node Helmholtz DGM element Q-16-4.
1103 A: 2d 4-node Helmholtz DGM element Q-32-8.
1110 A: 2d 3-node Helmholtz DGM element T-4-1.
1111 A: 2d 3-node Helmholtz DGM element T-8-2.
1120 A: 2d 4-node Helmholtz DEM element Q-4-1.
1121 A: 2d 4-node Helmholtz DEM element Q-8-2.
1122 A: 2d 4-node Helmholtz DEM element Q-16-4.
1123 A: 2d 4-node Helmholtz DEM element Q-32-8.
1130 A: 2d 3-node Helmholtz DEM element T-4-1.
1131 A: 2d 3-node Helmholtz DEM element T-8-2.
1150 A: 3d 8-node Helmholtz DGM element H-6-1.
1151 A: 3d 8-node Helmholtz DGM element H-26-4.
1152 A: 3d 8-node Helmholtz DGM element H-56-8.
1153 A: 3d 8-node Helmholtz DGM element H-98-12.
1160 A: 2d 3-node Helmholtz DGM element T-6-1.
1161 A: 2d 3-node Helmholtz DGM element T-26-4.
1162 A: 2d 3-node Helmholtz DGM element T-56-8.
1170 A: 3d 8-node Helmholtz DEM element H-6-1.
1171 A: 3d 8-node Helmholtz DEM element H-26-4.
1172 A: 3d 8-node Helmholtz DEM element H-56-8.
1173 A: 3d 8-node Helmholtz DEM element H-98-12.
1200 M: 2d 4-node Elastodynamic DGM element Q-4x2-2.
1201 M: 2d 4-node Elastodynamic DGM element Q-16x2-8.
1220 M: 2d 4-node Elastodynamic DEM element Q-4x2-2.
1250 M: 3d 4-node Elastodynamic DGM element H-6x3-3.
1251 M: 3d 4-node Elastodynamic DGM element H-26x3-15.
1252 M: 3d 4-node Elastodynamic DGM element H-50x3-28.

These should be listed in a stacked fashion on a single line.

CONNECTIVITY_NODES

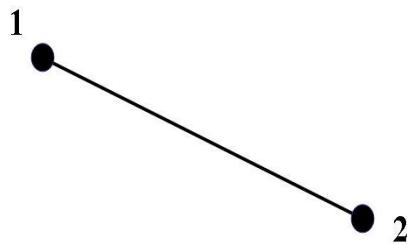


Figure 1: local node numbering for element types **1, 6, 7, 9, 12, 26, 56, 65--69, 75, 111, 301**

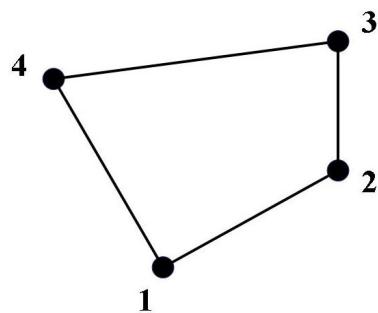


Figure 2: local node numbering for element types **2, 3, 88, 10, 13, 1515, 16, 18, 2020, 30, 31, 33, 34, 4646, 48, 58, 76, 81, 128, 301, 321**

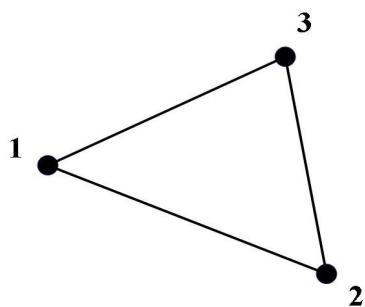


Figure 3: local node numbering for element types **4, 8, 15, 16, 19, 20, 35, 36, 46, 49, 53, 55, 73, 129, 312**

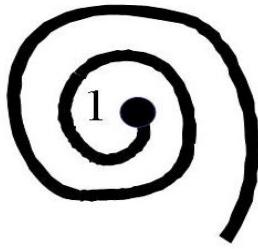


Figure 4: local node numbering for element type 11

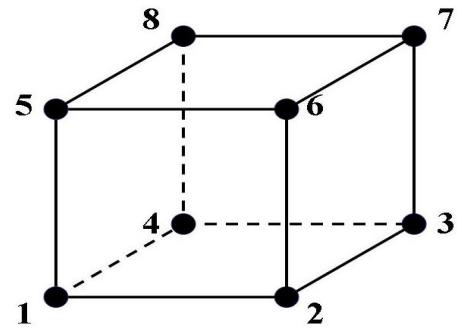


Figure 5: local node numbering for element types 17, 44, 45, 51, 70, 82, 201, 202

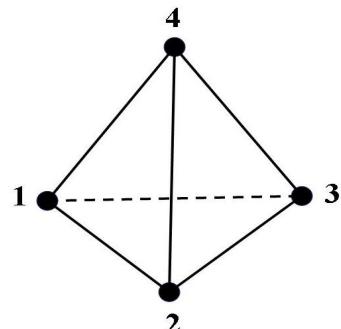


Figure 6: local node numbering for element types 23, 40, 41, 311, 331

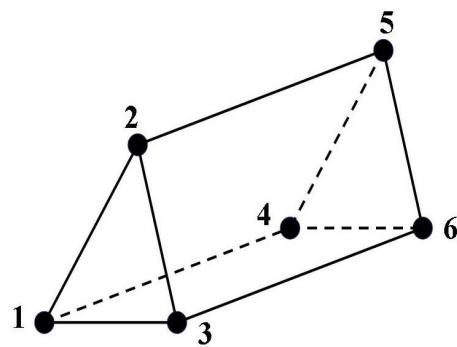


Figure 7: local node numbering for element types 24, 83, 90

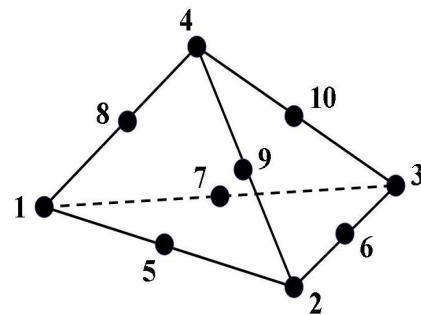


Figure 8: local node numbering for element types 25, 42

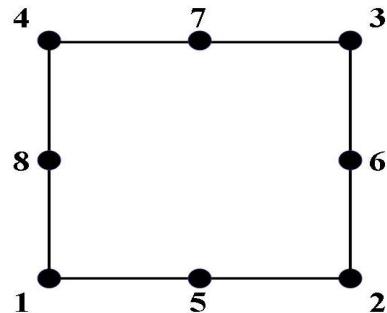


Figure 9: local node numbering for element type 32

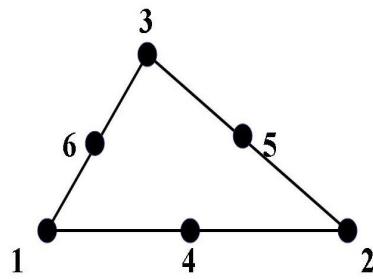


Figure 10: local node numbering for element types 38, 52

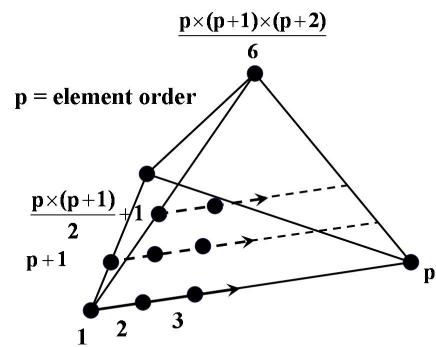


Figure 11: local node numbering for element types 50, 96, 103

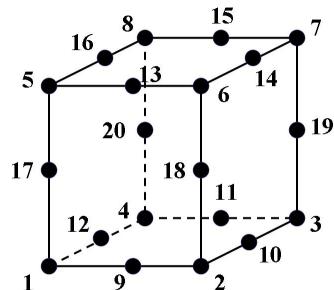


Figure 12: local node numbering for element types 71, 72

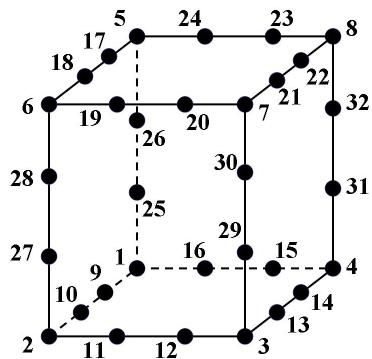


Figure 13: local node numbering for element types 74, 91, 93

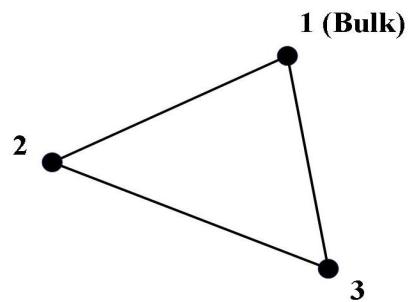


Figure 14: local node numbering for element type 84

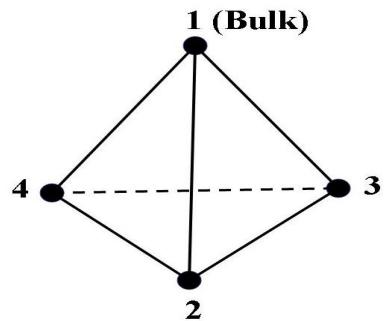


Figure 15: local node numbering for element type 85

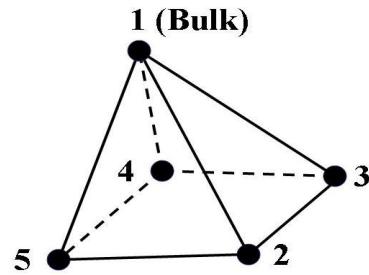


Figure 16: local node numbering for element type 86

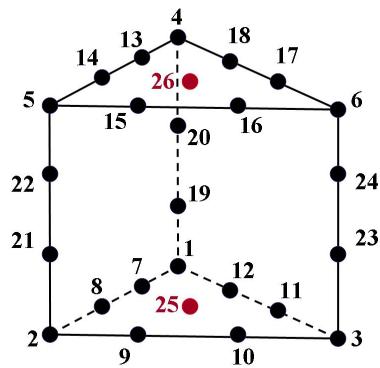


Figure 17: local node numbering for element types 92, 94

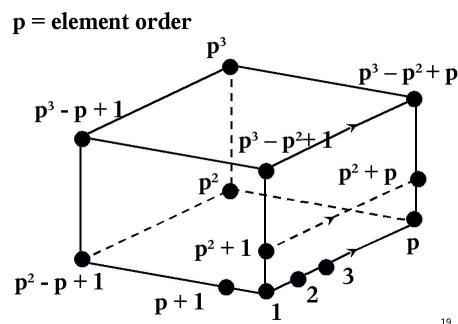


Figure 18: local node numbering for element types 95, 102, 105, 109

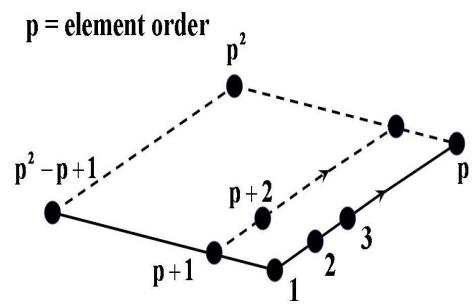


Figure 19: local node numbering for element types 98, 100, 108

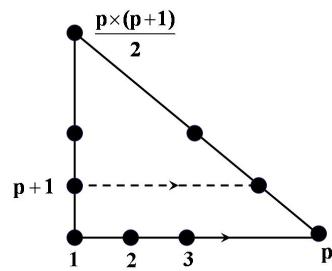


Figure 20: local node numbering for element types 99, 101

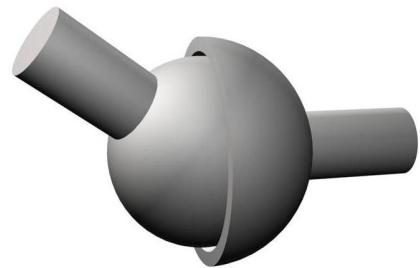


Figure 21: element type 120, 220

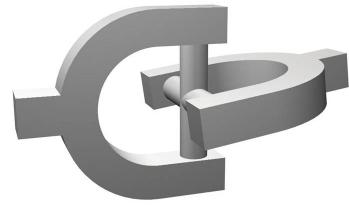


Figure 22: element type 122

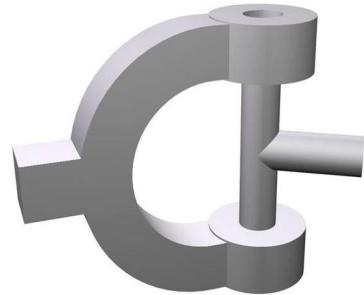


Figure 23: element types 123, 126



Figure 24: element type 124



Figure 25: element types 125, 134

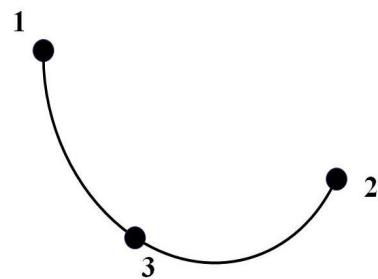


Figure 26: element type 126

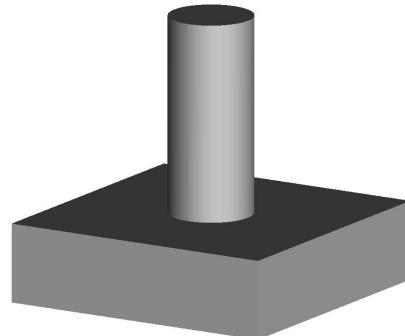


Figure 27: element type 118

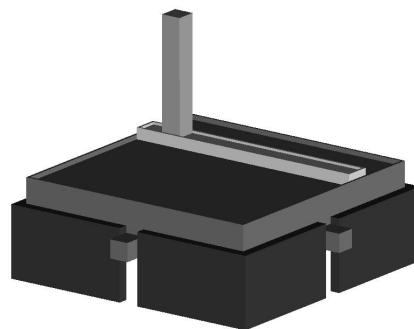


Figure 28: element types 121, 221



Figure 29: element types 127, 227

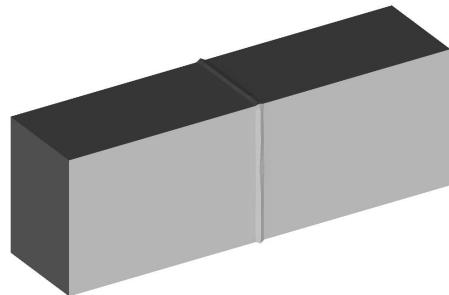


Figure 30: element type 119

Next: [USDD](#), Previous: [TOPOLOGY](#)

103 USER DEFINED FORCES *S*

Command Statement:	USDF
--------------------	-------------

The main purpose of the **usdf** command is to specify nodal forces and/or moments of the *follower* or *axial* type for a structural model via a user-defined subroutine. In this case, the user should: (1) write his/her own algorithm for specifying the nodal forces and/or moments within a subroutine

named `control.c` (see [APPENDIX 10](#)), (2) compile this subroutine and link it using the makefile provided for this command, and (3) use [LOAD](#) to activate it.

By default, all user-defined forces and moments are interpreted as being of the *axial* type — that is, as being defined in the *fixed* nodal degree of freedom reference frames (see [NODES](#) and [NFRAMES](#)). However, if a node has rotational degrees of freedom, the user can specify that the forces and/or moments prescribed at this node are of the *follower* type — that is, they act in a direction that remains constant in the local frame attached to the node where they are applied. This local frame coincides with the nodal degree of freedom reference frame (see [NODES](#) and [NFRAMES](#)) in the undeformed configuration. In the deformed configuration, the orientation of this local frame is defined by the rotation of the node to which it is attached. In other words, the specified nodal force or moment "follows" in this case the rotation of the node to which it is applied.

User-defined nodal forces and/or moments are not assembled neither at the element level nor at the subdomain level. They can be combined with those specified under the [FORCES](#) and [GRAVITY](#) commands, but will not be amplified by the [MFTT](#) or [HFTT](#) tables. In case of a conflict between this command and the [FORCES](#) and [GRAVITY](#) commands, the specified nodal forces are simply added. A more detailed description of the `control.c` subroutine is given in the form of a template in [APPENDIX 10](#).

Note 1: By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: All forces must be specified in the computational basis.

Note 3: The `USDF` command is redundant with the [ACTUATORS](#) command in the sense that [ACTUATORS](#) can achieve whatever `USDF` can achieve. However, the reverse is not true.

Note 4: Similarly, this command can be used to prescribe time-variant, Neumann boundary conditions (or source terms) for a time-domain acoustic simulation.

Note 5: Specifying a follower force or moment leads to an unsymmetric tangent "load" stiffness matrix during a [NONLINEAR](#) analysis.

The syntax for invoking this option is given below.

`USDF`

NODE#	DOF#	TYPE
-------	------	------

NODE#	Node number where the force/moment is specified (integer).	
DOF#	Degree of freedom local number where the force/moment is specified (integer).	

TYPE For structural models, all user-defined nodal sources (forces and/or moments in the case of a structural model) are by default of the axial type. However, if this parameter is set to FOLLOWER and the node **NODE#** has rotational degrees of freedom, the user-defined source (force or moment in the case of a structural model) at this node and the degree of freedom **DOF#** is considered to be of the follower type (characters).

Next: [WEIGHTS](#), Previous: [USDF](#)

104 USER DEFINED PRESCRIBED DISPLACEMENTS *S*

Command Statement: **USDD**

The purpose of the **usdd** command is to specify time-variant prescribed displacements for any node and its associated dofs using a user-defined subroutine. In this case, the user should: 1) write his/her own algorithm for specifying the prescribed displacement field within a subroutine named "control.C", 2) compile this subroutine and link it with **AERO-S** using the makefile that is provided for this purpose, and 3) use the **LOAD** command (see [LOAD](#)) to activate it. An example input file using the **usdd** command can be found in FEM.d/fem\examples/USDD.d/.

This command can be used simultaneously with the time-invariant command [DISPLACEMENTS](#) for prescribing displacements at a node. In the event of a conflict between the [DISPLACEMENTS](#) and **usdd** commands, **usdd** prevails. A more detailed description of the "control.C" subroutine is given in its template located in the Control.d directory of **AERO-S**.

The syntax for invoking this option is given below.

Note 1: All degrees of freedom referred to by this command are defined in the nodal degree of freedom reference frames defined at the nodes where these degrees of freedom are attached (see [NODES](#) and [NFRAMES](#)). By default, the nodal degree of freedom reference frames are the same as the global reference frame.

Note 2: Similarly, this command can be used to specify time-variant prescribed Dirichlet boundary conditions for a time-domain acoustic simulation by setting **DOF#** to 8 (see below).

USDD

NODE# **DOF#**

NODE#
DOF#

Node number where the displacement is specified (integer).
Degree of freedom local number where the displacement is specified (integer).

Next: [YMTT](#), Previous: [USDD](#)

105 WEIGHTS

Command Statement:	WEIGHTS
--------------------	----------------

The **WEIGHTS** command statement can be used to modify some or all of the weights attributed by default to an element type (see [TOPOLOGY](#)). These weights are exploited by the mesh partitioning algorithm (see [DECOMPOSE](#)) to achieve load balance when generating the subdomains.

Note 1: Variable weights attributed by default to elements with a variable number of nodes cannot be modified by this command.

The input format of this command is given below.

WEIGHTS

ELEMENT_TYPE#	DESIRED_WEIGHT
----------------------	-----------------------

ELEMENT_TYPE#	This is the element type id number as in the TOPOLOGY command (integer).
DESIRED_WEIGHT	This is the desired weight to be attributed to this element type (integer).

The weight default values are as follows:

M=Mechanic H=Heat C=Coupled Thermoelastic F=Fluid A=Acoustic W = default weight

- | | |
|----|--|
| 1 | M: 3d truss (bar) element with 3 dof/node (W = 1). |
| 2 | M: 2d 4-node quad element with 2 dof/node (W = 2). |
| 3 | H: 3d 4-node quad element with 1 dof/node (W = 2). |
| 4 | M: 2d 3-node triangular element with 2 dof/node (W = 2). |
| 6 | M: 3d Bernoulli beam element with 6 dof/node (W = 1). |
| 7 | M: 3d Timoshenko beam element with 6 dof/node (W = 1). |
| 8 | M: 3d 3-AQR shell element with 6 dof/node (W = 3). |
| 88 | M: 3d 4-node shell element element with 6 dof/node (splits into 2 elements of type 8) (W = 4). |
| 9 | H: 3d 2-node lineal element with 1 dof/node (W = 1). |
| 10 | H: 2d 4-node quad element with 1 dof/node (W = 2). |
| 11 | M: 3d torsion spring element with 3 dof/node (W = 1). |
| 12 | C: 3d thermomechanical truss element with 4 dof/node (W = 1). |
| 13 | C: 2d thermoelastic quad element with 3 dof/node (W = 2). |
| 14 | C: 3d thermoelastic Bernoulli beam element with 7 dof/node (W = 1). |
| 15 | M: 3d 3-node triangular AQR shell element with 6 dof/node and composite as well as nonlinear material capability (W = 3). |

- 1515 M: 3d 4-node quadrilateral AQR shell element with 6 dof/node and composite as well as nonlinear material capability (**W = 4**).
 16 M: 3d Belytschko-Tsay shell element with 6 dof/node (**W = 4**).
 17 M: 3d 8-node brick element with 3 dof/node (**W = 3**).
 18 M: 3d 4-node shear panel element with 3 dof/node (**W = 1**).
 19 M: 3d 3-node membrane element with 6 dof/node (**W = 3**).
 (only in-plane and drilling stiffnesses) (**W = 1**).
 20 M: 3d 3-node composite or orthotropic shell element with 6 dof/node (**W = 3**).
 2020 M: 3d 4-node composite or orthotropic shell element with 6 dof/node (splits into 2 elements of type 20) (**W = 4**).
 21 M: 3d translational spring-link element with 3 dof/node (**W = 1**).
 22 M: 3d torsion spring-link element with 3 dof/node (**W = 1**).
 23 M: 3d 4-node tetrahedral element with 3 dof/node (**W = 3**).
 24 M: 3d 6-node pentahedral element with 3 dof/node (**W = 3**).
 25 M: 3d 10-node tetrahedral element with 3 dof/node (**W = 4**).
 30 A: 2d 4-node quad element with 1 dof/node (**W = 1**).
 31 A: 2d 4-node quad GLS element with 1 dof/node (**W = 1**).
 32* A: 2d 8-node quad element with 1 dof/node (**W = 3**).
 33* A: 2d 4-node quad BUBBLE element with 1 dof/node (**W = 1**).
 34* A: 2d 4-node quad two-level BUBBLE element with 1 dof/node (**W = 1**).
 35 A: 2d 3-node triangular element with 1 dof/node (**W = 1**).
 36 A: 2d 3-node triangular GLS element with 1 dof/node (**W = 1**).
 38* A: 2d 6-node triangular element with 1 dof/node (**W = 3**).
 40 A: 3d 4-node tetrahedral element with 1 dof/node (**W = 2**).
 41 A: 3d 4-node tetrahedral GLS element with 1 dof/node (**W = 2**).
 42 A: 3d 10-node tetrahedral element with 1 dof/node (**W = 3**).
 44 A: 3d 8-node brick GLS element with 1 dof/node (**W = 3**).
 45* A: 3d 8-node brick element with 1 dof/node (**W = 3**).
 46 H: 3d 3-node triangular heat element (**W = 2**).
 4646 H: 3d 4-node triangular heat element (splits into 2 elements of type 46) (**W = 3**).
 47 H: 3d 2-node lineal element with 1 dof/node for boundary convection (**W = 1**).
 48 H: 3d 4-node quad element with 1 dof/node for boundary convection (**W = 2**).
 49 H: 3d 3-node triangular element with 1 dof/node for boundary convection (**W = 2**).
 50 H: 3d 4-node tetrahedral element with 1 dof/node (**W = 3**).
 51 H: 3d 8-node brick element with 1 dof/node (**W = 4**).
 52* M: 3d 6-node triangular shell element (**W = 1**).
 53 H: 2d 3-node triangular heat element (**W = 2**).
 56 H: 3d 2-node heat radiation element (**W = 1**).
 57 H: 3d 3-node triangular heat radiation element (**W = 2**).
 58 H: 3d 4-node quadrilateral heat radiation element (**W = 2**).
 65 M: 3d 2-node rigid truss (bar) element (**W = 1**).
 66 M: 3d 2-node rigid beam element (**W = 1**).
 67 M: 3d rigid link (translational and rotational) (**W = 1**).
 68 M: 3d rigid translational link (**W = 1**).
 69 M: 3d rigid rotational link (**W = 1**).

- 70 M: 3d rigid plane or solid element with 3 dof/node (3-node to 20-node element) (**W = 3**).
 71 M: 3d rigid plane or solid element with 3 dof/node and anywhere from 3 to 20 nodes per element (**W = 1**).
 72 M: 3d 20-node brick element (**W = 4**).
 73 M: 3d 3-node rigid shell element (**W = 3**).
 74 M: 3d rigid plane or solid element with 6 dof/node (3-node to 32-node element) (**W = 1**).
 76 M: 3d 4-node rigid shell element (**W = 4**).
 78 M: 3d 1-node point-to-line constraint element with 3 dof/node (**W = 1**).
 79 M: 3d 1-node point-to-plane constraint element with 3 dof/node (**W = 1**).
 81 H: 2d 4-node contact resistance thermal element with 1 dof/node that can be inserted between two thermal elements of type = 10 (**W = 2**).
 82 H: 3d 8-node contact resistance thermal element with 1 dof/node that can be inserted between two thermal elements of type = 51 (**W = 4**).
 83 H: 3d 6-node contact resistance thermal element with 1 dof/node that can be inserted between two thermal elements of type = 50 with 4 nodes each (**W = 4**).
 84 H: 2d 3-node triangular bulk fluid (thermal) element with 1 dof/node. The first node appearing in the connectivity list of this element must be inside the bulk fluid and the other two nodes must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid (**W = 2**).
 85 H: 3d 4-node tetrahedral bulk fluid (thermal) element with 1 dof/node. The first node appearing in the connectivity list of this element must be inside the bulk fluid and the other three nodes must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid (**W = 3**).
 86 H: 3d 5-node pyramidal bulk fluid (thermal) element with 1 dof/node. The first node appearing in the connectivity list of this element must be inside the bulk fluid and the other four nodes must be located at the interface boundary between the bulk fluid and the surrounding material. The temperature at the first node is by definition the average temperature of the bulk fluid (**W = 4**).
 90 A: 3d 6-node wedge element with 1 dof/node (**W = 3**).
 91 M: 3d 32-node serendipity brick element with 3 dof/node (**W = 6**).
 92 M: 3d 26-node serendipity wedge element with 3 dof/node (**W = 5**).
 93 A: 3d 32-node serendipity brick element with 1 dof/node (**W = 5**).
 94 A: 3d 26-node serendipity wedge element with 1 dof/node (**W = 4**).
 95 A: 3d 8-node, or 27-node, or 64-node, or 125-node hexahedral element with 1 dof/node (**W = 3, 4, 5, 6**).
 96 A: 3d 4-node, or 10-node, or 20-node, or 35-node tetrahedral element with 1 dof/node (**W = 1, 2, 3, 4**).
 97 M: 3d 15-node serendipity wedge element with 3 dof/node (**W = 4**).
 98 A: 2d 4-node, or 9-node, or 16-node, or 25-node quadrilateral element with 1 dof/node (**W = 2, 3, 4, 5**).
 99 A: 2d 3-node, or 6-node, or 10-node triangular element with 1 dof/node (**W = 2, 3, 4**).
 100 M: 2d 4-node, or 9-node, or 16-node, or 25-node quadrilateral element with 3 dof/node (**W = 2, 3, 4, 5**).

- 101 M: 2d 3-node, or 6-node, or 10-node triangular element with 3 dof/node (**W = 2, 3, 4**).
102 M: 3d 8-node, or 27-node, or 64-node, or 125-node hexahedral element with 3 dof/node (**W = 2, 3, 4, 5**).
103 M: 3d 4-node, or 10-node, or 20-node, or 35-node tetrahedral element with 3 dof/node (**W = 1, 2, 3, 4**).
105 A: 3d 8-node, or 27-node, or 64-node, or 125-node spectral hexahedral element with 1 dof/node (**W = 2, 3, 4, 5**).
108 A: 2d 4-node, or 9-node, or 16-node, or 25-node spectral quadrilateral element with 1 dof/node (**W = 2, 3, 4, 5**).
109 H: 3d 8-node, or 27-node, or 64-node, or 125-node hexahedral element with 1 dof/node (**W = 2, 3, 4, 5**).
111 M: 3d fabric truss element with 3 dof/node (**W = 1**).
118 M: 3d 2-node planar joint element with 6 dof/node (**W = 1**).
120 M: 3d 2-node spherical joint element with 3 dof/node (**W = 1**).
121 M: 3d 2-node translational joint element with 3 rotational dof/node (**W = 1**).
122 M: 3d 2-node universal joint element with 6 dof/node (**W = 1**).
123 M: 3d 2-node revolute joint element with 6 dof/node (**W = 1**).
124 M: 3d 2-node cylindrical joint element with 6 dof/node (**W = 1**).
125 M: 3d 2-node prismatic joint element with 6 dof/node (**W = 1**).
126 M: 3d 2-node joint-with-driver element with 6 dof/node (**W = 1**).
127 M: 3d 2-node pin-in-slot joint element with 6 dof/node (**W = 1**).
128 M: 3d 4-node plane stress/plane strain quadrilateral element with 3 dof/node (**W = 1**).
129 M: 3d 3-node plane stress/plane strain triangular element with 3 dof/node (**W = 1**).
134 M: 3d 2-node prismatic joint-with-driver element and 6 dof/node (**W = 1**).
177 M: 3d 2-node point-to-moving-point constraint element with 3 dof/node (**W = 1**).
178 M: 3d 3-node point-to-moving-line constraint element with 3 dof/node (**W = 1**).
179 M: 3d 4-node point-to-moving-plane constraint element with 3 dof/node (**W = 1**).
179 M: 3d 4-node point-to-moving-plane constraint element with 3 dof/node (**W = 1**).
201 M: 3d 2-node uniaxial translational spring with 6 dof/node for linear or nonlinear analyses (**W = 3**).
202 M: 3d 2-node uniaxial torsional spring with 3 rotational dof/node for linear or nonlinear analyses (**W = 3**).
220 M: 3d 2-node spherical joint spring combination with 6 dof/node (**W = 1**).
221 M: 3d 2-node translational joint spring combination element with 6 dof/node (**W = 1**).
222 M: 3d 2-node universal joint spring combination element with 6 dof/node (**W = 1**).
223 M: 3d 2-node revolute joint spring combination element with 6 dof/node (**W = 1**).
224 M: 3d 2-node cylindrical joint spring combination element with 6 dof/node (**W = 1**).
225 M: 3d 2-node prismatic joint spring combination element with 6 dof/node (**W = 1**).
226 M: 3d 2-node revolute joint-with-actuator element and 6 dof/node (**W = 1**).
227 M: 3d 2-node pin-in-slot joint spring combination element with 6 dof/node (**W = 1**).
234 M: 3d 2-node prismatic joint-with-actuator element and 6 dof/node (**W = 1**).
301 F: 2d 4-node sloshing (fluid) quadrilateral element with 1 dof/node (**W = 1**).

- 302 F: 2d 2-node free-surface (fluid) element with 1 dof/node for two-dimensional sloshing computations using element type = 301 (**W = 1**).
 311 F: 3d 4-node sloshing (fluid) tetrahedron element with 1 dof/node (**W = 3**).
 312 F: 3d 3-node free-surface (fluid) triangular element with 1 dof/node for three-dimensional sloshing computations using element type = 311 (**W = 2**).
 321 F: 2d 4-node hydroelastic vibration (fluid) quadrilateral element with 1 dof/node (**W = 2**).
 331 F: 3d 4-node hydroelastic vibration (fluid) tetrahedral element with 1 dof/node (**W = 3**).
 1100 A: 2d 4-node Helmholtz DGM element Q-4-1 (**W = 1**).
 1101 A: 2d 4-node Helmholtz DGM element Q-8-2 (**W = 2**).
 1102 A: 2d 4-node Helmholtz DGM element Q-16-4 (**W = 4**).
 1103 A: 2d 4-node Helmholtz DGM element Q-32-8 (**W = 8**).
 1110 A: 2d 3-node Helmholtz DGM element T-4-1 (**W = 1**).
 1111 A: 2d 3-node Helmholtz DGM element T-8-2 (**W = 2**).
 1120 A: 2d 4-node Helmholtz DEM element Q-4-1 (**W = 1**).
 1121 A: 2d 4-node Helmholtz DEM element Q-8-2 (**W = 2**).
 1122 A: 2d 4-node Helmholtz DEM element Q-16-4 (**W = 4**).
 1123 A: 2d 4-node Helmholtz DEM element Q-32-8 (**W = 8**).
 1130 A: 2d 3-node Helmholtz DEM element T-4-1 (**W = 1**).
 1131 A: 2d 3-node Helmholtz DEM element T-8-2 (**W = 2**).
 1150 A: 3d 8-node Helmholtz DGM element H-6-1 (**W = 1**).
 1151 A: 3d 8-node Helmholtz DGM element H-26-4 (**W = 4**).
 1152 A: 3d 8-node Helmholtz DGM element H-56-8 (**W = 8**).
 1153 A: 3d 8-node Helmholtz DGM element H-98-12 (**W = 12**).
 1160 A: 2d 3-node Helmholtz DGM element T-6-1 (**W = 1**).
 1161 A: 2d 3-node Helmholtz DGM element T-26-4 (**W = 4**).
 1162 A: 2d 3-node Helmholtz DGM element T-56-8 (**W = 8**).
 1170 A: 3d 8-node Helmholtz DEM element H-6-1 (**W = 1**).
 1171 A: 3d 8-node Helmholtz DEM element H-26-4 (**W = 4**).
 1172 A: 3d 8-node Helmholtz DEM element H-56-8 (**W = 8**).
 1173 A: 3d 8-node Helmholtz DEM element H-98-12 (**W = 12**).
 1200 M: 2d 4-node Elastodynamic DGM element Q-4x2-2 (**W = 2**).
 1201 M: 2d 4-node Elastodynamic DGM element Q-16x2-8 (**W = 8**).
 1220 M: 2d 4-node Elastodynamic DEM element Q-4x2-2 (**W = 2**).
 1250 M: 3d 4-node Elastodynamic DGM element H-6x3-3 (**W = 3**).
 1251 M: 3d 4-node Elastodynamic DGM element H-26x3-15 (**W = 15**).
 1252 M: 3d 4-node Elastodynamic DGM element H-50x3-28 (**W = 28**).

Previous: [WEIGHTS](#)

106 YOUNGS MODULUS TEMPERATURE TABLE

Command Statement:	YMTT
--------------------	-------------

The **YMTT** command statement can be used to describe the evolution of Young's modulus with temperature, for a given material. This evolution can be specified here in a curve (or table) defined

by pairs of temperature and Young's modulus values. Linear interpolation is used for "in between" points, and the extrema values are adopted for "outside" points. Several curves can be specified, one after the other. Each curve is identified by an "id number" as described below.

YMTT

```
CURVE curve_id
T_1  YM_1
.
.
.
T_1  YM_1
```

CURVE
curve_id
T_1
YM_1

"Id number" for the following curve (or table) (integer).

A specified temperature value (float).

A specified Young's modulus value at temperature T_1 (float).