Advanced CAE Applications for Professionals

Software that works — for you.[SM]

ASTROS
User's Reference Manual
for Version 20

UNIVERSAL ANALYTICS, INC.

First Edition, March 1997
Second Edition, November 1997

UNIVERSAL ANALYTICS, INC.

3625 Del Amo Blvd., Suite 370
Torrance, CA 90503
Tel: (310) 214-2922
FAX: (310) 214-3420

TABLE OF CONTENTS

This page is intentionally blank.

LIST OF FIGURES

LIST OF TABLES

This page is intentionally blank.

Chapter 1

INTRODUCTION

There are five manuals documenting ASTROS, the Automated Structural Optimization System:

- The User's Reference Manual
- The Theoretical Manual
- The Programmer's Manual
- The ASTROS eBASE Schemata Definition
- The Installation and System Support Manual

This User's Manual provides a complete description of the user interface to the ASTROS system in order to facilitate the preparation of input data. It introduces the features of the ASTROS system that enable the user to direct the software system and documents the mechanisms by which the user can communicate with the system. It is assumed that the reader is familiar, from a study of the Theoretical Manual, with the engineering capabilities of the ASTROS system and is using this manual to define the form of the particular input that directs the system to perform a desired function.

The Theoretical Manual describes the range of capabilities of the ASTROS system, while the Programmer's Manual is provided to give details of the internal function of the engineering and programming utility modules. The eBASE Schemata Manual documents all of the database entities. The Installation and System Support Manual describes how ASTROS is installed on host computers, and how it may be configured for customized use.

This manual is intended to provide the user with a convenient reference for all forms of input to the system and is therefore organized along the same lines as the input data stream. The discussion of each topic is brief and generic and is followed by detailed documentation of the user inputs. Information on ASTROS output formats is in a separate chapter as is the description of the Matrix Analysis Problem Oriented Language (MAPOL) used for programming ASTROS.

Finally, this manual is directed toward the engineer/designer/analyst who is using ASTROS to perform engineering design or analysis. While ASTROS is perfectly capable of performing many tasks not explicitly supported in the standard execution, the user must know the engineering software in considerable detail to direct the system to perform these alternative functions. The mechanisms by which these more advanced features are invoked are included in this manual but no attempt is made to provide sufficient information to the user to generate new analysis features or to grossly modify the existing capabilities of the system. These more advanced topics are treated in the Programmer's Manual which documents the individual modules in the system and their interactions. Rudimentary modifications to the execution sequence and changes to execution parameters are discussed in detail in this manual.

Machine and installation-dependent aspects of ASTROS are also contained in the Programmer's Manuals rather than in the User's Manual. Only those machine-dependent issues that are logically related to the preparation of the input are discussed in this manual. Machine-dependencies in the input are limited to the naming conventions for the run time database files and the parameters that can be used on the **ASSIGN DATABASE** entry. Other machine dependencies are handled as part of the installation of the system on each particular host machine. These issues are documented in the Programmer's Manual since they are relevant only to the *system manager*, not to the *user*.

It will be apparent to many readers that the NASTRAN structural analysis system was used as a guide in the design of the ASTROS program. Both NASTRAN and ASTROS comprise large scale, finite element structural analysis in executive driven software systems. Therefore, many of the input and output features are similar. NASTRAN has become an industry-standard in finite element structural analysis with many pre- and post-processors developed around NASTRAN data. To maintain maximal compatibility, many aspects of the ASTROS input are similar in form or purpose to those in NASTRAN and, in many other cases, the same nomenclature has been adopted. In some instances in this document, therefore, ASTROS input will be compared and contrasted to NASTRAN input in order to present a concise picture of the ASTROS input and to assist the reader familiar with NASTRAN in making the connection to the equivalent item in ASTROS. Although familiarity with NASTRAN is not a prerequisite to understanding the ASTROS documentation, sufficient numbers of potential ASTROS users are expected to be familiar with the NASTRAN system to justify the sometimes casual reference to NASTRAN features.

Chapter 1 contains a description of the ASTROS input file, database assignment and debug control inputs. Chapters 2 through 5 are organized to parallel the input file structure. Within each of these chapters, the function of the particular input packet is presented along with illustrations of how the data are prepared. Each packet is described in a generic fashion so as to indicate how the sophisticated user can make use of the more advanced features of the system without cluttering the discussion with details of the input structures. The detailed documentation of the separate input structures of the data packet then follow within each Chapter. This form of documentation enables this manual to be useful as a guide to the beginning user as well as a reference for the experienced user. While there are a number of advanced input features, the required input for most jobs is the **ASSIGN DATABASE** command, described in Section 1.3, and the Solution Control and Bulk Data packets described in Chapters 3 and 5, respectively.

In Chapter 6, following the input stream descriptions, the output features of the ASTROS system are documented. While these features are selected through directives in the input data stream, they are sufficiently numerous and complex to justify a separate chapter devoted solely to output requests. The

output capabilities of the system are described in very general terms while the output requests available for each analysis discipline and optimization feature are documented in detail. Most output is selected through Solution Control directives that are documented in Chapter 3, but some are selected through modifications to the executive (MAPOL) sequence. Chapter 2 documents all of the output utilities available to the user through MAPOL directives and gives several examples of modifying the MAPOL sequence to obtain additional output. Other features are described in the MAPOL Programmer's Manual which comprises Chapter 7.

Many examples of user input are used throughout this document. In order to ease the burden of interpretation, the conventions shown in Table 1-1 are used in the examples unless otherwise noted. Chapter 7, which describes the MAPOL programming interface, describes additional conventions required for the programming syntax of MAPOL.

Table 1-1. Command Syntax Conventions

`MAPOL NOGO`	Capital letters indicate that the phrase must appear exactly as shown
`MAPOL` *params*	Lower case italic symbols act as generic place holders indicating that an option or options can or must be included
`MAPOL` [**GO** / NOGO]	Symbol(s) enclosed in brackets [ ] are optional. If more than one symbol is available they will be stacked in vector notation with any defaults denoted by boldface.
`INCLUDE <`*filename*`>`	A required symbol is enclosed in angle brackets. If the angle brackets surround an option list, at least one of the available options must be selected.
`BEGIN_BULK`	The underscore (_) is used to signify a required blank space.

This page is intentionally blank.

Chapter 2

RUNNING ASTROS

As is the case with all major software systems that are available across a broad spectrum of host computers and operating systems† ASTROS has features which are implemented differently on different computers. The most common differences are in the way you execute ASTROS and other UAI software products, the management of dynamic memory, and the manner in which files are handled during execution. This Chapter describes these for the most commonly used operating systems.

†All computer models and operating system names are trademarks of their respective manufacturers and vendors.

2.1.OVERVIEW

This section provides you with an overview of the areas of ASTROS that are directly affected by your host computer and its operating system.

2.1.1.Executing ASTROS

The manner in which you invoke a ASTROS execution is completely dependent on the operating system of your host computer. Subsequent sections of this chapter describe this operation for the most common host computers upon which ASTROS is currently available. You will note that Section 2.2 includes all of the host computers using the Unix operating system and its derivatives.

2.1.2.The ASTROS Configuration and Preference Files

In general, UAI's suite of engineering software products uses computing resources intensively. As a result, there are a number of parameters that must be set to achieve optimal resource management on a given host computer. These parameters, taken as a group, are called the ***Configuration*** of the products. The configuration is provided through one or more files. These files include parameters which are used for controlling database locations, physical file characteristics, memory utilization, and algorithm control.

For maximum flexibility, configurations may be controlled by the site, i.e. the UAI support specialist, or the end user. Many different configurations may be defined for a site. For example, when configuring ASTROS, the UAI support specialist may create different configurations for very small and for very large analyses.

The starting point for configuring the UAI products is the ***Default Preference File***, `uaidef`, included in your delivery. The other modifications described above are made in other Preference Files. The actual configuration used for a given execution is determined by applying the specified Preference Files in the following sequence:

- First, the Default Preference File is processed and all parameters included in this file are set to their specified values
- Second, the System Preference File is processed, and any parameters included in it replace those previously defined
- Third, the User Preference File is processed, and again, any parameters included in it replace those previously defined.

In summary, the final configuration is the union of the Preference files. The Default Preference file contains a value for every parameter used by the product suite. The other Preference Files need only contain those parameters that differ from, and override, the default values.

Each Preference File is composed of as many as six ***Sections***:

- The Host Section
- The ***eBase*** Section
- The ***eBase:applib*** Section
- The ***eBase:matlib*** Section
- The ***eShell*** Section
- The ASTROS Section

The format of the Preference File and a brief description of its various sections are described in the following sections.

2.1.2.1. The Format of Preference Files

A Preference File is a text file which is composed of as many as six Sections indicated above. Each Section includes a header followed by the parameters associated with the Section. For ease-of-use, the **[eBase]** and **[ASTROS]** Sections are subdivided into groups which contain related parameters. The form of the file is shown in Table 2-1.

2.1.3.Configuration Parameters

Configuration parameters are defined using one of the forms:

```
param_name = value
param_name = ( value,value,...,value )
```

The *param_names* are case-insensitive. The *values*, when character strings or floating point numbers with exponents, are also case-insensitive unless they are enclosed in single quotations (tics) as:

```
param_name = 'This is a Case-Sensitive String'
```

Only one parameter may be specified on each line of the file. Any characters that appear after *value* are treated as commentary and ignored. You may also enter comments into the file by beginning a line with any of the characters $, *, or #.

2.1.4.The Configuration Sections

The following sections provide an overview of the six Configuration Sections. Details of each section, as well as information needed to define specific configuration parameters, are found in the ASTROS ***System Support Manual***. Contact your System Support Specialist if you require this information.

2.1.4.1. The Host Configuration Section

The Host Configuration Section includes parameters which identify the type of the host computer, and specify the Preference File templates.

Table 2-1. The Preference File Format

[Host]
HOST_params
[eBase]
< Computing Resources >
eBase_params
< I/O System Parameters >
eBase_params
< **Program Authorization** >
eBase_params
[eBase:applib]
eBase:applib_params
[eBase:matlib]
eBase:matlib_params
[ASTROS]
< Print File Controls >
ASTROS_params
< Computing Resources >
ASTROS_params
< Matrix Conditioning >
ASTROS_params
< Data Checking >
ASTROS_params
< Analysis Output Control >
ASTROS_params
< Solution Techniques >
ASTROS_params
< Element Options >
ASTROS_params
< I/O System Parameters >
ASTROS_params
< Optimization Control Options >
ASTROS_params
< Program Authorization >
ASTROS_params
[eShell]
eShell_params

2.1.4.2. The eBase Kernel Configuration Section

The ***eBase*** Configuration File Section includes parameters which control the ***eBase*** Engineering Database Management System kernel. These include such information as default paths were databases are stored, physical block sizes for databases, and security information.

2.1.4.3. The ASTROS Configuration Section

The ASTROS Configuration Section includes parameters which control the program. These include controls on peripheral and computing resources, model data checking, program defaults, and so forth.

2.1.4.4. The eBase:applib and eBase:matlib Sections

The ***eBase:applib*** and ***eBase:matlib*** Configuration Sections include such items as dynamic memory sizes for ***applib***, and timing constants for the ***matlib*** high-performance matrix utilities.

2.1.4.5. The eShell Configuration Section

The ***eShell*** Configuration Section includes parameters which control the ***eShell*** interactive interface to ***eBase***. It includes such items as system database locations and dynamic memory specifications.

2.1.5.Dynamic Memory

The architecture of ASTROS allows the modeling and analysis of finite element models of virtually unlimited size. Most numerical calculations perform at maximum efficiency when all data for the operation fits in the ***working memory*** space of the program. Many operations may be performed even when all data that they require does not fit in memory by using what is called ***spill logic***. Spill logic simply involves the paging of data to and from disk storage devices as necessary. For very large jobs, spill commonly occurs. In such cases, providing ASTROS with additional working memory can often improve performance. On the other hand, you do not want to give ASTROS excess memory, because it will reduce resources that could be used for other processes on your system. Under certain circumstances, excess memory may actually degrade the performance of ASTROS and, in extreme cases, even your computer system.

ASTROS has a second independent dynamic memory which is used to operate on databases that are attached to the execution. This memory is typically much smaller than the working memory. The main factor influencing the amount of database memory required is the block size used by the active databases. This is described in detail in subsequent sections.

The working memory for ASTROS is dynamically acquired during execution. The amount of space that is actually used by the program is typically controlled by the ASTROS execution procedure or the **MEMORY** Executive Control command. Some host computers have alternate means of controlling this memory.

2.1.6.The eBase Database

With ASTROS Version 13, UAI introduced the Engineering Database Management System, ***eBase***, into ASTROS. This advanced scientific database technology greatly enhances the data handling capabilities of ASTROS compared with the older CADDB database found in the original ASTROS program.

2.1.6.1. The Two Types of Databases

There are two types of ***eBase*** databases in ASTROS. The first type is the ***run-time database***, or RUNDB. This database is used to store the relations and matrices which are used in performing your analysis task. At the end of your job, the RUNDB may be deleted. The second type is the ***archival database***. This type of database represents any ***eBase*** database that you wish to use during an ASTROS execution. The database may be created by ASTROS, or by a second application which uses the ***eBase applib*** or ***matlib*** Applications Programming Interface (API).

2.1.6.2. The Logical and Physical Views of the Database

To fully understand the database technology, you must understand the two views of the database. Each database is called a ***logical database.*** This term is used because from an engineering viewpoint, the database is a single entity which is used in its entirety. The manner in which the logical database is stored on your host computer depends on the amount of data it contains and the availability of disk storage devices. The physical view is a mapping of a logical database to some number of physical files on your host computer. It may be necessary for you to understand the physical model because, for very large analyses, it may be more efficient to organize the actual files in a manner that allows higher performance on your host.

2.1.6.3. The Physical Model

Each ***eBase*** database, regardless of its use, has two components manifested as a minimum of two physical files. The first of these components is called the INDEX component. This component is always a single physical file. It contains information which identifies and locates actual database entities. These entities themselves are stored in the DATA component. To provide the maximum flexibility for a wide variety of data storage requirements, the data components may be stored in a number of different physical files. Most database systems are organized in this manner, because the index component is generally small in size and referenced often, while the data component may be extremely large and not fit in a single file or even on a single disk drive.

2.1.6.4. ASSIGNing Databases

Each logical database must be defined in the ASTROS job stream. Details of this are found in Chapter 2.

2.1.6.5. Database File Names

The naming of database files follows a convention that is different from that of other **UAI/*NASTRAN*** files. The file names are generated automatically at execution time. The conventions used are also described starting in Section 2.2 of this chapter.

2.1.6.6. Very Large Databases

You may be solving extremely large problems with ASTROS. In such cases it may be possible that a databases exceeds the capacity of a single disk drive. ASTROS has made provision for this and you must contact your UAI System Support Specialist for details describing the use of this advanced feature.

2.1.7.Host Computer Dependencies

The sections that follow provide detailed information describing the differences in ASTROS execution procedures and commands which depend on your host computer system.

2.2.UNIX-BASED COMPUTERS

This section describes the host-dependent information that you need to execute ASTROS on Unix-based computer systems. UAI supports a wide variety of manufacturers including Silicon Graphics, Hewlett Packard, IBM RS/6000, Sun, and more. For a complete list of platforms, please contact your UAI sales representative.

2.2.1.Executing ASTROS

An ***sh*** script file, called **`astros`**, is provided to execute ASTROS. To execute you enter:

```
astros [ –m  memory [ K ][ W ] ] [ -p prefname ]
       [ M ][ P ]
            [ B ]

            [ -e astros_exe ] [ -c config_file ] filelist
```

where *`memory`* specifies the amount of memory that the job will use. Options allow you to use shorthand notation for large values and allocation types. The options **`K`** and **`M`** indicate that the memory value is specified in thousands or millions of units, respectively. The units may be specified in single precision words (**`W`**), bytes (**`B`**), or machine precision words (**`P`**). If none of these arguments are used, then memory is assumed to be single precision words. Chapter 3 has a further discussion of memory allocation. The *`prefname`* specifies the substitution string used to generate Preference File names. When performing software development with ASTROS, there may be several versions of the program. The name *`astros_exe`* specifies the name of the ASTROS executable program version to use. By default, a file named `astros.out` located in the local directory, or the version in the installation directory, will be used. The *`config_file`* allows you to override the default **`uaidef`** file that is referenced through the **`UAICONFIG`** global variable. This may be a useful option if more than one ASTROS system is available at your site. Finally, *`filelist`* specifies a list of one or more file names, separated by spaces, that contain ASTROS input data streams. The actual file names must have the proper trailing component, which is usually **`.d`**. The script file will execute ASTROS using each of the data files that you provide. Examples illustrating the use of the script are shown below.

1. Execute ASTROS using the input file **test.d**

```
astros test
```

2. Execute ASTROS in the background for all of the input files in directory **/astros/demodata**.

```
astros /astros/demodata/*.d &
```

3. Execute ASTROS using the input file **test.d** and request one million words of memory.

```
astros -m 1000000 test or
astros -m 1mw test or
astros -m 1000kw
```

4. Suppose that you have created a Preference File name **my.pref**, execute ASTROS using the input file **test.d** using these preferences.

```
astros -p my test
```

5. Suppose you have created your own version of ASTROS, named *myastros.out*, in your local directory. Execute input file **test.d** using this program version.

```
astros -e myastros test
```

2.2.2.ASTROS File Names

When you execute the **astros** script a number of files may be created which have names that are automatically generated by the program. These are described in this section.

2.2.2.1. Unique ASTROS files

There are three unique files that are used frequently by ASTROS. These are unique in the sense the program will automatically define file names for these if you do not explicitly **ASSIGN** them. These files, and their default names, are shown in the table below:

FILE	May Override with ASSIGN Command?	Generated Name if ASSIGN Command is Not Used
The print file	**NO**	*filename*.**prt**
The log file	**NO**	*filename*.**log**
The **PUNCH** file	**YES**	*filename*.**pch**

The *filename* represents the name of the file containing the ASTROS input data stream. The ***log file*** is a special file that contains the history of your execution. You may monitor the progress of your job by viewing the log file periodically. Upon completion of the job, the log file is appended to the print file, and then deleted.

2.2.2.2. Databases

Each database that you use during an execution is comprised of at least two physical files. The trailing components of these file names is always generated by ASTROS. When you **ASSIGN** a database with a status of **NEW** and provide a physical file name, *phys_name*, the program generates the file names:

phys_name.EDB and *phys_name*.00

There may be times, most often in the case of the **RUNDB**, that you **ASSIGN** a database with a status of **TEMP**. In such cases, the program internally generates file names that are unique to your job. The detailed rules used to generate these names are given in the ***System Support Manual***. These simple rules pertain to the simplest and most used **ASSIGN**ments of databases. If you are using very large databases, then there are additional rules. These will be provided by your ASTROS System Support Specialist.

2.2.3.The eShell Program

If your site has the ***eShell*** interactive ***eBase*** interface program, then to execute this program you enter:

```
eshell [-ps prefname] [-pu prefname]
                      [-pl prefname] [database]
```

where:

prefname Specifies the substitution string used to generate Preference File names. You may specify a different string for the System (**-ps**), the User (**-pu**) and the Local (**-pl**) preference files. If you have the unusual case where all of these files have the same name, you may use the option **-p** followed by the *prefname*.

database Is the name of a database to be opened with read access.

This places you in the command mode. Unless directed otherwise by ***eShell*** commands, all subsequent output will be sent to the terminal device. The optional *prefname* information is an advanced feature used for customizing ***eShell*** which is described in the ***Installation Guide and System Support Manual***.

The ***eShell*** Tutorial Problem library is available. Contact your Systems Support Specialist to obtain the name of the directory where these problems may be found. A description of how you may use them is given in the ***eShell User's Manual.***

2.2.4.Automatic Preference Files

Both the **astros** and **eshell** scripts provide arguments which allow you to specify the substitution string needed to generate Preference File names. If these arguments are not used, both of these scripts look for a Preference file named **uai.pref**. First, the local directory is checked for this file, and, if it is not found, then your home directory is checked. If a file with this name is found, then a substitution string of **uai** is automatically used. If the Preference File **uai.pref** is found in your home directory, then it is copied to your local directory for the duration of your ASTROS job. Upon termination, the copy is deleted.

2.2.5.Online Manuals

The entire suite of ASTROS manuals is available online in the Adobe Portable Document Format (PDF). This allows you to view the documentation on any computer that has the Adobe® Acrobat® Reader 3.0. Readers for the MAC, PC, Sun (OS and Solaris), and HP were delivered with your system.

To use the documents, from the command line you enter:

```
uaidoc [manual_name]
```

If you omit the manual_name, then you will see a splash screen that allows you to navigate to the appropriate manual. You may also go directly to a manual by placing its name on the command lines. The names of the astros manuals are:

- ***astros_theory***
- ***astros_prog***
- ***astros_ref***
- ***astros_schema***
- ***eshell***
- ***system_support***

This page is intentionally blank.

Chapter 3.

THE INPUT DATA STREAM

3.1.INTRODUCTION

The ASTROS user directs the system through an input data stream composed of a ***Resource Section***, which allocates ASTROS databases and specifies memory utilization, which is followed by multiple ***Data Packets***. Each packet contains a set of related data providing the information needed to execute ASTROS. The packets begin with a keyword indicating the nature of the data within the packet and terminate with an ending keyword or with the start of the next data packet. All the packets in the input data stream are optional, although the order in which they must appear is fixed. The purpose of this section is to document the structure of the input data stream. Detailed documentation of the data within each data packet is then presented in separate chapters.

Figure 3-1 shows the general form of the input data stream and Figure 3-2 illustrates the actual input stream features with a sample stream for a ten bar truss model. The first non-blank record of the input file must be either the **ASSIGN** command or a Resource Section. If an **ASSIGN** command is used, then this command will enable you to attach the run-time database(s) that are used during the execution of the ASTROS system. There are four optional data packets following the **ASSIGN** command which, if they are present, must appear in the order shown. The first is the DEBUG packet which contains *debug* commands to control or select specific actions within the executive and database management systems. The second packet is the MAPOL packet containing the executive system control directives consisting of either a standalone MAPOL program or EDIT commands to modify the standard MAPOL program. If the MAPOL packet is absent, the unmodified standard MAPOL sequence directs the execution. The Solution Control commands appear in the third optional packet denoted by the keyword SOLUTION. These commands select the engineering data to be used in each subcase from the set of data provided in the Bulk Data packet. The fourth packet is the FUNCTION packet. It contains the definition of functions which allow the user to define new design constraints or an objective function. These functions may combine nodal and element response quantities for various boundary conditions and disciplines. The final

```
_______
_______          Any number of leading blank lines
_______
```

```
The Resource Section

   ASSIGN RUNDB = <name> <status> <PASSWORD=password> [params]

   MEMORY
```

```
   DEBUG
   _______       DEBUG directives
   _______       used for tracing input stream errors
```

```
   MAPOL [option-list]  or  EDIT [option-list]

   _______       MAPOL program or EDIT commands allow user
   _______       modifications to the standard ASTROS solution sequence
```

```
   SOLUTION

   _______       Solution Control Directives
   _______       select optimization and analysis disciplines
```

```
   FUNCTIONS [option_list]

   _______       Function definitions
   _______       Used to define objective or constraint functions
```

```
   BEGIN_BULK [option_list]

   _______       Bulk Data Entries
   _______       defines the structural and aero models, boundary
   …             conditions, loading cases, and other engineering data and the
   …             design model — design variables and constraints — required
   …             when performing design. Similar to NASTRAN
```

```
   [ENDDATA]     Bulk Data Terminator
```

Figure 3-1. Structure of the ASTROS Input Data Stream

```
ASSIGN RUNDB=TENBAR,NEW,REALLOC,PASSWORD=SHAZAM
DEBUG
DESIGN=5
EDIT NOLIST
INSERT 1463
CALL UTMPRT (,[AMAT]);
SOLUTION
TITLE = TEN BAR TRUSS
OPTIMIZE
      PRINT DCON=ALL, HIST
   BOUNDARY SPC = 1
      LABEL = STATIC ANALYSIS
      STATICS (MECH = 1), CONST (STRESS = 100, GENERAL = 100)
END
ANALYZE
   BOUNDARY SPC = 1, METHOD = 2
      STATICS ( MECH = 1 )
         LABEL = FINAL STATIC ANALYSIS
         PRINT DISP = ALL
      MODES
         LABEL = FINAL MODAL ANALYSIS
         PRINT (MODES=ALL) DISP = ALL, ROOT=ALL
END
BEGIN BULK
GRID,    1,    , 720.0,  360.0,   0.0
GRID,    2,    , 720.0,    0.0,   0.0
GRID,    3,    , 360.0,  360.0,   0.0
GRID,    4,    , 360.0,    0.0,   0.0
GRID,    5,    ,   0.0,  360.0,   0.0
GRID,    6,    ,   0.0,    0.0,   0.0
CROD,    1,  10,     3,      5
CROD,    2,  10,     1,      3
CROD,    3,  10,     4,      6
...
CROD,    9,  10,     2,      3
CROD,   10,  10,     1,      4
PROD,   10,   2,  15.0
MAT1,    2,  1.E+7,  ,  0.3,  0.1, , , ,  25000.0,  -25000.0
SPC1,    1,   123456,  5,   6
SPC1,    1,   3456,    1,   THRU,  4
FORCE,   1,   2,  ,  -1.E5,   0.0,    1.0,    0.0
FORCE,   1,   4,  ,  -1.E5,   0.0,    1.0,    0.0
CONVERT, MASS, 2.59E-3
EIGR,       2,    GIV,  0.0,  700.0,   2,   2,  ,  , ABC, +BC,  MAX
MPPARM, ISCAL, 1
DESELM,  1,  1,  CROD,  6.667E-3,  1000.0,   2.0,  ,  ROD1
DESELM,  2,  2,  CROD,  6.667E-3,  1000.0,   2.0,  ,  ROD2
...
DESELM,  9,  9,  CROD,  6.667E-3,  1000.0,   2.0,  ,  ROD9
DESELM, 10, 10,  CROD,  6.667E-3,  1000.0,   2.0,  ,  ROD10
DCONVMM,  100, 2.5+4, -2.5+4, , 2
DCONDSP,  100,  1,  UPPER, 2.0,  POSNOD1,  1,  2,   1.0
DCONDSP,  100,  2,  UPPER, 2.0,  POSNOD2,  2,  2,   1.0
...
DCONDSP,  100,  8,  LOWER, -2.0,  NEGNOD1,  4,  2,  1.0
ENDDATA
```

Figure 3-2. Features of a Sample ASTROS Input Stream

data packet is the BULK DATA packet. The BULK DATA packet contains the engineering data describing the finite element structural model, the aerodynamic model(s), and the design model, as well as all the data needed to perform the specific analysis and/or optimization tasks. The MAPOL, SOLUTION and BULK DATA packets are analogous to the NASTRAN executive control, case control and bulk data decks, respectively.

In interpreting the input data stream, ASTROS recognizes the keywords shown in Figure 3-1. These keywords must be the first nonblank characters on the line (leading blanks are allowed) and have the structure shown. In some cases the keyword is also a command line that makes up part of the data packet which it initiates. In these cases, the command parameters are documented in the User's Manual chapter discussing the details of the associated data packet. For example, the MAPOL keyword is part of a command that directs the MAPOL compiler to take certain actions. The detailed discussion of the MAPOL command is therefore contained in Chapters 2 and 7 of this manual. ASTROS automatically converts the case of the input data stream when necessary. The only portions of the data which are not converted are file names which are used in the **INCLUDE** command, the Resource Section, and the Solution Control commands **TITLE**, **SUBTITLE** and **LABEL**. This allows the user to freely enter data in any case. Be aware, however, that file names are never converted and that when using an ASTROS host computer in which case is important, such as Unix, then the correct case must always be used in file names.

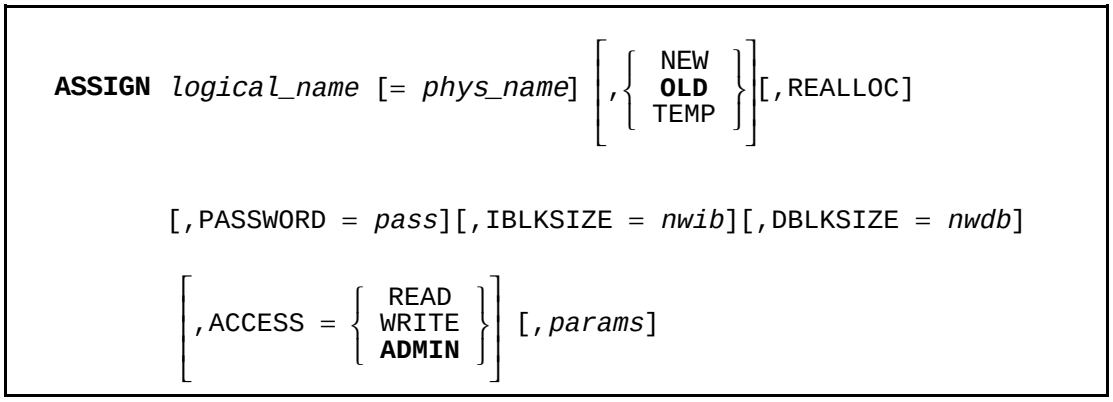
The section of the ASTROS input data steam that appears before the first packet header (e.g. **DEBUG**, **EDIT**, or **SOLUTION**) is called the Resource Section. This section may contain any number of **ASSIGN** commands and a **MEMORY** command. The **INCLUDE** command is used only as a convenience. These are discussed in detail in the following sections.

3.2.THE RESOURCE COMMANDS

As introduced earlier, there are two commands that may appear in the Resource Section. One or more **ASSIGN** commands and an optional **MEMORY** command. These are described in the following sections.

3.2.1.THE ASSIGN COMMAND

The **ASSIGN** command identifies the run-time database files to be used in the current ASTROS execution and specifies certain parameters associated with the files. The format of this command is:



where,

dbname is a name identifying the run time database files (maximum of 8 characters or fewer, depending on the local host.

password Passwords are used, but they are not required, only when **USE** is **RUNDB**, **ARCHIVE**, **SOF** or **NLDB**. For databases with a **STATUS** of **NEW**, the same password is used for the **READ**, **WRITE** and **ADMIN** privileges. The eSHELL command:

SET PASSWORD

may be used to change any or all of the passwords as desired. For **OLD** databases, the password must match the access type specified by the **ACCESS** parameter.

{ NEW / **OLD** / TEMP } Defines the status of the file. The file may be **NEW**, in which case it is allocated at run-time, an existing or **OLD** file, which is the default, or a **TEMP** file which is deleted at the end of the run.

REALLOC Requests that a ***new*** physical file be ***reallocated*** if it already exists. If you specify **NEW** for a file that already exists, and you do not include the **REALLOC** parameter, your job will be terminated.

params are optional (installation dependent) parameters e.g., **DBLKSIZE** = n, **IBLKSIZE** = n, etc.

logical_name Defines a logical ASTROS file name.

The entries on the **ASSIGN** commands are keyword controlled, but ***the options within the command must be entered in the order shown***. The keywords may be separated by commas or blanks. An example is:

```
ASSIGN RUNDB=ASTDB,OLD,PASSWORD=SECRET
```

Figure 3-3 illustrates the function of the **ASSIGN** command. In the case shown, the installation dependent parameters **DLOC** and **ILOC** have been used to select the physical devices on which the requested files for **DB1** reside.

The optional ***params*** on the **ASSIGN** command may or may not be keyword controlled and are installation dependent. They provide a mechanism for the user to direct machine or installation dependent file operations to be performed by the ASTROS procedure. At each site, the installation of the code involves a definition of these parameters and the form they must take. The ASTROS system is currently functional on numerous host systems including VAX/Ultrix, IBM/AIX, SGI 4D series and Crimson/Indigo series, HP/9000 series, CRAY/UNICOS, DECStation, Convex, and SunSparcstation. The availability on a specific computer may be obtained by contacting UAI.

The next section documents the installation-dependent **ASSIGN** parameters for some of the more common features/hosts. These features, however, may be customized to a very high degree and may be modified by the local system manager. Further documentation of the **ASSIGN** command is left to the local installation or will be included in the delivery material. The Programmer's Manual contains the detailed description of how these and other machine dependent parameters are defined.

3.2.2. ASSIGN COMMAND DESCRIPTIONS FOR HOST COMPUTERS

This section contains the descriptions of the machine and installation dependent parameters on the **ASSIGN** command for three machines on which ASTROS is currently functional. The parameters that are available at each site are listed and details of their use are presented. The user is cautioned that these are site dependent parameters which may be different for each installation even if the host system is the same. This documentation is provided both as an example to the system programmer and because the features that have been made available on these machines are very likely to exist on most machines that may be used. The user is referred to their ASTROS system manager for the particulars of the interface between the local host system and ASTROS. The following section describes parameters for computers using Unix-based Operating Systems

3.2.2.1. UNIX SYSTEM IMPLEMENTATION

The **ASSIGN** command supplies the ASTROS system with the root name of the database files, the status of those files, and a set of user parameters. The status is selected from **NEW**, **OLD** or **TEMP** and the set of user parameters can be any of the following keyword commands:

ILOC=*path*	Specifies the location of the Index Component file.
DLOC=*path*	Specifies the location of the Data Component files.
DBLKSIZE=*n*	CADDB data files block size in words.
IBLKSIZE=*n*	CADDB index file block size in words.

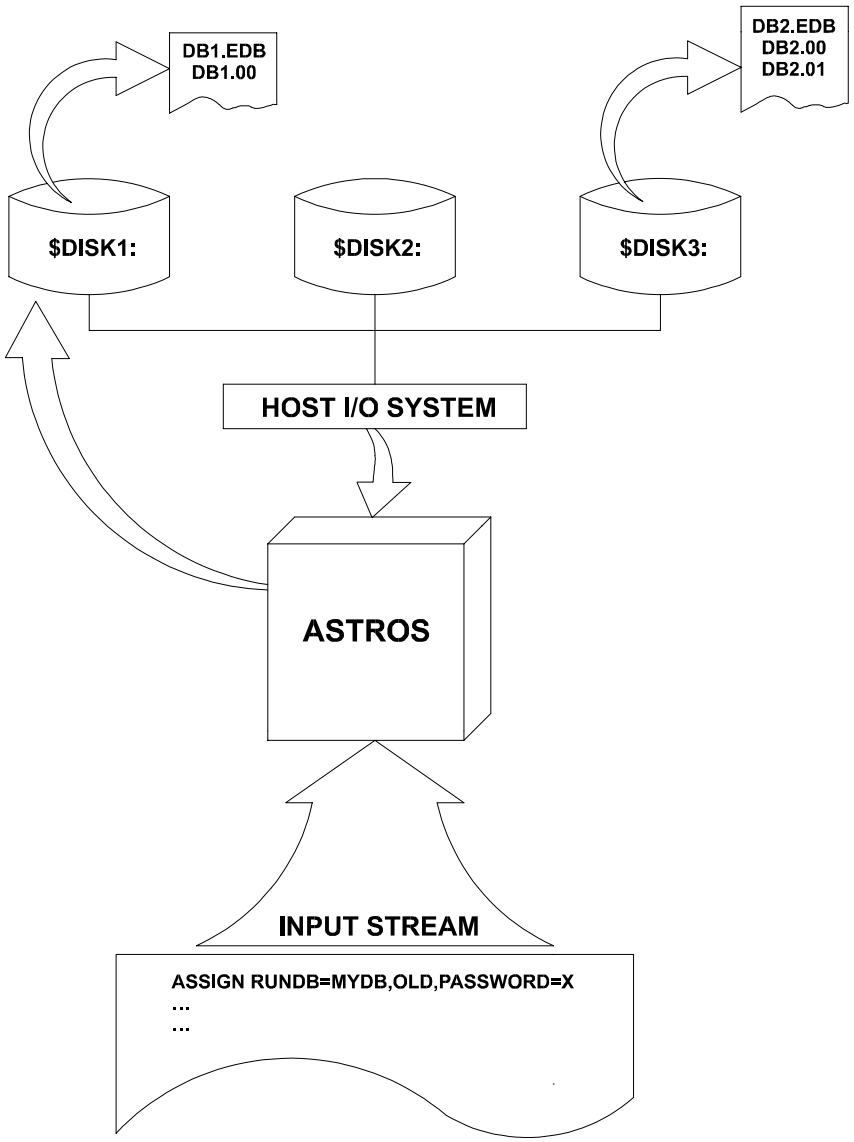


Figure 3-3. Function of the ASSIGN Command

Note that the **DLOC** parameter may specify a series of locations when very large databases are being created. The format is then:

```
DLOC=(path_1,path2,...)
```

TEMP *Database Example:*

When the status is **TEMP** a temporary database is created and no data is kept after the run. The other legal optional parameters are **DBLKSIZE** and **IBLKSIZE**.

```
ASSIGN RUNDB=MYDB,TEMP,PASSWORD=X,DBLKSIZE=2048,IBLKSIZE=256
```

NEW *Database Example:*

When the status is **NEW**, a new database is created. If the files already exist, they will be overwritten.

```
voldbroot.EDB - index file
voldbroot.00  - data file 1
```

Other legal parameters are **DLOC**, **ILOC**, **DBLKSIZE** and **IBLKSIZE**.

```
ASSIGN RUNDB=MYDB,NEW,PASSWORD=X,REALLOC,DLOC=/tmp/,ILOC=/tmp/
```

In this example, a database with files **/tmp/MYDB.EDB** and **/tmp/MYDB.00** are created. If existing files are found they will be overwritten.

OLD *Database Example:*

When the status is **OLD**, an old database is used. The physical files that make up the database must exist. Two, or more, files are used to store the database. The names of these files are as follows:

```
voldbroot.EDB - index file
voldbroot.00  - data file 1
```

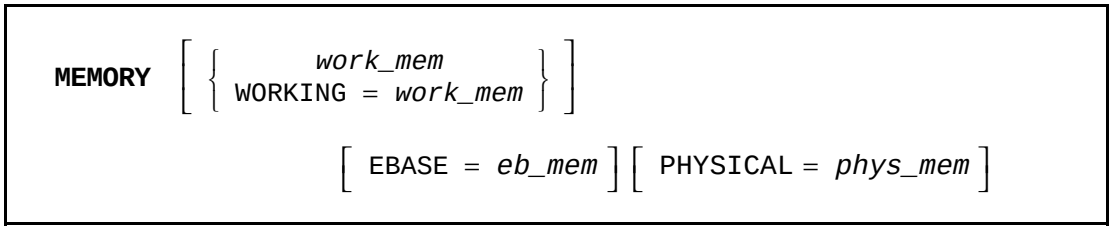
The **DLOC** and **ILOC** parameters, along with the root database name, are used to form the names. No other parameters are legal. Here is an example:

```
ASSIGN RUNDB=X,OLD,PASSWORD=X,DLOC=/home/dir1/,ILOC=/home/dir1/
```

In this example a database with files **/home/dir1/MYDB.EDB** and **/home/dir1/MYDB.00** are used.

3.2.3.THE MEMORY COMMAND

The MEMORY command specifies the amount of memory ASTROS will use for the internal storage of data. The format of this command is:



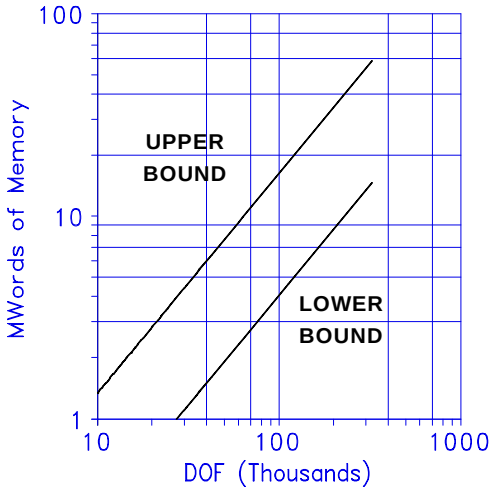
work_mem	Specifies the dynamic working memory size used by ASTROS modules. The working memory may be increased for large problems to reduce the amount of physical I/O. Note, however, that this may cause increased paging. Contact your ASTROS Support Specialist for additional details.
eb_mem	Specifies the eBASE database memory size. This memory is a separate memory pool used by the database during execution. Normally the default value in the Preference File is sufficient, but if you use block sizes larger than the default for any database, this value may need to be increased.
phys_mem	Specifies the real physical memory memory size. The physical memory is used to control certain advanced algorithms. Contact your ASTROS Support Specialist for additional details.

The units of the memory size are determined by the two optional command arguments.

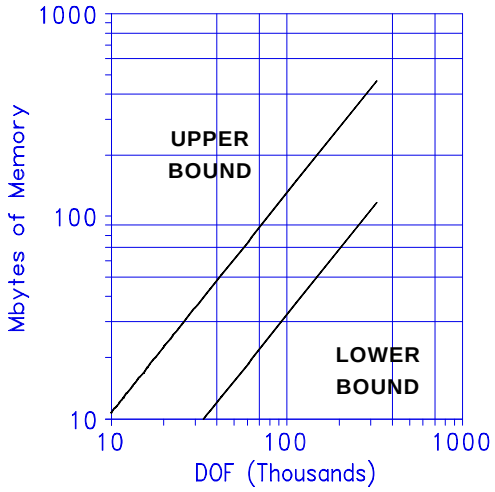
The first argument indicates an order of magnitude for *memory_space*, **M** for millions, **K** for thousands. The second argument indicates the unit specifier as single precision words (**W**), bytes (**B**), or computer precision words (**P**). If neither is present, then *memory_space* is taken to be ***single precision computer words***. The working memory for ASTROS is dynamically acquired during execution. The amount of memory used is determined, in order of precedence, by the **MEMORY** Resource command, the **- m** option of the **astros** script, and the configuration parameter *working_memory*.

You may provide default values for this command in the **<Computing Resources>** group of the **[ASTROS]** Section of the Preference File.

ASTROS has two high-performance solvers which take advantage of the latest developments in sparse matrix algorithm technology. The first of these is the symmetric matrix decomposition used in static analyses, and the second is the Lanczos eigenextraction method. This latter method is used for extracting a modest number of eigenvalues from very large systems. When these solvers are used, memory requirements may become significant. The figures below give upper and lower bound estimates for the amount of



Cray



Others

memory that you should specify on your **MEMORY** Command. Although the eigensolver takes slightly more memory, about 20%, the same figures may be used to approximate the requirements for either solver. Note, that in the case of the Linear Solver, if you do not specify enough memory for the new algorithm, the program will revert to the old solution algorithm. This is not the case for Lanczos — the job will terminate. These curves have been created using a representative sample of real analysis jobs. They are intended only to be used as guidelines — a specific job may take significantly more or less memory than indicated.

For example, to execute ASTROS using 12 million words of working memory, any of the following commands may be used:

```
MEMORY = 12000000; or
MEMORY = 12000KW; or
MEMORY = 12MW
```

3.3. THE INCLUDE DIRECTIVE

The input data stream typically resides in a single file, but the user can direct the input stream interpreter to include other files through the use of the **INCLUDE** directive in the primary input stream. The format of the **INCLUDE** command is:

```
INCLUDE <filename>
```

where,

filename is a name identifying the file to be included (maximum of 72 characters).

The filename, which is used in a FORTRAN **OPEN** statement, must satisfy the requirements of the particular host system for file names. Beyond this restriction, the user is free to have any set of contigu-

ous non-blank characters in the filename. In order to avoid the possibility of an infinite recursion, there is a restriction on the include feature that no **INCLUDE** statement can appear in a file that is being included. For example, if the file "TENBAR" is being included, it may not itself contain an **INCLUDE** directive. The input stream interpreter will terminate with an appropriate error message should this occur.

The **INCLUDE** directive can appear anywhere in the input stream after the **ASSIGN** command. The **ASSIGN** must always appear as the first non-blank line in order to allow the use of the run time database in the subsequent input stream interpretation. A single data packet can be split among included files or an **INCLUDE** file may contain parts of multiple data packets. The input interpreter merely replaces the **INCLUDE** directive with the data contained in the named file so the only requirement is that the input stream that results from the combination of all **INCLUDE**s have the form of a normal input stream. The **INCLUDE** feature can be very useful in certain circumstances. For example, a special user developed MAPOL sequence can be stored and maintained external to the files containing the engineering data for particular runs; or, conversely, the bulk data representing a large model can be included into the file containing the solution control directives.

3.4. THE DEBUG PACKET

The debug packet represents a development tool and is intended to be used primarily by those responsible for maintaining the software. The debug packet provides the system programmer with the means to invoke or control certain executive and database management system functions that are helpful in tracking the ASTROS execution and/or testing the executive and database management system software. However, because some of the debug options can be useful to the general user, the debug packet is fully documented in the User's Manual rather than in the Programmer's Manual. This section documents each of the debug options and indicates how the option can be useful in debugging the ASTROS procedure. Emphasis is placed on those debug options that are of interest to the general user.

The debug packet is initiated by the keyword **DEBUG**, which must appear alone on the line of the input stream that follows the last command in the Resource Section, and that precedes any other data packet. Following the initiator, any number of debug lines can be included in the data stream. Each debug command line can be composed of a number of debug commands, appearing in any order, separated by blanks or commas. The DEBUG packet is terminated when a new data packet initiator, or the end of the input stream, is encountered. Most debug commands consist of single keywords which toggle flags activating the debug functions. The appearance of these debug keywords is all that is required to activate the option. Other debug commands select that a flag take on a particular value. These commands have the form:

```
<command> = <value>
```

There can be any number of blanks between the end of the command keyword, the value and the equal sign, but neither the command nor the value can contain imbedded blanks. Any errors in the DEBUG packet input will result in warnings but will not terminate the execution and the erroneous command will be ignored.

The tables shown in this section indicate the list of keywords that can be included in the DEBUG packet. The debug commands are grouped into executive system and database management system debugs. Each of these groups is described in greater detail in the following sections.

3.4.1. EXECUTIVE SYSTEM DEBUG COMMANDS

The first four executive system keywords are intended to assist the system programmer in following the actions of the MAPOL compiler and execution monitor. The options are shown in Table 3-1.

As such, they are of limited value to the general user. The **MATRIX** option, however, can be useful in tracking the execution of the MAPOL program. It echoes the matrix utility calls for all matrix operations that are in the MAPOL sequence. For example, if the MAPOL program includes the expression:

```
[A] := TRANS( [B] ) * [C] + [D];
```

the **MATRIX** trace echoes the resultant call to the **MPYAD** large matrix utility with the arguments shown in detail. This trace can be very useful in determining which particular MAPOL instruction is being executed when a problem occurs. Large MAPOL programs with many loops and a large number of matrix expressions can be debugged quite simply using the **MATRIX** trace. All MAPOL statements that result in calls to any of the large matrix utilities, such as **PARTN**, **MERGE**, **MPYAD**, and **MXADD** are echoed.

LOGBEGIN and **LOGMODULE** provide expanded echoes of the module timing summary that is found at the end of each ASTROS output file. When problems cause early termination of the job, these options provide the name of the last module entered prior to the failure. This provides a starting point to diagnose the problem.

Table 3-1. Executive (MAPOL) Debug Commands

KEYWORD	DESCRIPTION
MSTACK	MAPOL compiler stack output
MEXEC	MAPOL execution debug flag
MOBJ	MAPOL object code debug packet
MTRACE	MAPOL trace debug output
MATRIX	MAPOL peeper matrix operation trace
LOGMODULE	Expanded log entries for each module
LOGBEGIN	Beginning entries for each module in log file

3.4.2. DATABASE AND MEMORY MANAGER DEBUG COMMANDS

The database management system has a number of debug options which can be divided into three categories: trace options, control options and memory manager options. These are shown in Table 3-2.

The first group of database debug commands contain two tracing options: **TRACE** and **IOSTAT**. The **IOSTAT** keyword selects either a **FULL** tracing or a **SUM**mary. The first of these options and the **IOSTAT=FULL** option are further controlled by the **ENTITY** option which completes the first group of keywords. Note: the tracing keywords generate an overwhelming amount of data which are often of limited use unless the user is familiar with the internal structure of the database files. The **ENTITY** keyword limits the activation of the tracing options to those times when the named database matrix, relation or unstructured entity is open. If no entity specification is made, the traces are active for all database operations. In addition to their role in debugging the database software, the trace options provide a useful means of debugging the interface between a user written module and the database.

The database control options **CALLSTAT** and **NOCOREDIR** provide user control over two internal database functions. The **CALLSTAT** option compiles a summary of the number of calls made to each database subroutine. This summary, in combination with the **IOSTAT** option, provides statistics on the number of database operations in the execution. The **NOCOREDIR** option is made available for machines with limited core memory resources. If **NOCOREDIR** is selected, the database manager stores the database directories on the database files rather than in core. This can substantially reduce the database memory requirements at the cost of increasing the number of input/output operations.

Table 3-2. Database Debug Commands

KEYWORD	DESCRIPTION	
TRACE	Traces all database CALLs	
IOSTAT=parm	Database I/O tracing	
	FULL	Full trace of I/O activity
	SUM	Summary of I/O activity
ENTITY=name	Restricts tracing to entity **name**	
CALLSTAT	Compiles statistics on the number of calls to each database routine at the end of a job.	
NOCOREDIR	Turns off the option to store directories in core	
NODELAYCRE	Turns off the option that delays entity creation until the entity is opened	
MEMORY	Memory manager debug print	

The last group of database debug options consists of the **MEMORY** command. This option causes an echo of all the memory management calls made in the modules. The user can then track the ASTROS execution into the engineering modules themselves. In addition to the echo, the **MEMORY** option invokes a *checksum* operation which checks for the integrity of the memory block headers on every memory manager operation. If the checksum fails, a message is written to the effect that a block header has been overwritten. This option is very effective in uncovering errors in engineering modules that make use of dynamic memory allocation.

3.4.3. INTERMEDIATE RESULTS PRINTING COMMANDS

Many of the ASTROS engineering modules have intermediate output print options that are useful in tracing the details of an analysis or in reviewing the quality of the inputs. These many options are listed in Table 3-3.

Table 3-3. Intermediate Results Debug Commands

KEYWORD	DESCRIPTION	
AMP=n	Intermediate unsteady Aero matrices	
	1	Prints the SKJ matrix and, if only one group, includes AJJ, QKJ and QJJ if they exist
	>1	Includes D1JK, D2JK and AJJT matrices
DESIGN=n	**1**	Prints initial design information
	2	Includes function values at each iteration
	3	Includes internal Microdot parameters
	4	Includes search directions
	5	Includes gradient information
	6	Includes scaling information
	7	Includes one-dimensional search information
FLUTTRAN=n	Additional flutter eigenextraction information	
	1	Prints the number of iterations required to find each flutter root
	>1	Includes the estimated roots for each iteration
MKUSET=n	MKUSET redundant set warnings	
	>0	Prints warning messages if the same degree of freedom is placed in a set more than once.
SAROGEOM=n	Planar steady aerodynamics geometry option	
	>0	ASTROS execution stops in module STEADY after the steady aerodynamic geometry has been computed. No printed output is generated unless the **STEADY** debug is also used.
SCEVAL=n	Prints additional constraint data	
	>0	Prints the stress or strain components, the constraint type and the constraint value for each constrained element/layer.
SHPGEN=opt	Prints or punches **SHAPE** or **SHAPEM** Bulk Data entries automatically created by the **SHPGEN** capability.	
	PRINT to print the generated Bulk Data entries	
	PUNCH to punch the generated Bulk Data entries	
	BOTH to print and punch the generated Bulk Data entries	
STEADY=n	Steady preface USSAERO output	
	1	Prints steady aerodynamic model geometry
	2	Includes stability coefficient data
	3	Includes pressure data
	4	Includes velocity components and matrix output

3.4.4. MISCELLANEOUS DEBUG COMMANDS

Table 3-4 shows several miscellaneous DEBUG commands which are used to control optimization and looping, optimization scaling, and geometry checking of plate elements.

Table 3-4. Miscellaneous Debug Commands

KEYWORD	DESCRIPTION	
DVWARNING=opt	Sets a limit for the number of design override linking warnings (e.g. Torsion Set to Zero for Design) issued by any one element type. Use keyword **ALL** or integer value **n**. Default=50.	
MPSCAL=opt	Controls scaling of design variables	
	ON	Scales global variables to unity before Microdot is invoked (Default).
	OFF	Does not scale variables.
PBKDIAG=n	Enables Panel Buckling diagnostics	
	0	Turn off diagnostic print.
	1	Print roots only.
	2	Print roots and eigenmatrix.
PBKNINIT=n	**n**	Initial number of terms in the Panel Buckling series solution, must be less than **PBKNMAX**.
PBKNMAX=n	**n**	Maximum number of terms in the Panel Buckling series solution.
PBKPWER=val	**val**	Power used in linearizing the Panel Buckling constraint $G = F\left(\frac{\lambda_{req}}{\lambda}\right)^{\frac{1}{val}}$ Default = 3.0.
STRESS_DV=opt	**opt**	Determines whether the check for stress constraints on undesigned elements will be a warning or fatal error. Value is **WARNING** (Default) or **FATAL**.
WARPMX=val	**val**	The maximum allowed warping value for QUAD4 elements.
ZERODOBJ=val	**val**	Specifies a tolerance value for defining the objective function to be the same from one iteration to the next.
ZEROITER=n	**n**	Specifies the maximum number of design iterations that may have the same objective function value before ASTROS is terminated.

3.4.5. SEQUENCER INTERMEDIATE PRINT COMMANDS

There are a number of print and control options for the grid point sequencer that are shown in Table 3-5.

Table 3-5. Sequencer Debug Commands

KEYWORD	DESCRIPTION	
SEQPRINT=opt	Selects sequencing intermediate print	
	DETAIL	Requests detailed print of sequencing
	DIAGNOSTIC	Requests diagnostic print
SEQMETH=meth	Select sequencer method	
	CM	Selects the Cuthill-McKee Method
	GPS	Selects the Gibbs-Poole-Stockmayer Method
	ALL	Selects the best of both 1 and 2
NOSEQMPC	Requests that MPCs not be processed during sequencing	
SEQCRIT=crit	Selects sequencing method	
	BAND	Selects minimum bandwidth criteria
	PROF	Selects profile criteria
	RMS	Selects RMS criteria
	WAVE	elects minimum wavefront criteria
SEQPUNCH	Requests punching of **SEQGP** bulk data entries	
SEQOFF	Deselects sequencing	

Chapter 4.

THE EXECUTIVE SYSTEM AND MAPOL

The ASTROS system is controlled by an ***Executive System***. One of the functions of the ASTROS executive system, described in detail in Reference 1, is to determine the sequence in which the modules of the program are invoked. For ASTROS, the Matrix Analysis Problem Oriented Language (MAPOL) has been developed to perform this executive system task. The MAPOL language has its conceptual roots in the Direct Matrix Abstraction Program (DMAP) capability developed for the NASTRAN structural analysis system (Reference 2). MAPOL provides the same advantages to the ASTROS system and represents a considerable advance over DMAP in that MAPOL is a structured, procedural language that directly supports high order matrix operations, manipulation of database entities and complex data types. Moreover, the syntax of the language looks much like that of any scientific programming language and so is easily learned by anyone who knows FORTRAN or PASCAL.

From the user's point of view, ASTROS is directed by a sequence of control statements "coded" in the MAPOL language just as a NASTRAN rigid format is coded in the DMAP "language."

☞ **The majority of users will use the standard MAPOL sequence. This is the default, and as such it requires no special action. Advanced users may optionally edit the standard sequence or write their own "program". The methods used to do this are described in this Chapter. Because changes to the executive system are an advanced topic, first-time users may proceed directly to Chapter 4.**

The executive system within ASTROS compiles the MAPOL program and executes the resultant "ASTROS machine code" which directs the execution of the ASTROS procedure. (Note that ASTROS is NOT written in MAPOL, only the executive control algorithm is written in the MAPOL language. In fact, ANSI standard FORTRAN was used to write the compiler for MAPOL and for all the engineering software of the ASTROS system.) MAPOL allows you to manipulate the software system in many ways to tailor the available capabilities to perform particular tasks. At a higher level of sophistication, you may add modules to the system or replace modules that already exist. Obviously, some of these features

require a knowledge of the ASTROS system that is beyond the scope of the User's Manual. Those features that require detailed information are more fully discussed in the Programmer's Manual, but their existence is emphasized here in order to introduce you to the flexibility that the executive system provides.

This Chapter presents the mechanics of the MAPOL packet. The potential of the executive system to tailor the ASTROS software is explored in this discussion of the standard sequence. In addition to this Chapter, Chapter 8 presents a detailed description of the MAPOL language, its syntax and features. It cannot be overemphasized that, while the capabilities implemented in the ASTROS software are significant, the true power embodied in the ASTROS system is its immense flexibility, largely provided by the executive system and its MAPOL language.

The MAPOL packet is initiated either by the keyword MAPOL or by the keyword EDIT and is terminated upon encountering the SOLUTION CONTROL packet, the BULK DATA packet or the end of the input stream. In addition, each of the initiator keyword commands act as directives to the MAPOL compiler to take specific actions. The MAPOL and EDIT commands are:

```
MAPOL [ GO   ] [ LIST   ]
      [ NOGO ] [ NOLIST ]

EDIT  [ GO   ] [ LIST   ]
      [ NOGO ] [ NOLIST ]
```

where:

[**GO** / NOGO]	selects whether the MAPOL program is to be executed after compilation.
[**LIST** / NOLIST]	selects whether the MAPOL source code is to be written to the output file.

The MAPOL command is followed by a MAPOL program which can be any syntactically complete set of MAPOL statements as described in the chapter on MAPOL Programming (Chapter 9). The EDIT command indicates that the MAPOL packet will consist of edit commands that INSERT, DELETE or REPLACE lines of the standard executive sequence.

4.1. THE MAPOL PROGRAM

If the MAPOL packet begins with the MAPOL command line, the compiler assumes that the remaining statements in the packet constitute a complete MAPOL program. That program can be any set of MAPOL statements that satisfy the rules of the language as presented in Chapter 8. The program can call any of a number of intrinsic functions (including most of the common FORTRAN intrinsic functions) and any of the "engineering" utilities and modules that are available in ASTROS. You can access these modules in any desired order, subject only to limits imposed by the engineering modules themselves. In addition, you can write special purpose modules and define them to the compiler through the SYSTEM GENERATION (SYSGEN) program discussed in the Programmer's Manual. Thus, a wide range of tasks can be performed using the ASTROS system in combination with a MAPOL program.

The MAPOL language can be read and written easily by anyone familiar with a scientific programming language. This feature opens the advantages of the executive system to the average user without requiring specialized knowledge in computer science or requiring effort to learn a radically different programming language. You will often find the simplicity and power of the MAPOL language enables many tasks to be performed using the ASTROS system that are not explicitly supported in the standard executive sequence.

4.2. MAPOL EDIT COMMANDS

If the MAPOL packet begins with the EDIT command line, the compiler assumes that the remainder of the packet (if any) is composed of MAPOL edit commands and new MAPOL statements that modify the standard executive sequence. The set of edit commands is given in Table 4-1. They allow you to insert, delete and replace lines of the standard MAPOL sequence. All of the edit commands reference a line number or range of line numbers. The line numbers are those in a compiled listing of the standard MAPOL sequence which is written as part of the system generation task. When editing the standard sequence, you are cautioned to obtain the most recent listing either from the SYSGEN output or by executing ASTROS with an input stream containing only an **ASSIGN** command and the one line MAPOL packet:

```
EDIT LIST NOGO
```

This input stream will result in an output file containing the current listing of the standard executive sequence.

4.3. THE STANDARD EXECUTIVE SEQUENCE

As previously mentioned, the MAPOL language has its conceptual roots in the DMAP "language". In order to allow the user of NASTRAN to perform certain predefined analyses, a set of "rigid formats" or DMAP algorithms were written, alleviating the user of the need to learn the details of the control language. Each rigid format allowed the user to perform analyses in a different engineering discipline; for example, static structural analyses, normal modes analyses, or transient analyses. In a similar manner, a standard executive sequence or MAPOL algorithm is available in the ASTROS system which supports all

Table 4-1. MAPOL Edit Commands

STATEMENT	FUNCTION
EDIT	Modify the standard solution
DELETE a[,b]	Remove lines **a** through **b** inclusive
REPLACE a[,b]	Removes lines **a** through **b** inclusive and replaces them with the following lines
INSERT a	Insert the lines following the command after line **a**

the engineering disciplines and optimization features of the procedure. Unlike the multiple DMAP rigid formats, however, there is a single MAPOL sequence that supports all the available engineering disciplines as well as optimization. This fundamental difference is necessary to permit multidisciplinary optimization.

One consequence of having a single multidisciplinary algorithm is that the standard sequence appears to be very complicated. The purpose of this section is to present the internal structure and flow of the standard MAPOL sequence thereby providing the user with sufficient information to tailor the standard sequence to suit individual needs. The discussion in this section will be general in order to provide the necessary overview and to introduce the concepts embodied in the standard sequence. Modifications to the standard sequence will be presented primarily in terms of capabilities but the presentation will be supported by examples that represent both simple and more complex modifications. Finally, the Chapter closes with a detailed line-by-line presentation of the standard executive sequence. The reader is also referred to the Programmer's Manual for information on the addition of modules to the ASTROS engineering library.

4.4. STANDARD EXECUTIVE SEQUENCE STRUCTURE

The standard MAPOL sequence consists of two major components: the variable declarations and the solution algorithm. The solution algorithm can be further divided into ***preface modules***, the ***optimization segment*** and the ***final analysis segment***. The declaration segment declares all variables used in the MAPOL sequence. This includes all integer and real scalar variables as well as high order variables: relations, matrices and unstructured database entities. Within the solution algorithm, the preface modules comprise a group of engineering modules exercised prior to the boundary condition loops to perform a number of system initialization tasks; e.g. loads generation and the computation of invariant aerodynamic matrices. The separate optimization and analysis segments consist of a loop on the number of (optimization or analysis) boundary conditions in the current execution. In the optimization segment, a second boundary condition loop is performed to obtain the sensitivities of active boundary condition dependent constraints in preparation for the optimization task.

Figure 4-1 provides the standard algorithm structure showing how multidisciplinary optimization is performed in ASTROS. It is readily apparent that the structure of the standard MAPOL sequence has been determined by the requirement to perform multidisciplinary optimization. Each of the segments of the standard sequence are discussed in greater detail in the following sections.

PREFACE SEGMENT		Initialization (PREFACE) Segment
		WHILE NOT CONVERGED DO

OPTIMIZATION SEGMENT	ANALYSIS PHASE	**For Each Boundary Condition Do**
		Discipline 1
		Subcase 1 Constraints
		Subcase 2 Constraints
		…
		Discipline 2
		…
		End Do

	SENSITIVITY PHASE	Select Active Constraints
		For Each Active Boundary Condition Do
		Active Discipline 1
		Active Subcase 1 Constraint Sensitivities
		Active Subcase 1 Constraint Sensitivities
		…
		Active Discipline 2
		…
		End Do

	OPTIMIZATION PHASE	Redesign Based on Current Active Constraints and Constraint Sensitivities
		END DO

FINAL ANALYSIS SEGMENT		**For Each Boundary Condition Do**
		Discipline 1
		Subcase 1
		Subcase 2
		…
		Discipline 2
		…
		End Do

Figure 4-1. Structure of the Standard MAPOL Sequence

4.4.1. MAPOL Declarations

MAPOL is a strongly typed language that requires all variables used in a program unit (either the main program or a procedure) to be declared. This applies to both simple variables like real and integer scalar or array variables and to high order variables (like MATRIX) that refer to database entities. The first several hundred lines of the standard sequence consist solely of these variable declarations. Tables 4-2 through 4-7 give a summary of the scalar parameters used in the standard MAPOL sequence. These parameters, initialized in engineering modules or in the MAPOL sequence, are used as either logic control flags or arguments to the engineering modules. The tables, which are catagorized by function, provide a brief description of each variable and a list of modules (where applicable) that use the parameter. For a description of all the variables used as arguments of the engineering modules, refer to the ASTROS Programmer's Manual. It should be noted that all of these variables can be directly modified within the MAPOL algorithm at your discretion. A discussion of those parameters that you are most likely to want to modify is given in Section 4.4.3, but the experienced user is free to change any variable in the MAPOL sequence.

Higher order variables fall into two categories: MAPOL entities and ***hidden*** entities. MAPOL entities are those that actually appear in the MAPOL sequence while hidden entities are those that are declared but do not subsequently appear in the sequence. Their declaration ensures that the corresponding database entity is created and can be used by a number of engineering modules without requiring the entity name to appear in the argument list. Hidden entities are typically those that contain the raw data needed by many modules; e.g. bulk data, geometry data and connectivity data. The declarations of the higher order variables are arranged to place logically related entities together. Several of the matrix entities, it should be noted, are subscripted, for example **`[KLLINV(1000)]`**. The subscripted matrix entity allows the ASTROS software to perform multiple analyses in several boundary conditions and retain the information needed to compute the sensitivities of the active constraints retained from each of these boundary conditions. The ASTROS executive system generates a name for each subscripted variable, and that name is used by all the engineering modules receiving the subscripted entity name as an argument. The actual database entity name need not be known. This does, however, impose the following restriction: a subscripted entity may not be used as a hidden entity in any engineering module; it must appear in the calling list for the module because only the executive system knows the actual name of the database entity corresponding to the current subscript value. In the standard sequence, provision has been made for up to 1000 entities (doubly subscripted arrays of entities are set up for 30 boundary conditions and 33 secondary subscript values), but you can change the declared number of subscripts to match the required range of indices.

Table 4-2. Real Parameters in the Standard Sequence

PARAMETER NAME	USED IN MODULES	DESCRIPTION	DEFAULT
ALPHA	SOLUTION FSD	Exponential move limit for fully stressed design. Set through the Solution Control **OPTIMIZE** command.	0.90
CNVRGLIM	DESIGN FSD	Convergence test limit specifying the maximum percent objective change for the appropriate problem to be considered converged. Output from **SOLUTION.**	0.50
CTL	ACTCON DESIGN FSD	Criteria for defining a constraint to be active in determining convergence in **ACTCON**. If value > **CTL**, the constraint is active. Set in **DESIGN** or **FSD**.	
CTLMIN	ACTCON DESIGN FSD	Criteria for denoting a constraint to be feasible in determining convergence in **ACTCON**. If maximum constraint value <**CTLMIN**, the design is feasible. Set in **DESIGN**, or **FSD**.	
EPS	SOLUTION ACTCON	Criteria used in **ACTCON** for selecting active constraints. All constraints with values greater than **EPS** will be retained. Set through the Solution Control **OPTIMIZE** command. (See also **NRFAC**)	-0.10
FDSTEP	NLEMG EBKLEVAL MAKDFV MAKDFU	Finite difference step size for sensitivity calculations.	0.001
FMAX	GRD1 GRD2	The maximum frequency value associated with the **NEIV** eigenvalues computed for dynamic reduction in the current boundary condition. Output from **GDR1.**	
K6ROT	EMG	Stiffness value for plate element drilling degrees of freedom.	0.0
MACH	SAERODRV	Mach number for the current case. Set in **SAERODRV.**	
MOVLIM	SOLUTION DESIGN MAKDFV TCEVAL	A move limit applied to the physical design variable (**V**) for mathematical programming methods. The move is: **V/MOVLIM < V < V * MOVLIM.** Set through the Solution Control **OPTIMIZE** command.	2.0 (>1.0)
NRFAC	SOLUTION ACTCON	Criteria used in **ACTCON** for selecting active constraints. At least **NRFAC** times **NDV** constraints will be retained. Set through the Solution Control **OPTIMIZE** command. (See also **EPS**)	3.0
QDP	SAERODRV SAERO others	Dynamic pressure value used in the current steady aeroelastic subcase. Output from **SAERODRV** used subsequently in MAPOL expressions and modules.	
WINDOW	TCEVAL	The window in which the **MOVLIM** bound is overridden to allow local variables to change sign. If **WINDOW** is 0.0, then the local variable may not change sign. If it is nonzero, the half-width of a band around zero, called **EPS**, is computed by: **EPS = WINDOW/100 * MAX(ABS(TMIN),ABS(TMAX))** If the local variable falls within the band, then the new minimum or maximum for the current iteration is changed to lie on the other side of zero from the local variable. Output from **SOLUTION**.	0.0 (≥0.0)

Table 4-3. Integer Modelling Parameters

NAME	MODULES	DESCRIPTION
ASIZE	**GDR3**	The number of a-set degrees of freedom.
BC	**N/A**	Boundary condition loop counter.
BCID	**many**	Boundary condition identification number.
DDFLG	**DDLOAD**	Indicates if the current statics subcases contain design dependent (gravity or thermal) loads. Output by **DDLOAD**.
ESIZE	**BCBGPDT** **others**	The number of extra points in the boundary condition.
GNORM	**GDR3**	The sum of **LJSET** and **LKSET**.
GSIZEB	**IFP** **others**	The number of structural degrees of freedom in the model. Output from **IFP** and subsequently used in many modules.
GSIZE	**GDR4** **others**	**GSIZEB** modified subject to dynamic reduction.
HSIZE	**FLUTTRAN** **OFPEDR** **REIG** **others**	Number of eigenvectors extracted by the **REIG** module. Set in **REIG**.
LJSET	**GDRi**	Number of degrees of freedom in the j-set in dynamic reduction. Set in **GDR1**.
LKSET	**GDRi**	Number of degrees of freedom in the k-set in dynamic reduction. Set in **GDR1**.
LSIZE	**GDR1**	The number of l-set degrees of freedom.
NEIV	**GDR1** **GDR2**	An output from **GDR1** indicating the number of eigenvalues below the maximum frequency specified for dynamic reduction.
NGDR	**BOUND**	Logical flag equal to negative one if dynamic reduction is selected for the current boundary condition.
NMPC	**BOUND** **ABOUND**	Logical flag equal to the number of degrees of freedom in the multipoint constraint set for the current boundary condition.
NOMIT	**BOUND** **ABOUND**	Logical flag equal to the number of omitted degrees of freedom in the current boundary condition.
NRSET	**BOUND** **ABOUND**	Logical flag equal to the number of support degrees of freedom in the current boundary condition.
NSPC	**BOUND** **ABOUND**	Logical flag equal to the number of single point constraint degrees of freedom in the current boundary condition.
PSIZE	**BCBULK** **others**	The number of grid and extra point degrees of freedom.
SINGASET **SINGLSET** **SINGOSET**	**SDCOMP**	Used to process singular matrix columns.

Table 4-4. Integer Design Parameters

NAME	MODULES	DESCRIPTION
FSDE	SOLUTION	The last iteration to use fully stressed design. Output from **SOLUTION**.
FSDS	SOLUTION FSD	The first iteration to use fully stressed design. Output from **SOLUTION**.
MAXITER	SOLUTION ACTCON	Parameter set in the MAPOL sequence indicating the maximum number of resizing cycles that are to be performed. Set through the Solution Control **OPTIMIZE** command. (Def = 15)
MPE	SOLUTION	The last iteration to use mathematical programming. Output from **SOLUTION**.
MPS	SOLUTION FSD	The first iteration to use mathematical programming. Output from **SOLUTION**.
NACSD	ABOUND	The number of active stress and displacement constraints in the current active boundary condition. Used to select either the virtual load or gradient method in sensitivity analysis. Set in **ABOUND**.
NAUS	ABOUND	The number of active displacement vectors for statics. Set in **ABOUND**.
NBNDCOND	SOLUTION	The total number of boundary conditions in the solution control packet. Equal to the number of optimization boundary conditions plus the number of analysis boundary conditions. Output from **SOLUTION**.
NDV	MAKEST, others	The number of global design variables in the design model. Set by **MAKEST** and used in many subsequent modules.
NITER	N/A	The current optimization iteration number.
NUMOPTBC	SOLUTION	The number of optimization boundary conditions in the solution control packer. Set in **SOLUTION**.

Table 4-5. Integer Aerodynamic Parameters

NAME	MODULES	DESCRIPTION
MINDEX	ABOUND, AEROSENS, BOUND, PFAERO	The index value for the Mach number dependent subscripted steady aerodynamic matrices. Typically has a value used to select the proper matrices for the current boundary condition.
SUB S	SAERODRV SAERO others	Identifies the subcase subscript. **SAERO** subcases with the same symmetry Mach number, **MINDEX**, trim type, and dynamic pressure are processed using the same subscript. This occurs with multiple load conditions with the same aero correction.
SYM	BOUND	A control flag denoting whether the symmetric (**SYM=1**) or antisymmetric (**SYM=-1**) steady aeroelastic matrices are to be used are to be used in the current boundary condition.

Table 4-6. Integer Discipline Parameters

NAME	MODULES	DESCRIPTION
BDFR	**BOUND**	Indicates if there are any direct **FREQUENCY** response subcases in the current boundary condition.
BDRSP	**BOUND**	Indicates if there are either **TRANSIENT** or **FREQUENCY** response disciplines in the current boundary condition.
BDTR	**BOUND**	Indicates if there are any direct **TRANSIENT** response subcases in the current boundary condition.
BDYN	**BOUND**	Indicates if there are any dynamic analyses (**FLUTTER**, **TRANSIENT** or **FREQUENCY**) in the current boundary condition.
BFLUTR	**BOUND**	Indicates if there are any **FLUTTER** analyses in the current boundary condition.
BGUST	**BOUND**	Indicates if there are any gust loads for either **TRANSIENT** or **FREQUENCY** disciplines in the current boundary condition.
BLOAD	**BOUND**	Indicates if there are any mechanical, thermal or gravity static applied loads in the current boundary condition.
BMASS	**BOUND**	Indicates if a mass matrix exists in the current boundary condition.
BMFR	**BOUND**	Indicates if there are any modal **FREQUENCY** response subcases in the current boundary condition.
BMODES	**BOUND**	Indicates if there are any disciplines that require that a normal **MODES** analysis be performed.
BMTR	**BOUND**	Indicates if there are any modal **TRANSIENT** response subcases in the current boundary condition.
BSAERO	**BOUND**	Indicates if there are any **SAERO** subcases in the current boundary condition.
DMODES	**BOUND**	Indicates if there are any modal disciplines in the current boundary condition.
NGDR	**BOUND**	Indicates if dynamic reduction is selected for the current boundary condition.

Table 4-7. Logical Discipline Parameters

NAME	MODULES	DESCRIPTION
ACTAEFF	ABOUND	**TRUE** if the current boundary condition has any active aeroelastic effectiveness constraints.
ACTAERO	ABOUND	**TRUE** if the current boundary condition has any active constraints associated with **SAERO** analyses.
ACTBAR	ABOUND	**TRUE** if the current boundary condition has any active Euler buckling constraints.
ACTBOUND	ABOUND	**TRUE** if the current boundary condition has any active constraints.
ACTDYN	ABOUND	**TRUE** if the current boundary condition has any active frequency constraints.
ACTFLUT	ABOUND	**TRUE** if the current boundary condition has any active flutter constraints.
ACTPNL	ABOUND	**TRUE** if the current boundary condition has any active panel buckling constraints.
ACTUAG	AROSNSDR	**TRUE** if the current boundary condition has any active displacements or accelerations.
ACTUAGG	MAKDFU	**TRUE** if the current boundary condition has any active displacement or stress constraint sensitivities.
AEFLG	SAERO	Logical array which indicates whether the current **SAERO** subscript value has aeroelastic effectiveness constraints applied to it.
APPCNVRG	DESIGN ACTCON	**TRUE** when the approximate problem was converged in a previous iteration.
GLBCNVRG	ACTCON	**TRUE** when global convergence has been reached.
K2GGFLG	MK2GG	Set **TRUE** in **MK2GG** if a **K2GG** matrix is input for the current boundary condition.
LOOP	—	General logical used to control **DO-WHILE** loops.
M2GGFLG	MK2GG	Set **TRUE** in **MK2GG** if an **M2GG** matrix is input for the current boundary condition.
PFLAG	ACTCON DESPUNCH	Set **TRUE** in **ACTCON** if **DESPUNCH** needs to punch a new model.

4.4.2. The Solution Algorithm

Finite element structural analysis, which forms the core of the ASTROS system, requires the manipulation of large matrices. The MAPOL control language is designed with this requirement in mind and, therefore, is able to directly support the manipulation of matrices. Consequently, the majority of the MAPOL sequence consists of matrix equations. The algorithmic nature of the MAPOL syntax allows the reader to follow these matrix operations fairly easily, and the notation roughly follows that used in the Theoretical Manual. Therefore, the focus of this section is the description of modules called by the MAPOL sequence.

There are a number of engineering and utility modules called to perform tasks associated with the several analysis disciplines supported by the ASTROS system. Table 4-8 of Section 4.4.2.1 lists the modules defined to the ASTROS executive system, and provides a brief description of each. Not all of these modules appear in the standard solution sequence. These are included in the table to ensure its completeness and usefulness in modifying the standard sequence. The use of these modules is discussed in more detail in the section on modifying the standard MAPOL sequence and are more fully documented in the Programmer's Manual. The brief descriptions of the remaining segments of the standard algorithm that follow, coupled with the inherent readability of MAPOL syntax, provide a complete picture of the flow through the standard sequence.

4.4.2.1. MAPOL Engineering and Utility Modules

This section contains a brief description, shown in Table 4-8, of each of the MAPOL addressable modules defined to the ASTROS executive system. The intrinsic mathematical functions of the MAPOL language are not included. The **TYPE** column indicates whether a module is used for **ENG**ineering functions, **MAT**rix manipulations, **UTIL**ity operations, or to address the **eBASE** database.

Table 4-8. Summary of ASTROS Modules

MODULE	TYPE	DESCRIPTION
ABOUND	ENG	Generates flags for the current boundary condition during the sensitivity calculation. These are then returned to the executive sequence to direct the execution of the required sensitivity analyses.
ACTCON	ENG	Determines whether the design task has converged. If the optimization has not converged, this module selects which constraints are to be included in the current redesign. On termination or print request, this routine computes the values of the local design variables.
AEROEFFS	ENG	Evaluates aeroelastic effectiveness sensitivities.
AEROSENS	ENG	Computes the sensitivities to active strength constraints and/or aeroelastic effectiveness constraints for active steady aeroelastic optimization boundary conditions.
AMP	ENG	Computes the discipline dependent unsteady aerodynamic matrices for **FLUTTER** and **GUST** analyses.
ANALINIT	ENG	Initializes the final analysis pass.
APFLUSH	ENG	Flushes the current values of user function responses and gradients at the beginning of each design iteration.
APPEND	MAT	Appends one matrix to another.
AROSNSDR	ENG	Driver for **SAERO** sensitivity analysis.
AROSNSMR	ENG	Merges **SAERO** sensitivities for each subscript into **[MATOUT]** in case order for active subcases.
BCBGPDT	ENG	Builds the boundary condition dependent grid point coordinate relation, **BGPDT**, for the specified boundary condition.
BCBULK	ENG	Builds boundary condition dependent matrices, transfer functions and initial conditions.
BCEVAL	ENG	Evaluates the constraints of **PBAR1** cross-sectional parameters.
BCIDVAL	ENG	Converts the boundary condition index value (**BC**) into the user assigned value.

Table 3-8. Summary of ASTROS Modules — Continued

MODULE NAME	TYPE	DESCRIPTION
BOUND	ENG	Returns flags to the MAPOL sequence that define the matrix reduction path for the current boundary condition.
BOUNDUPD	ENG	Updates boundary condition definitions.
CEIG	ENG	Computes the complex eigenvalues and eigenvectors of a matrix.
COLMERGE	MAT	Merges two or more submatrices into a single matrix based on column partitioning vectors.
COLPART	MAT	Partitions a matrix into two or more submatrices based on column partitioning vectors.
CONORDER	ENG	Reorders constraints in boundary condition order to match the order in which constraint sensitivities are computed.
DCEVAL	ENG	Evaluates displacement constraints in the current boundary condition.
DDLOAD	ENG	Computes the sensitivities of design dependent loads for active boundary conditions.
DECOMP	MAT	Decomposes a matrix into its triangular factors.
DESIGN	ENG	Performs redesign by math programming methods based on the current set of active constraints and constraint sensitivities.
DESPUNCH	UTIL	Writes new modified Bulk Data entries for the current design iteration to the PUNCH file.
DMA	ENG	Assembles the direct and/or modal stiffness, mass and/ or damping matrices including extra point degrees of freedom for dynamic analysis disciplines.
DYNLOAD	ENG	Assembles the direct and/or modal time and/or frequency dependent loads including extra point degrees of freedom for dynamic response disciplines.
DYNRSP	ENG	Computes the direct or modal displacements, velocities and accelerations for **TRANSIENT** and **FREQUENCY** analyses.
EBKLEVAL	ENG	Evaluates Euler buckling constraints.
EBKLSENS	ENG	Evaluates Euler buckling constraint sensitivity.
EDR	ENG	Computes the stresses, strains, grid point forces and strain energies for elements selected for output for the particular boundary condition.
EMA1	ENG	Assembles the linear element stiffness and mass matrices (stored in the KELM and MELM entities) into the linear design sensitivity matrices **DKVI**, **DMVI**.
EMA2	ENG	Assembles the element stiffness and mass matrix sensitivities (stored in the **DKVI** and **DMVI** entities) into the global stiffness and mass matrices for the current design iteration.
EMG	ENG	Computes the element linear stiffness, mass, thermal load and stress component sensitivities for all structural elements.
EXIT	UTIL	Terminates the execution of the MAPOL sequence. Useful to terminate modified MAPOL sequences.
FBS	MAT	Performs the forward-backward substitution to solve systems of linear equations.
FCEVAL	ENG	Evaluates the current value of all frequency constraints.
FLUTDMA	ENG	Assembles the dynamic matrices for the **FLUTTER** disciplines.
FLUTDRV	ENG	Driver for **FLUTTER** analyses.

Table 3-8. Summary of ASTROS Modules — Continued

MODULE NAME	TYPE	DESCRIPTION
FLUTQHHL	ENG	Processes the **[QKKL]** matrix with normal modes for **FLUTTER**.
FLUTSENS	ENG	Computes the sensitivities of active flutter constraints in the current active boundary condition.
FLUTTRAN	ENG	Performs flutter analyses in the current boundary condition and evaluates any flutter constraints if it is an optimization boundary condition with applied flutter constraints.
FNEVAL	ENG	Evaluates user-defined objective and constraint functions.
FPKEVL	ENG	Compiles the **FUNCTION** Packet and instantiates user functions that have been invoked in Solution Control.
FREDUCE	ENG	Reduces the symmetric or asymmetric f-set stiffness, mass and/or loads matrix to the a-set if there are omitted degrees of freedom.
FREQSENS	ENG	Computes the sensitivities of active frequency constraints in the current active boundary condition.
FSD	ENG	Performs redesign by fully stressed design methods based on the set of applied stress constraints. All other applied constraints are ignored.
GDR1	ENG	Computes the shifted stiffness matrix and the rigid body transformation matrix **[GGO]** to be used in Phase 2 of Generalized Dynamic Reduction.
GDR2	ENG	Computes the orthogonal basis **[PHIOK]** for the general Krylov subspace to be used in Phase 3 of Generalized Dynamic Reduction.
GDR3	ENG	Computes the transformation matrix **[GSUBO]** for Generalized Dynamic Reduction.
GDR4	ENG	Computes transformations between displacement sets useful for data recovery from Generalized Dynamic Reduction.
GDVGRAD	ENG	Computes design variable sensitivity for intricsic functions.
GDVRESP	ENG	Computes design variable responses for intrinsic functions.
GFBS	MAT	Performs the forward-backward substitution phase to solve general systems of linear equations that have been decomposed with module **DECOMP**.
GPSP	ENG	Processes the n-set stiffness matrix to identify singularities and, if requested, automatically remove them.
GPWG	ENG	Grid point weight generator module.
GREDUCE	ENG	Reduces the symmetric g-set stiffness, mass or loads matrix to the n-set if there are multipoint constraints in the boundary condition.
GTLOAD	ENG	Assembles the current static applied loads matrix for any statics subcases in the current boundary condition from the constant simple load vectors and the design dependent load sensitivities.
IFP	ENG	Reads the Bulk Data File and loads the input data to relations. Computes the external coordinate system transformation matrices and creates the basic grid point data. Also performs bandwidth minimization.
INERTIA	ENG	Computes the rigid body accelerations for statics analyses with inertia relief.
ITERINIT	ENG	Initializes the **CONST** relation for the current iteration.
LAMINCON	ENG	Computes constraint values for laminate thickness constraints.

Table 3-8. Summary of ASTROS Modules — Continued

MODULE NAME	TYPE	DESCRIPTION
LAMINSNS	ENG	Computes constraint sensitivities for laminate thickness constraints.
LODGEN	ENG	Assembles the simple load vectors and simple load sensitivities for all applied loads in the Bulk Data File.
MAKDFU	ENG	Assembles the sensitivities to the displacements of active stress and displacement constraints in the current active boundary condition.
MAKDFV	ENG	Assembles the sensitivities of active thickness constraints.
MAKDVU	ENG	Multiplies the stiffness or mass design sensitivities by the active displacements or accelerations.
MAKEST	ENG	Generates the element summary relational entities for all structural elements. Determines the design variable linking and generates sensitivities for any thickness constraints.
MERGE	MAT	Merges two or more submatrices into a single matrix based on row and column partitioning vectors.
MKAMAT	ENG	Assembles the constraint sensitivity matrix from the sensitivity matrices formed by **MAKDFU** and the sensitivities of the displacements for active static load conditions in the current active boundary condition.
MKDFDV	ENG	Computes the sensitivity of PBAR1 cross-sectional parameters with respect to design variables.
MKDFSV	ENG	Computes the matrix [DFSV] which contains the design variable nonlinear s-matrix derivatives related to active stress and strain constraint sensitivity terms.
MKPVECT	MAT	Generates partitioning vectors.
MKUSET	ENG	Generates the structural set definition entity **USET** for each boundary condition and forms the partitioning vectors and transformation matrices used in matrix reduction.
MK2GG	ENG	Interprets solution control and generates the **[M2GG]** and **[K2GG]** matrices if necessary.
MSWGRESP	ENG	Computes element mass or weight intricsic response function.
NLEMA1	ENG	Assembles the element design variable linear and nonlinear stiffness and mass matrices into the design sensitivity matrices.
NLEMG	ENG	Computes the element nonlinear stiffness, mass, thermal load and stress component sensitivities for all structural elements.
NLLODGEN	ENG	Assembles the simple nonlinear load vectors and simple nonlinear load sensitivities for all applied loads in the Bulk Data File.
NREDUCE	ENG	Reduces the symmetric n-set stiffness, mass or loads matrix to the f-set if there are single point constraints in the boundary condition.
NULLMAT	ENG	Breaks database equivalences from previous boundary conditions.
OFPAEROM	ENG	Solves for the **SAERO** applied loads and displacements on aero boxes for output requests.
OFPDISP	ENG	Prints selected displacements, velocities and/or accelerations from any analyses in the current boundary condition.
OFPALOAD	ENG	Solves for the **SAERO** applied loads and constraint forces for output processing.
OFPDLOAD	ENG	Processes output requests for dynamics loads (transient frequency, and gust).
OFPEDR	ENG	Prints selected element stress, strain, force and/or strain energies from any analyses in the current boundary condition.

Table 3-8. Summary of ASTROS Modules — Continued

MODULE NAME	TYPE	DESCRIPTION
OFPGRAD	ENG	Processes output requests for objective and constraint gradients.
OFPLOAD	ENG	Prints selected applied external loads from any analyses in the current boundary condition.
OFPMROOT	ENG	Processes output requests for normal modes.
OFPSPCF	ENG	Processes output requests for single-point constraint forces.
PARTN	MAT	Partitions a matrix into two or more submatrices based on row and column partitioning vectors.
PBKLEVAL	ENG	Evaluates panal buckling constraints.
PBKLSENS	ENG	Evaluates panal buckling constraint sensitivity.
PFBULK	ENG	Performs a number of preface operations to form additional collections of data.
QHHLGEN	ENG	Computes the discipline dependent unsteady aerodynamic matrices for **GUST** analyses in the modal structural system.
RBCHECK	ENG	Outputs to the print file the rigid body checks computed for each support point.
RECEND	CADDB	Terminates setting conditions on a MAPOL relational access.
RECOVA	ENG	Recovers the symmetric or asymmetric f-set displacements or accelerations if there are omitted degrees of freedom.
REIG	ENG	Computes the eigenvalues and eigenvectors of the system as directed by the boundary **METHOD** selection.
RELCND	CADDB	Sets conditions on attribute values for MAPOL retrieval of relational entities.
RELADD	CADDB	Adds a tuple to an entity opened with **RELUSE**.
RELEND	CADDB	Closes an entity opened from the MAPOL sequence using **RELUSE**.
RELGET	CADDB	Retrieves a relational tuple into execution memory for a relation opened for use in the MAPOL sequence.
RELUPD	CADDB	Performs a relational update from execution memory of a tuple retrieved using **RELGET**.
RELUSE	CADDB	Opens a relational entity for access from the executive sequence.
ROWMERGE	MAT	Merges two or more submatrices into a single matrix based on row partitioning vectors.
ROWPART	MAT	Partitions a matrix into two or more submatrices based on row partitioning vectors.
SAERO	ENG	Solves the trim equation for steady aeroelastic trim analyses. Computes the rigid and flexible stability coefficients for steady aeroelastic analyses and the aerodynamic effectiveness constraints for constrained optimization steady aerodynamic analyses.
SAERODRV	ENG	Driver for **SAERO** disciplines.
SAEROMRG	ENG	Merges the **SAERO** results into **[MATOUT]** in case order.
SCEVAL	ENG	Computes the stress and/or strain constraint values for the statics or steady aeroelastic trim analyses in the current boundary condition.
SDCOMP	MAT	Decomposes a symmetric matrix into its lower triangular factor and a diagonal matrix.
SHAPEGEN	UTIL	Generates a set of **SHAPE** entries based on the element centroidal locations for a group of selected elements.

Table 3-8. Summary of ASTROS Modules — Continued

MODULE NAME	TYPE	DESCRIPTION
SOLUTION	ENG	Interprets the solution control packet, forms the **CASE** entity and outputs certain key parameters to the executive sequence.
SPLINES	ENG	Generates interpolation matrix relating displacements and forces between the steady aero and structural models.
SPLINEU	ENG	Generates interpolation matrix relating displacements and forces between the unsteady aero and structural models.
STEADY	ENG	Computes rigid unit forces and aeroelastic corrections for steady aero.
STEADYNP	ENG	Computes rigid trimmed forces for non-planar models.
TCEVAL	ENG	Computes the current values of thickness constraints for this optimization iteration.
TRNSPOSE	MAT	Transposes a matrix.
UNSTEADY	ENG	Computes unsteady generalized forces.
USETPRT	UTIL	Prints the structural set definition table from the **USET** entity for the specified boundary condition.
UTGPRT	UTIL	Prints several specific matrix entities in an interpretable form.
UTMPRG	UTIL	Purges matrix entities.
UTMPRT	UTIL	To print any matrix entity.
UTRPRG	UTIL	Purges relational entities.
UTRPRT	UTIL	To print any relational entity. Only the first twelve attributes are printed and character attributes must be eight characters in length or they will be ignored.
UTUPRG	UTIL	Purges unstructured entities.
UTUPRT	UTIL	To print any unstructured entity.
WOBJGRAD	ENG	Computes the default objective function (weight) sensitivity.
YSMERGE	ENG	A special purpose merge utility for merging **YS**-like vectors (vectors of enforced displacements) into matrices for data recovery.

4.4.2.2. The Preface Segment

In the context of optimization, invariant data is computed only once and reused subsequently for each iteration. This is the underlying principle used in defining preface modules. In each instance, the data generated are invariant with respect to the design variables.

The preface segment begins with a call to the solution control interpreter to determine the number and types of analyses to be performed. The input file processor (**IFP**) is then called. The element connection data and element matrices are then formed. **PFBULK** is then called to perform error checking operations on a variety of user input data. The **EMA1** is called to compute the design invariant stiffness and mass sensitivities to the global design variables. Then the simple loads and load sensitivities are computed in **LODGEN**. If any planar static aerodynamic analyses are requested in the solution control, the **STEADY** and **SPLINES** modules are called to create the aerodynamic matrices required for the aeroelastic analysis. Finally, unsteady aerodynamics matrices are computed for **GUST** and **FLUTTER** analyses in **UNSTEADY**, **AMP** and **SPLINEU**.

4.4.2.3. The Analysis/Optimization Segments

The remainder of the MAPOL algorithm consists of the optimization and analysis segments. Any particular boundary condition is either an optimization boundary condition (implying that the quantities computed in the disciplines selected in the solution control are constrained and that the structure is to be optimized subject to those constraints) or an analysis boundary condition. The design of the ASTROS system requires that all optimization boundary conditions precede any analysis boundary conditions. The analysis segment (labeled the "final analysis") is intended to follow an optimization with analyses in disciplines whose output values are not constrained but are of interest to the designer or to provide the user with an opportunity to view additional output not desired within the optimization loop. Also the analysis segment can be used on a stand alone basis to perform any desired analyses.

Both the optimization and analysis segments consist of an initial loop on the number of boundary conditions. The analyses in these loops support all the disciplines currently available in the ASTROS system and differ only in the respect that the analysis segment does not have calls to constraint evaluation modules and the optimization segment has convergence tests and design iteration initialization outside the analysis boundary condition loop. The first step in these loops is to assemble the boundary condition dependent number of degrees of freedom (extra points are BC selectable in ASTROS). Then additional **PFBULK**-like operations are performed in **BCBULK** to ensure that BC-dependent user input is correct. Then the global stiffness and/or mass matrices are assembled and, if needed, the global loads matrix. Following these tasks, there are several BLOCK IF statements on the various dependent structural sets. In executing each block, the required matrix partitions and reductions are performed. Once the reduced matrices have been obtained for the analyses being performed within the loop, the lowest level response quantities (e.g. displacements, eigenvalues, etc.) are computed. Following the solution, the execution proceeds through another group of dependent set BLOCK IF's to recover the solution vectors to the global set. At this point, the analysis segment is completed with calls to the output file processor modules to compute and output high level response quantities (e.g. stresses).

In the optimization phase of the optimization segment, the **ACTCON** module determines the status of the global convergence flag **CONVERGE** and, if the optimization is not complete, the redesign task is per-

formed. Three redesign methods are supported by the standard sequence and selected through the Solution Control. If the option for Fully Stressed Design (**FSD**) is selected, the redesign is performed in the **FSD** modules. The mathematical programming method only requires sensitivity information. In this case, the sensitivities of the active constraints (chosen by **ACTCON** based on the **NRFAC** and **EPS** parameters) are computed.

The sensitivities of the active constraints which are explicit functions of the design variables are computed first in the **MAKDFV** module. Then the second boundary condition loop within the optimization segment begins. The **ABOUND** routine determines the types of active constraints in each boundary condition and outputs logic flags to control the subsequent sensitivity computations. Then boundary condition dependent constraints which are explicit functions of the design variables (frequency and flutter) are computed. Next, the sensitivities of the constraints to the displacements for those **STATICS** constraints which are explicit functions of the displacements (e.g., stress and displacement constraints) are computed using the **MAKDFU** module. For these types of constraints, the product of the stiffness sensitivities and the displacements and the mass sensitivities and the accelerations are also computed and modified appropriately to account for design dependent loads and inertia relief. The resulting matrix is then reduced and used to solve for the sensitivities of the displacements to the design variables. This matrix is recovered to the free displacement set in a manner similar to the recovery of the outputs in the analysis phase of the optimization segment. The final module within the boundary condition loop for sensitivity evaluation is **MKAMAT**. Within this module the constraint sensitivities to the design variables are formed from the product of the two sensitivity matrices previously obtained.

For static aeroelastic analyses, a procedure similar to that for **STATICS** is used twice: once for "pseudo-displacements" that allow computation of aeroelastic effectiveness derivatives and once for real displacements that support the static strength constraints. The static aeroelastic sensitivity code is further complicated by the generality of the aeroelastic correction matrix selections, which are subcase dependent.

After all the active optimization boundary conditions have been processed, the **DESIGN** module is called. Within this module, the approximate design problem is arranged for use by the optimizer and is solved. Following convergence of the approximate problem, execution returns to the top of the optimization loop and a complete reanalysis of all the boundary conditions is performed. Once completed, the **ACTCON** module determines if the global problem is converged and, if so, sets the global convergence flag to TRUE causing the execution to pass to the top of the analysis segment. If any analysis boundary conditions exist, they will be processed in a manner similar to the analysis phase of the optimization segment. After performing the requested final analyses (if any) the executive system terminates the ASTROS execution.

4.4.3. Modifying the Standard MAPOL Sequence

The standard MAPOL sequence is provided to allow you to run the ASTROS system without detailed knowledge of the MAPOL language or the standard sequence. There is not, however, any requirement that the standard sequence be used. Chapter 8 outlines the procedure for writing a valid MAPOL sequence, and any series of syntactically correct MAPOL statements may be used to direct the ASTROS procedure. All the engineering, utility and matrix manipulation modules shown in Subsection 4.4.2.1 are available to any MAPOL sequence used to direct the system. In addition, there are a number of intrinsic functions, such as **SIN** and **ABS**, that are also available. Their use is detailed in the MAPOL Programming chapter. The sophisticated MAPOL user is thus provided with a very flexible control language to

manipulate the ASTROS system. This Section describes simple modifications to the standard algorithm to print out additional data items, to fine tune the optimization algorithm and to restore an ASTROS analysis that was partially executed on a previous run. No set of examples, however, can possibly indicate the full range of available capabilities; the user is therefore cautioned not to be overly constrained by this discussion.

In order to avoid vast quantities of output and to limit the execution time, the standard output is kept to a minimum. Several utilities, listed in Section 4.4.2.1, can, however, be inserted in the standard sequence to output data stored on the database. In addition, a utility has been written to print out the structural set definition table to aid in the debugging of the structural model. The **UTMPRT**, **UTGPRT**, **UTRPRT** and **UTUPRT** print utilities dump the contents of specified database entities to the user's output file. These can be used anywhere in the MAPOL sequence after the specified entity has been filled with data. The **USETPRT** utility provides the user with the ability to print the structural set definition table (**USET**) in a format which aids in debugging the structural model. These utilities provide the user with some simple tools to allow closer interaction with the data stored on the database and to provide capability to more closely track the execution.

The print utilities provide data visibility without modifying the basic execution of the standard sequence. At a slightly more complex level, the user might desire to fine tune the optimization procedure or to track the iterations of the optimizer more closely. Table 4-4 includes a number of parameters which are used by ASTROS to direct the optimization. All of these parameters can be modified through the **OPTIMIZE** command in solution control. That modification, however, only occurs once. Any of these parameters can be changed by the user at any point in the MAPOL sequence. For example, the **MOVLIM** parameter could be changed to a different value after the fifth iteration by placing the following statement immediately after the **WHILE** test on **GLBCNVRG**:

```
IF NITER > 5 MOVLIM = 1.5;
```

Obviously, the conditional testing can become as complex as the MAPOL programmer desires.

The brief discussion above does not begin to describe all the options open to the sophisticated ASTROS user. It does, however, outline some of the most commonly performed modifications to the standard MAPOL algorithm. The concepts described can be extended to a large number of similar changes; e.g., modifying the input dynamic pressure value within the MAPOL sequence could be done to avoid re-running the base run of an ASTROS execution. At a more advanced level, the MAPOL relational database entity utilities can be used to directly modify the design variable values or objective sensitivities.

4.4.4. Restart Capability

Although ASTROS does not support a formal restart capability, this does not imply that restarts cannot be performed in ASTROS. The restart capability in ASTROS is limited in that you must use a modified MAPOL sequence in order to terminate the system early and the restarted job **MUST** use a tailored MAPOL sequence to restart the job at the desired point. Otherwise, there are no limits to what can be done by the experienced user. The ASTROS restart capability is best described as a full featured ***Manual Restart*** — ASTROS does not have an ***Automated Restart***.

There are several reasons why you may wish to suspend an ASTROS execution and then perform a restart, and the program supports this basic capability. For example, you may wish to examine the progress of a design after each optimization iteration. With the run stopped, you would then have the freedom to use eSHELL and alter ASTROS data to redirect the optimization path, if desired. As a second example, you may suspend execution and, again using eSHELL, replace ASTROS-computed data with their external equivalent (such as the **QHHL** or **QKKL** matrices of unsteady aerodynamic influence coefficients). Clearly, the ability to suspend/restart executions in combination with the eSHELL environment opens limitless possibilities.

Without an automated restart, however, you are responsible for ensuring that several requisite tasks are completed. These are

- ☞ Ensuring that the run-time database has the proper **STATUS** on suspension and on restart.
- ☞ Selecting where in the MAPOL sequence to suspend execution.
- ☞ Writing a MAPOL sequence to restart execution. This may or may not be a modification to the standard sequence.
- ☞ Ensuring that those scalar variable(s) that are common to both the original and the restart MAPOL sequences are initialized to the correct value(s)

Each of these tasks are discussed in the following sections.

4.4.4.1. Ensuring proper STATUS of the run-time database

When suspending execution, the run-time database must be saved. ASTROS stores all the information that it has generated during the execution on the run-time database and, on any restart, the downstream modules will expect that those data will exist when they are executed in the restart environment. The run-time database is also the location of the data that the user may wish to modify or add to using eSHELL prior to initiating the restart. Saving the run-time database is done by selecting a **STATUS** of **NEW** (with the optional user parameter, **KEEP**, if required on the local host) on the **ASSIGN DATABASE** entry. For example,

```
ASSIGN DATABASE CALVIN HOBBES NEW
```

```
ASSIGN DATABASE CALVIN HOBBES NEW KEEP
```

When restarting ASTROS using an existing database whose contents are to be preserved, ASTROS must be notified to attach the existing run-time database files without re-initializing them. This is done by selecting a **STATUS** of **OLD** on the **ASSIGN DATABASE** entry. For example

```
ASSIGN DATABASE CALVIN HOBBES OLD
```

If the **STATUS** of **OLD** is not given, existing database files are typically overwritten by the system. The **STATUS** flag indicates the status of the ***data*** not of the ***files*** so the files may exist with a **STATUS** of **NEW** and will result in the database contents being replaced by the new execution.

4.4.4.2. Suspending/Restarting Execution

ASTROS execution is controlled by the MAPOL sequence that is supplied in the MAPOL packet. This may be the standard sequence (if the packet is omitted), an edited version of the standard sequence, or a user supplied sequence. To suspend execution, a MAPOL sequence must be defined which results in clean termination (one without fatal errors) of the ASTROS execution. This may be the standard execution or, more typically, an edited standard sequence or even a standalone MAPOL program.

Most commonly, the suspension is performed by editing the standard MAPOL sequence and inserting an **EXIT** call after the last line that is to be executed in the current execution. Alternatively, if no missing **IF-THEN-ENDIF**s or **ENDDO**s result, portions of the sequence can simply be deleted. Some care should be taken in suspending execution in the middle of **DO** and **DO WHILE** loops or block **IF**s. Although possible to do, suspensions during execution of these repetitive segments can leave the system in a state that is more difficult to reinitialize on the restart exectution. Some experience with MAPOL and with ASTROS is needed before attempting these more complex suspensions. Suspending execution at the beginning or end of the preface, analysis phase (of either the optimization or final analysis segements) or the sensitivity phase is most likely to yield success.

To restart the execution, you ***must*** generate a special MAPOL program either by editing the standard sequence or by writing a new sequence. The restart execution of ASTROS does not have any information on where the initial execution terminated. Only the data on the database is saved (i.e., available for the current execution). Obviously, the new execution may start up at any point the user wishes and need not be associated with the area where the initial run terminated (although only experienced ASTROS users should attempt to drastically alter the flow of the MAPOL sequence).

To generate the special MAPOL sequence, the user may use a **GOTO** statement to jump ahead in the standard sequence to the restart point or the user may use the **EDIT** commands to delete those initial sections that no longer need execution. The latter is typically the case when the preface segment is "saved" for restart. If the user deletes lines, care should be taken not to delete half of a looping construct or block IF since that will result in a MAPOL compilation error. The restart MAPOL sequence must also contain any new statements that are required to reset values of MAPOL scalar parameters.

4.4.4.3. Resetting MAPOL Parameters

In the ASTROS system, all the values of MAPOL variables are stored on the database. For the complex data types like RELATIONs and MATRICEs, this is obvious, since their data resides on the database for

eSHELL execution or other processing. Less obviously, the simple data types like **REAL** and **INTEGER** (including arrays) are also saved on the database. These data are not easily viewed in the eSEHLL context, but are saved in a way that the MAPOL compiler can recognize. When a restart job is performed, the existence of these old data causes the MAPOL compiler to determine the correspondence between the original data and the new MAPOL sequence. Whenever a variable of the same name and type is found, its initial value is recovered from the old data thus "restoring" the value of the original variable to the last value it contained.

In the restart execution, however, the user must make sure that the ***last*** value of the variable is the desired ***initial*** value for restart. In some cases, the variables contain "invariants" like the variable NDV which contains the number of global design variables. In other cases, like BC, the variable is a loop counter that should be reset. The MAPOL sequence may perform the reinitialization automatically (for example if a DO loop is re-executed for values 1 through 10, the do loop counter will be reset to 1 no matter what value it contains). If, however, the restart MAPOL omits the loop, the last loop counter that was achieved will be stored in the loop counter on restart.

Determining which MAPOL parameters should be left alone and which should be reset (rather than default to their last value) is the challenge of the manual restart. Tables 4-2 through 4-7 (section 4.4.1) of this Manual have a list of all the MAPOL parameters. These tables list each parameter and give a description of how and where the parameter is used. Together with the ASTROS Programmer's Manual, which document the actions that occur in each ASTROS module, the user can decide which parameters should be reset and which should be allowed to default to the value set in the initial execution.

4.5. MAPOL PROGRAM LISTING

The current MAPOL listing is not given here because it is subject to change, if you wish to obtain the current listing, you may print the file MAPOLSEQ.DAT, which is delivered with your software, or use the method described in Section 4.2. Contact your UAI Systems Support Specialist for information about this file.

Chapter 5.

THE SOLUTION CONTROL PACKET

The solution control packet provides the means by which the user selects the optimization and analysis tasks to be performed by the ASTROS system, their order of execution and the engineering data related to each. The solution control commands are analogous in purpose to the NASTRAN Case Control commands but they are very different in form and subtly different in interpretation. Understanding the differences between ASTROS and NASTRAN in the area of solution control is fundamental in understanding multidisciplinary optimization in the ASTROS system because the solution control command structure follows directly from the ASTROS capability to perform multidisciplinary analyses in a single run. It is ***critical*** that the user clearly understand the subtleties of solution control syntax and hierarchies. This section, therefore, augments the presentation of the solution control mechanics with a discussion of the design considerations that are embodied in the solution control commands. The detailed definition of all solution control commands follows at the end of the chapter.

In ASTROS, the solution control is very closely linked to the structure of the standard MAPOL sequence. It may be advantageous for the beginning user to read the standard MAPOL sequence discussion in the preceding section and to study the Theoretical Manual discussion of multidisciplinary optimization before reading the remainder of this section.

The solution control packet is initiated with the keyword **`SOLUTION`** which follows the DEBUG and MAPOL packets (if present) in the input data stream. The packet is terminated when the BULK DATA packet, or the end of the input stream, is encountered. The data are composed of solution control statements which can begin in any column and can extend over multiple physical records. Each statement is formed from a combination of keywords separated by blanks or commas as indicated in the detailed syntactical descriptions at the end of the chapter. Further, each command keyword can be abbreviated by the first four (or more) characters of the keyword. The solution control packet follows a prescribed hierarchy with the following levels:

```
INITIAL LEVEL (Level 1)
   TYPE OF BOUNDARY CONDITION (Level 2)
      BOUNDARY CONDITION(S) (Level 3)
         DISCIPLINE(S) (Level 4)
```

Each of these levels is discussed in the following sections and compared and contrasted to their NASTRAN counterparts. In addition to these hierarchical commands, there are commands for output processing that can occur at several levels in the hierarchy. This section presents the available commands and output quantities, but the reader is referred to Chapter 5 of this document for the in-depth presentation of ASTROS output processing.

The hierarchical nature of solution control means that, if the user enters a command at one level in the hierarchy, it remains in effect at ***all*** subsequent levels ***at*** or ***below*** the current one unless overridden. If it is overridden at the ***same*** level, that overwrites the original command. If, on the other hand, the command is overridden at a ***lower*** level, it only supercedes the original command for the duration of that level and lower levels. Solution Control reverts to use the higher level default after the lower level has been left. Table 5-1 describes how the commands move from one level to the next and the defaults that they use in each.

Table 5-1. Levels of Solution Control

CURRENT LEVEL IS:	INCREASING LEVELS			DECREASING LEVELS		
	IF COMMAND IS:	USE DEFAULTS FROM:	THEN MOVE TO:	IF COMMAND IS:	USE DEFAULTS FROM:	THEN MOVE TO:
LEVEL 1 (Initial)	**ANALYZE OPTIMIZE**	LEVEL 1	LEVEL 2			
LEVEL 2	**BOUNDARY**	LEVEL 2	LEVEL 3			
LEVEL 3	**Discipline commands** (e.g.STATICS)	LEVEL 3	LEVEL 4			
LEVEL 4				**Discipline commands** (e.g.STATICS)	LEVEL 3	LEVEL 4
				BOUNDARY	LEVEL 2	LEVEL 3
				END	LEVEL 1	LEVEL 1

The user must be aware of these hierarchies especially when requesting output at higher levels. It is possible to get print requests by *default* where they are not expected if one is not careful with the solution control hierarchy. Another common problem is to place an output request on the wrong side of a level-incrementing solution command thus placing a command at a higher level than expected. Consider the following two examples:

EXAMPLE 1	EXAMPLE 2

OPTIMIZE
  BOUNDARY SPC=1
  LABEL = CASE 1
  STATICS (MECH=10)
  ...
  LABEL = CASE 2
  STATICS (MECH=20)
  ...
  LABEL = CASE 3
  STATICS (MECH=30)
  ...
END
	OPTIMIZE
	  BOUNDARY SPC=1
	  STATICS (MECH=10)
	  LABEL = CASE 1
	  ...
	  STATICS (MECH=20)
	  LABEL = CASE 2
	  ...
	  STATICS (MECH=30)
	  LABEL = CASE 3
	  ...
	END

In example 1, there are three discipline commands, **STATICS**, and three **LABEL** commands, one for each discipline. The indenture in the example helps to explain the results of these commands. The first **STATICS** case will be labelled **CASE 2**, because the **LABEL** command appears at LEVEL 4 with the **STATICS (MECH=10)** command. Similarly, the second **STATICS** case will be labelled **CASE 3**. Finally, the third **STATICS** case will be labelled **CASE 1** because that particular **LABEL** command appeared at LEVEL 3 prior to **STATICS (MECH=10)**. Example 2 illustrates the probable intent of the user. Here, the **LABEL** commands are placed below the **STATICS** command. As a result, the **LABEL**s match the cases.

5.1. OPTIMIZE AND ANALYZE SUBPACKETS

ASTROS has been designed primarily to be an automated design tool, but it can also perform analyses without doing any design. This is reflected in the division of the solution control packet into two subpackets, either of which is optional. The first, or **OPTIMIZE**, subpacket defines the boundary condition(s) and discipline(s) which will generate design constraints to be used in the redesign task. In defining an optimization boundary condition, the user either implicitly or explicitly specifies that constraints be applied to certain (discipline dependent) response quantities. ASTROS then considers the complete set of constraints from all disciplines in all optimization boundary conditions in the redesign task. The second, **ANALYZE**, subpacket defines analyses that are to be performed on the possibly redesigned structure. The **ANALYZE** subpacket is intended to provide the designer with the means to obtain additional output that is not desired during the optimization phase or to perform additional analyses which were not performed in the design task. It can also be used to perform analyses on structures that are not to be designed at all. The form of the solution control packet is then:

```
SOLUTION
   OPTIMIZE
      ...
      ...   Optimization Subpacket
      ...
   END
   ANALYZE
      ...
      ...   Analysis Subpacket
      ...
END
```

If optimization is being performed, the **OPTIMIZE** subpacket must precede the **ANALYZE** subpacket. Any number of boundary conditions and/or disciplines can be performed in either subpacket.

5.2. BOUNDARY CONDITIONS

Each analysis discipline requires a set of physical boundary conditions and, in the case of unrestrained structures, a set of fictitious supports. These are defined in ASTROS in a manner very similar to that in NASTRAN; namely, through the definition of multipoint constraints (**MPC**), single point constraints (**SPC**) and support points (**SUPORT**). Unlike NASTRAN, however, ASTROS requires a more rigorous definition of a boundary condition. The reason for this is that the user must ensure that the system matrices at each stage of matrix reduction up to the analysis set are uniquely defined by the boundary condition specification.

At or below the analysis set, certain disciplines allow looping over families of direct matrix input, damping options, transfer functions, etc. For example, if the user intends to perform a normal modes analysis, a modal transient analysis and a modal flutter analysis in the same boundary condition, ASTROS requires that the modal representation of the system under analysis be the same for each discipline in the boundary condition. This requirement, which is necessary to efficiently perform multidisciplinary analysis, adds a number of additional parameters to the boundary condition definition beyond the **MPC**, **SPC** and **SUPORT** definitions. They include definitions to perform matrix reductions (available in NASTRAN through Bulk Data but not always selectable in the Case Control Packet) as well as selection of additional point degrees of freedom. In NASTRAN, these data are either implicitly selected through the rigid format selection and/or bulk data or are a "discipline option" in the case control packet. While the boundary condition definition in ASTROS appears to be very complex, it is relatively simple if one realizes that the fundamental purpose of the **BOUNDARY** command is to uniquely specify the system level matrices and the matrix reductions that should be performed on them. The ASTROS automatic singularity feature, **AUTOSPC**, is the default in all cases. Unlike NASTRAN, this feature is selectable by boundary condition.

There is one level of *boundary condition* specification which is not treated in the **BOUNDARY** command. It deals with symmetry options which play a restrictive role in multidisciplinary analysis, especially for aerodynamic disciplines. The symmetry options are often limited by the nature of the structural and/or aerodynamic models that are defined in the bulk data packet. For example, if the structural model is a half model only, the user cannot specify that asymmetric structural boundary conditions be analyzed. As a more common example, the user might want to perform an asymmetric aeroelastic analysis with a

structural half model. Unfortunately, this is not possible in ASTROS. Whenever possible, the implicit (model-defined) boundary condition specifications that existed in the NASTRAN bulk data definitions and in the interface between bulk data and solution control have been replaced with solution control dependent options. There are, however, still limitations imposed through the interactions between the model and the solution control on combining symmetric/antisymmetric and asymmetric boundary conditions within a single run. The eleven boundary condition specifications in ASTROS are shown in the following table:

OPTION	DESCRIPTION
AUTOSPC	Controls the automatic singularity processor.
BCID	Optional boundary condition identification number.
CMETHOD	Specifies an **EIGC** bulk data entry which gives eigenvalue extraction data if an eigenanalysis is to be performed.
DYNRED	Invokes dynamic reduction.
ESET	Specifies the extra point DOF's to be included in dynamic response analyses.
INERTIA	Specifies a **JSET** bulk data set for dynamic reduction.
M2GG	Specifies the name of the direct mass matrix input in the structural set (g-set) to be included in ALL analyses.
K2GG	Specifies the name of the direct stiffness matrix input in the structural set (g-set) to be included in ALL analyses.
METHOD	Specifies an **EIGR** bulk data entry which gives eigenvalue extraction data if an eigenanalysis is to be performed.
MPC	Selects multi-point constraints defining dependency relations among specific DOF's.
REDUCE	Defines the DOF's to be retained after a Guyan reduction.
SPC	Selects single point constraints defining DOF's with fixed or prescribed motion.
SUPPORT	Defines DOF's to provide support conditions for free-free modal extraction, inertial relief and aeroelastic analyses.

A boundary condition is defined by the **BOUNDARY** request and one or more of these further specifications, all of which, except **BCID** and **AUTOSPC**, point to bulk data entries. The boundary condition identification number, **BCID**, is only used by the Function Packet (see Section 4) when user-defined constraint functions are defined which span two or more different boundary conditions. Note that all boundary conditions must have identification numbers, or none may have them. User functions may still span boundary conditions by using default **BCID** values. The default is the ordinal numbering of the boundaries from 1 to n.

As enumerated above, the specification of **METHOD** and **ESET** at this level in the hierarchy is in recognition of the fact that a number of the disciplines could require different sets of data for the associated items and it is desirable to group operations with one set of items together. This does, by definition, create a restriction that only one eigenanalysis and only one size of p-size matrices can be accommodated per boundary condition. Examples of boundary definitions are:

```
BOUNDARY SPC = 100
BOUNDARY MPC = 10, SPC = 100
BOUNDARY SPC = 10, MPC = 20, REDUCE = 30, SUPPORT = 40
BOUNDARY SPC = 10, REDUCE = 20, AUTOSPC = NO, METHOD = 100
BOUNDARY SPC = 1, K2GG = STIFF, M2GG = MASS
BOUNDARY SPC = 4, DYNRED = 2, INERTIA = 4
```

Note that all desired specifications are listed and that their order of appearance is not important. At least one option is required.

Several boundary conditions may appear within a given subpacket. For example:

```
ANALYZE
   BOUNDARY SPC = 10
      STATICS (MECH=5)
         ...
         ...
   BOUNDARY SPC = 20, REDUCE = 30, METHOD = 1111
      STATICS (THERM=10)
         ...
         ...
      MODES
         ...
         ...
END
```

In this case, a **STATICS** analysis is performed using the first boundary condition followed by a **STATICS** and modes analysis for the second boundary condition. Note that unlike NASTRAN, the sets of points to be retained in the Guyan reduction and used for the support definition are selected.

The appearance of a **BOUNDARY** command leads to expensive matrix partitioning and decomposition operations. Therefore, some thought should be expended to avoid unnecessary computer resource use. For example, suppose an ASTROS execution was directed to perform static analyses with two boundaries: **SPC=10** and **SPC=20**, and a dynamic analysis with two boundaries: **SPC=10** and **SPC=100**. The direct solution sequence could be:

```
ANALYZE
   BOUNDARY SPC = 10
      STATICS (MECH=10)
   BOUNDARY SPC = 20
      STATICS (MECH=20)
         ...
         ...
   BOUNDARY SPC = 10, METHOD = 30
      MODES
         ...
         ...
   BOUNDARY SPC = 100, METHOD = 40
      MODES
         ...
         ...
END
```

This sequence would cause four separate partitionings of the system level matrices. On the other hand, the sequence:

```
ANALYZE
   BOUNDARY SPC = 10, METHOD=30
      STATICS (MECH=10)
         ...
      MODES
         ...
   BOUNDARY SPC = 20
      STATICS ((MECH=20)
         ...
         ...
   BOUNDARY SPC = 100, METHOD = 40
      MODES
         ...
         ...
 END
```

eliminates one of the four partitioning operations.

5.3. DISCIPLINES

A number of types of analyses, or disciplines, can be performed during a given **ANALYZE** or **OPTIMIZE** boundary condition. In fact, it is this multidisciplinary capability that makes the ASTROS code viable in a preliminary design context. The preceding sections have already alluded to the fact that each of these disciplines has an associated set of commands:

```
ANALYZE
   BOUNDARY SPC = 30
      DISCIPLINE 1
         ...
         ...
      DISCIPLINE 2
         ...
         ...
END
```

A suite of eight disciplines are available in ASTROS as shown in Table 5-2. Of these options, **TRANSIENT** and **FREQUENCY** do not generate any design constraints and so are not useful in **OPTIMIZE** boundary conditions. Should the user wish to see output from these disciplines during the optimization, however, they are supported in the **OPTIMIZE** subpacket.

The standard MAPOL sequence contains almost no restrictions on the combination of disciplines and subcases in a boundary condition. **SAERO** disciplines, for example, require multiple symmetry, Mach number and dynamic pressure dependent correction matrices. The standard algorithm automatically re-sorts the input subcases to solve the maximum number of right hand sides for a given aeroelastic correction matrix. The results are then returned to the order specified by the user, with no limitations imposed. Similarly, the flutter discipline loops over a set of direct dynamic input matrices to accommodate multiple closed loop systems using a single set of structural matrices. The only limits are those of symmetry discussed earlier in which the structural and aerodynamic symmetries should be the same for all subcases in a boundary condition and the restriction to a single transient and a single frequency response per boundary condition.

Table 5-2. Summary of ASTROS Disciplines

DISCIPLINE	DESCRIPTION
STATICS	Static structural analysis
MODES	Normal modes of vibration
SAERO	Steady-state aeroelastic analysis
FLUTTER	Aeroelastic stability analysis
TRANSIENT	Transient response analysis
FREQUENCY	Frequency response analysis

5.3.1. DISCIPLINE OPTIONS

Each of the disciplines requires further options to completely define the execution process. These options point to set IDs in the bulk data packet that define engineering data. For example, the **STATICS** discipline requires that loads information be supplied. This is implemented in ASTROS by a parenthetical "phrase" attached to the **STATICS** discipline:

```
SOLUTION
   OPTIMIZE STRATEGY=FSD
         ...
         ...
      STATICS (MECH=10)
         ...
         ...
END
```

In this case, bulk data applied load entries with a set ID of 10 are used to construct a mechanical load vector in a **STATICS** analysis. In general, the discipline commands have the form:

```
<disc> <type> [<caseid>] [(<option> = <n>, <option> = <n>)]
```

The discipline options that are available are:

OPTION	DESCRIPTION
MECHANICAL **GRAVITY** **THERMAL**	Specify load set IDs for the **STATIC** discipline.
TRIM	Specifies a **TRIM** bulk data entry which gives flight condition information for the **SAERO** discipline.
DCON **DCONSTRAINT**	Specifies the set IDs of constraint bulk data entries that apply for the given discipline.
DCFUNCTION	Specifies the set ID of a **DCFUNC** Bulk Data entry.
STRESS **STRESSCONSTRAINT**	Specifies the set IDs of stress constraint bulk data entries that apply for the given **STATICS** or **SAERO** discipline.
STRAIN **STRAINCONSTRAINT**	Specifies the set IDs of strain constraint bulk data entries that apply for the given **STATICS** or **SAERO** discipline.
DLOAD	Specifies applied loads for the **TRANSIENT** and **FREQUENCY** disciplines.
TSTEP	Specifies the time step for the **TRANSIENT** discipline as well as for the discrete form of the **GUST** discipline.
FSTEP	Specifies the frequencies for the **FREQUENCY** and the harmonic form of the **GUST** discipline.
IC	Specifies the initial conditions that are to be used in the direct method for the **TRANSIENT** discipline.
FFT	Specifies that the Fast Fourier technique is to be used in the **TRANSIENT** or **GUST** disciplines.

OPTION	DESCRIPTION
FLCOND	Specifies parameters for the **FLUTTER** discipline.
CONTROL	Specifies the name of a control surface modifier matrix for flutter analysis.
GUST	Specifies that a gust analysis is to be performed for the accompanying transient or frequency discipline.
K2PP	Specifies an input stiffness matrix on the physical degrees of freedom for **FREQUENCY**, **TRANSIENT** and **FLUTTER** disciplines.
M2PP	Specifies an input mass matrix on the physical degrees of freedom for **FREQUENCY**, **TRANSIENT** and **FLUTTER** disciplines.
B2PP	Specifies an input damping matrix on the physical degrees of freedom for **FREQUENCY**, **TRANSIENT** and **FLUTTER** disciplines.
TFL	Specifies transfer functions that are to be included in **FREQUENCY**, **TRANSIENT** and **FLUTTER** disciplines.
DAMPING	Specifies structural or viscous damping to be used in **FREQUENCY**, **TRANSIENT** and **FLUTTER** disciplines.

The discipline types are:

OPTION	DESCRIPTION
DIRECT	Specifies that the direct method is to be used in the **TRANSIENT** or **FREQUENCY** disciplines.
MODAL	Specifies that the modal method is to be used in the **TRANSIENT** or **FREQUENCY** disciplines.
SYMMETRIC	Specifies that the **SAERO** subcase is to use aerodynamics derived with symmetric conditions about the Y=0 plane.
ANTISYMMETRIC	Specifies that the **SAERO** subcase is to use aerodynamics derived with antisymmetric conditions about the Y=0 plane.

The case identification number, `caseid`, is only used by the Function Packet (see Section 4) when user-defined constraint functions are defined which span two or more different analysis disciplines. Note that all disciplines must have identification numbers, or none may have them. User functions may still span disciplines by using default `caseid` values. The default is the ordinal numbering of the disciplines from 1 to n.

Table 5-3 presents a matrix that defines options and types available for each of the disciplines. In addition, disciplines requiring particular boundary condition specifications are noted; for example, modal disciplines require a **METHOD** specification on the **BOUNDARY** command. The following subsections present each discipline in turn to more explicitly define the discipline options. Most importantly, these subsections present the definition of a "subcase" of the discipline as it is defined in the ASTROS system and present the response quantities that can be constrained in the optimization task.

Table 5-3. Summary of Discipline Options

COMMAND	DISCIPLINE					
	STAT	MODE	SAER	FLUT	TRAN	FREQ
MECH	❍					
GRAV	❍					
THERM	❍					
TRIM			●			
DCONSTRAINT	❍	❍	❍	❍		
DCFUNCTION	❍	❍	❍	❍		
STRESS	❍		❍			
STRAIN	❍		❍			
DLOAD					●	●
TSTEP					●	
FSTEP						●
IC					❍	
FFT					❍	❍
DIRECT					❍	❍
MODAL					❍	❍
FLCOND				●	❍	
GUST					❍	❍
K2PP				❍	❍	❍
M2PP				❍	❍	❍
B2PP				❍	❍	❍
TFL				❍	❍	❍
DAMPING				❍	❍	❍
SYMMETRIC			❍			
ANTISYMMETRIC			❍			
Notes:						
Required Commands:	●					
Optional Commands:	❍					

5.3.2. STATICS Discipline Options

One or more of the **MECHANICAL**, **GRAVITY** or **THERMAL** load specifications must be called out as a discipline option for **STATICS**. Each **STATICS** discipline constitutes one subcase (one load vector) so specifying a combination of load types will generate a linear combination of the selected loads. A reference to the **LOAD** bulk data entry as a **MECHANICAL** load can also be used to obtain linear load combinations. If the **STATICS** discipline appears in the **OPTIMIZE** subpacket, the **DCONSTRAINT** option can be used to refer to **DCONDSP** bulk data entries to apply displacement constraints. Stress constraints defined on **DCONTW**, **DCONTWM**, **DCONTWP**, **DCONVM**, **DCONVMM**, **DCONVMP** are selected by the **STRESSCONSTRAINT** option. Strain constraints defined on **DCONFT**, **DCONFTM**, **DCONFTP**, **DCONEP**, **DCONEPM** and **DCONEPP** are selected by the **STRAINCONSTRAINT** option. All **DCONxxx** bulk data entries, such as **DCONTHK**, that do not have **SETID** fields will be applied to the model in combination with set selectable constraints to make up the set of design constraints. Finally, the **DCFUNCTION** option may be used to select functional constaints that are applied to the **STATIC** responses from the current solution.

5.3.3. MODES Discipline Options

MODES is completely defined for analysis by the **METHOD** boundary specification, which refers to an **EIGR** bulk data entry selecting the eigenvalue extraction method. If, however, the modal analysis is performed in the **OPTIMIZE** subpacket, the **DCONSTRAINT** option can be used to apply frequency constraints through the **DCONFRQ** bulk data entry. Note that more than one frequency can be constrained and that more that one constraint can be placed on the same modal frequency. The user is warned against defining the frequency constraints in such a way as to specify an excluded range of frequencies for a mode; for example, requiring that a modal frequency be below 10 Hz OR above 20 Hz. ASTROS treats all applied constraints as Boolean **AND** statements so the above example would be interpreted by ASTROS as an inconsistent requirement that the frequency be both above 20 Hz and below 10 Hz. All **DCONxxx** bulk data entries, such as **DCONTHK**, that do not have **SETID** fields will be applied to the model in combination with set selectable constraints to make up the set of design constraints. Additionally, the **DCFUNCTION** option may be used to select functional constaints that are applied to the **MODES** responses from the current solution.

In ASTROS, each eigenvector is considered to be a separate subcase. It is important to note in this case that more than one subcase is represented by a single solution control discipline statement. In output requests, therefore, the subcases for which output is desired must be explicitly selected. This is presented in greater detail in Section 5.4 and in Chapter 6.

5.3.4. SAERO Discipline Options

The **SAERO** discipline must have a **TRIM** condition and symmetry type specified in the solution control. The symmetry default is **SYMMETRIC**. For analysis, this selection completes the specification of the discipline with each **TRIM** condition generating one subcase. In the **OPTIMIZE** subpacket, the **DCONSTRAINT** option can be used to select a number of different constraint types which depend on the type of **TRIM** analysis selected. In general the **DCONSTRAINT** can refer to **DCONDSP** bulk data entries for displacement constraints, **DCONCLA** for lift effectiveness constraints, **DCONALE** for aileron effectiveness constraints, **DCONSCF** for stability coefficient constraints and **DCONTRM** for constraints on trim parameters. The **SAERO** discipline ***always*** generates a static displacement field to which any static constraint may be applied. Stress constraints defined on **DCONTW**, **DCONTWM**, **DCONTWP**, **DCONVM**, **DCONVMM**, **DCONVMP** are

selected by the **STRESSCONSTRAINT** option. Strain constraints defined on **DCONFT**, **DCONFTM**, **DCONFTP**, **DCONEP**, **DCONEPM** and **DCONEPP** are selected by the **STRAINCONSTRAINT** option. All **DCONxxx** bulk data entries, such as **DCONTHK**, that do not have **SETID** fields will be applied to the model in combination with set selectable constraints to make up the set of design constraints. Finally, the **DCFUNCTION** option may be used to select functional constaints that are applied to the **SAERO** responses from the current solution.

5.3.5. FLUTTER Discipline Options

The **FLUTTER** discipline must have a flight condition specified in the solution control through the **FLCOND** option. In addition, the **K2PP**, **B2PP**, **M2PP**, **TFL** and **DAMPING** options may be used with or without an **ESET** Boundary Condition option to impose a case-by-case set of additional inputs/degrees-of-freedom for modelling control systems, etc. For analysis, this selection completes the specification of the discipline with each **FLCOND** condition generating up to one "subcase" (consisting of up to one flutter eigenvector) for each Mach number and density ratio if flutter occurs. In the **OPTIMIZE** subpacket, the **DCONSTRAINT** option can be used to select **DCONFLT** bulk data entries to place a required damping limit on each of the roots extracted in the flutter analysis. The **DCFUNCTION** option may also be used to select functional constaints that are applied to the **FLUTTER** responses in the current solution.The actual flutter root and eigenvector cannot be obtained in the **OPTIMIZE** subpacket.

5.3.6. TRANSIENT Discipline Options

The **TRANSIENT** discipline must have time step and load information specified in the solution control through the **TSTEP** and **DLOAD** options. This discipline has no associated constraints and, while it is fully supported in the **OPTIMIZE** subpacket, it will not generate data for use in the re-design task. There are many additional options which can be selected in transient analysis. These are 1) initial conditions, which can be selected through the IC option for **DIRECT** transient analyses; 2) Fast Fourier Transform techniques, which are selected with the **FFT** option; and 3) discrete gust loads, which are applied using the **GUST** option. In each case, the solution control option points to a bulk data entry having the same name. In addition, the **K2PP**, **B2PP**, **M2PP**, **TFL** and **DAMPING** options may be used with or without an **ESET** Boundary Condition option to impose a case-by-case set of additional inputs/degrees-of-freedom for modelling control systems, etc.

In ASTROS, each time step for which output is saved is considered to be a separate subcase. It is important to note that, like the **MODES** discipline, more than one subcase is represented by a single solution control discipline statement. In output requests, therefore, the subcases for which output is desired must be explicitly selected. This is presented in greater detail in section 5.4 and in Chapter 6.

5.3.7. FREQUENCY Discipline Options

The **FREQUENCY** discipline is very similar to the **TRANSIENT** discipline presented in the preceding subsection. Frequency step and load information are specified in the solution control through the **FSTEP** and **DLOAD** options. This discipline has no associated constraints and, while it is fully supported in the **OPTIMIZE** subpacket, it will not generate data for use in the re-design task. There are two additional options which can be selected in frequency response analysis. These are 1) Fast Fourier Transform techniques, which are selected with the **FFT** option; and 2) harmonic gust loads, which are applied using the **GUST** option. In each case, the solution control option points to a bulk data entry having the same name. In addition, the **K2PP**, **B2PP**, **M2PP**, **TFL** and **DAMPING** options may be used with or without an

ESET Boundary Condition option to impose a case-by-case set of additional inputs/degrees-of-freedom for modelling control systems, etc.

In ASTROS, each frequency step for which output is saved is considered to be a separate subcase. It is important to note that, like the **MODES** discipline, more than one subcase is represented by a single solution control discipline statement. In output requests, therefore, the subcases for which output is desired must be explicitly selected. This is presented in greater detail in Subsection 5.4 and in Chapter 6.

5.4. OUTPUT REQUESTS

Most analysis disciplines in ASTROS have response quantities (displacements, stresses, strains, etc.) computed at either grid points, structural elements or aerodynamic elements. The user can select that these results be written to the print (output) file through the **PRINT** command and its associated options or written to a punch file through the **PUNCH** command. In addition, there are a number of solution control commands that can be used to label the output. This subsection documents the **PRINT** and **PUNCH** commands and the labeling commands and discusses their use. The **PRINT** and **PUNCH** commands are identical in form and interpretation, so the **PRINT** command will be used to represent both commands in the following discussion. There are also many features and utilities available to the user to obtain output through modifications to the executive MAPOL sequence. These include direct use of MAPOL utilities, modification of print parameters in functional module calling sequences and user written procedures or modules. These output capabilities and a more complete discussion of the output processing (**PRINT** and **PUNCH**) capabilities of the ASTROS system is presented in Chapter 6 of this manual.

The **PRINT** and **PUNCH** commands have a number of options which can be separated into three groups: subset options, response quantity options and form options. The subset options select a set of subcases and/or design iterations to which the **PRINT** command applies while the remaining options select the actual data quantities that are desired; (e.g. stresses, strains, and displacements) and the form in which complex quantities are to be printed. The output selection can appear at any level of the solution control hierarchy and will apply at that level until it is overridden. When more than one discipline is covered by a print request at the boundary level, ASTROS will consider only the relevant print requests for each discipline. For example, if **STATICS** and **FLUTTER** are performed, the **STATICS** discipline will ignore any **ROOTS** requests and the flutter discipline will ignore any **STRESS** requests.

5.4.1. Subset Options

As indicated in the preceding subsections, some disciplines have more than one *subcase* per solution control statement. Others, like **STATICS** and **SAERO** have a separate solution control statement for each subcase. In all cases, disciplines within the **OPTIMIZE** subpacket may be analyzed at one or more design iterations. When one subcase is defined per statement, the user is free to modify the print requests from subcase to subcase; for example:

```
ANALYZE
   BOUNDARY SPC = 10
      STATICS ( MECH = 10 )
         PRINT STRESS = ALL, DISP = 100
      STATICS ( MECH = 20, GRAV = 100 )
         PRINT DISP = ALL
```

specifies that stresses for all elements and displacements for nodes listed in set 100 be printed for the first subcase (mechanical loads with set identification 10) and only the displacements be printed for the next load condition. When the discipline generates more than one subcase, however, the user ***must*** specify the subcases to which the **PRINT** request applies. For example:

```
ANALYZE
   BOUNDARY SPC=10, METHOD=1000
      MODES
      PRINT (MODES=ALL) DISP=100
```

selects the displacements (eigenvectors) for all the computed mode shapes be printed. If the **MODES=ALL** selection were not included in the **PRINT** statement, the user would get ***no*** output at all. The user is cautioned that the output processing in ASTROS is designed to limit output to those quantities that are explicitly selected and, therefore, the default for subcase option **MODES** is that no modes are selected. Whenever multiple subcases are generated by a discipline, as in the case of **MODES**, **TRANSIENT** and **FREQUENCY**, a subcase selection option is ***required*** on the **PRINT** command in order to get any output.

If the discipline appears in the **OPTIMIZE** subpacket, the user may request that the output appear only at certain iterations. For example:

```
OPTIMIZE
   BOUNDARY SPC=10, METHOD=1000
      MODES (DCON=1000)
      PRINT (ITER=10, MODES=ALL) DISP=100
```

selects the displacements (eigenvectors) for all the computed mode shapes be printed at the iterations given in **ITERLIST 10**. Unlike the other subset selectors, the default for **ITER** is **ALL**. Omission of the **ITER** selector therefore implies that the quantity will be printed at every iteration. This default is a consequence of compatibility with early versions of ASTROS in which there was no **ITERATION** selection at all.

The subset selections can be specified at two levels as parenthetical *phrases* attached to the print or punch statement. At the higher level, the subset options generate defaults for the entire print or punch statement. For example:

```
PRINT (ITER=10, TIME=ALL) STRESS=ALL, STRAIN=ALL
```

requests that all stresses and strains at all time steps for the iterations in **ITERLIST 10** be printed. In addition, the subset options can be attached to the individual quantity options to override the print default. For example:

```
PRINT (ITER=10, TIME=ALL) STRESS=ALL, STRAIN(TIME=10)=ALL
```

overrides the **TIME=ALL** default for the strain output. At both levels, the defaults are **NONE** for **TIME**, **FREQ** and **MODE** and **ALL** for **ITER**.

The subset options in ASTROS are:

OPTION	DESCRIPTION
FREQUENCY	Selects the frequency steps of frequency response disciplines at which output is desired by referencing a **FREQLIST** bulk data entry.
ITERATION	Selects the design iterations at which output is desired by referencing a **ITERLIST** bulk data entry.
MODE	Selects the eigenvectors of a normal modes discipline at which output is desired by referencing a **MODELIST** bulk data entry.
TIME	Selects the time steps of transient response analysis at which output is desired by referencing a **TIMELIST** bulk data entry.

5.4.2. Response Quantity Options

ASTROS is able to compute a number of response quantities for each discipline type. Each discipline type generates a different set of quantities so that the quantity selected by a particular keyword can sometimes change from one discipline to another. In addition, the available quantities are sometimes a function of the boundary condition type. For example, the flutter mode shape is not available as an output from a flutter analysis performed in the **OPTIMIZE** subpacket. This subsection will present the available quantities, the **PRINT** options which select them and the limitations (if any) on their availability. Table 5-4 summarizes the available **PRINT** and **PUNCH** response quantity options.

As in NASTRAN, stresses, strains and element forces are computed in the element coordinate system at predetermined or user selected points in the element. Nodal quantities are computed in the global coordinate system. **CGRA**, **DCON**, **GDES**, **KSNS**, **MODEL**, **MSNS**, **OGRA** and **HIST** are only applicable in the **OPTIMIZE** subpacket ***above*** the first **BOUNDARY** (since these requests transcend all analyses). The **DISP** option for flutter analyses is only applicable in the **ANALYZE** subpacket. Other options are available independent of the boundary condition type. Table 5-5 presents a matrix of response quantity options for each discipline type, showing the applicability of each option. Any requests for quantities that do not apply to the particular discipline will be ignored by the output processor without warning.

Most options can be **ALL, NONE** or an integer value which selects bulk data entry sets listing the items for which the response quantity is desired. For example, the **STRESS** option points to the **ELEMLIST** bulk data entity which lists the elements for which stresses are desired. The **NONE** option is used to override a default established through a print or punch request at a higher level in the hierarchy. The ASTROS output philosophy is similar to that of NASTRAN in that it is assumed that mistakes in the output requests should not terminate execution. If, for example, the requested structural element does not exist in the model, the output request will be ignored without any warning to the user. Other output request errors in ASTROS are treated in a similar manner, occasionally generating a warning message, but more typically resulting in no visible indication that the request was in error. Therefore the user can, in most cases, request output that does not apply to the discipline, for entities (nodes or elements) which do not exist and/or for subcases that are not defined without causing termination of the execution.

5.4.3. Form Options

For complex response quantities, the form option is provided to select either **RECTANGULAR** or **POLAR** form. Rectangular form gives the cartesian components of the quantity in the rectangular complex plane in which the first number represents the real component and the second number the imaginary component. Polar form gives the components in polar coordinates in which the first number represents the radial distance from the origin (the magnitude) and the second represents the angular displacement from the real coordinate axis (the phase angle). The phase angle is computed in degrees.

The form can be specified at two levels as parenthetical *phrases* attached to the print or punch statement. At the higher level, the form option generates a default for the entire print or punch statement. For example:

```
PRINT (POLAR) STRESS=ALL, STRAIN=ALL
```

requests that polar form be used for both stress and strain response quantities. In addition, the form option can be attached to the individual quantity options to override the print default. For example:

```
PRINT (POLAR) STRESS=ALL, STRAIN(RECT)=ALL
```

overrides the polar default for the strain output. At both levels, the default form is rectangular and any polar requests for real output quantities are ignored.

5.4.4. Labeling Options

Labeling of printed output is performed through the use of three optional commands identical in form to their NASTRAN counterparts:

OPTION	DESCRIPTION
TITLE	A title header that will appear as the first line on each page of output.
SUBTITLE	A secondary header that will appear on the second line of each page of output.
LABEL	A tertiary header that is typically used to identify subcase (discipline level) output.

Each of these commands can appear at any level in the solution control hierarchy and will be applied until superseded.

5.5. SOLUTION CONTROL COMMANDS

The ASTROS Solution Control Commands are described in this section.

Table 5-4. Response Quantity Output Options

OPTION	DESCRIPTION
ACCELERATION	Selects accelerations at nodal points.
AIRDISPLACEMENT	Selects displacements on aerodynamic boxes.
CGRADIENT	Selects gradients of active constraints.
DCONSTRAINT	Selects active constraints at each iteration.
DISPLACEMENTS	Selects displacements at nodal points.
ENERGY	Selects strain energy at structural elements.
FORCE	Selects element forces at structural elements.
GDESIGN	Selects global design variables.
GPFORCE	Selects grid point forces at nodal points.
GPWG	Selects print of grid point weight summary.
KSNS	Selects stiffness sensitivities at design variables.
LDESIGN	Selects local design variables.
LOAD	Selects applied loads at nodal points.
MASS	Selects mass matrix at nodal points.
MODEL	Selects Bulk Data at current design point. (**PUNCH** only)
MSNS	Selects mass sensitivities at design variables.
OGRADIENT	Selects gradient of the objective function.
QHH	Selects QHH generalized unsteady aerodynamic forces at modes.
QHJ	Selects QHJ generalized unsteady aerodynamic forces at modes.
ROOT	Selects flutter and normal modes roots (eigenvalues).
SPCFORCE	Selects forces of single point constraint at nodal points.
STIFFNESS	Selects stiffness matrix at nodal points.
STRAIN	Selects strains at structural elements.
STRESS	Selects stresses at structural elements.
TPRESSURE	Selects trim pressures at aerodynamic boxes.
TRIM	Selects trim and stability coefficients for steady aeroelastic analyses.
VELOCITY	Selects velocities at nodal points.

Table 5-5. Response Quantities by Discipline

OPTION	DESIGN	STAT	MODE	SAERO	FLUT	TRANS	FREQ
ACCEL		✓		✓		✓	✓
AIRDISP				✓			
CGRADIENT	✓						
DCONSTRAINT	✓						
DISP		✓	✓	✓	✓	✓	✓
ENERGY		✓	✓	✓		✓	
FORCE		✓	✓	✓		✓	
GDESIGN	✓						
GPFORCE		✓		✓			
GPWG		✓	✓	✓	✓	✓	✓
KSNS	✓						
LDESIGN	✓						
LOAD		✓		✓		✓	✓
MASS		✓	✓	✓	✓	✓	✓
MODEL	✓						
MSNS	✓						
OGRADIENT	✓						
QHH					✓		✓
QHJ							✓
ROOT			✓		✓		
SPFORCE		✓		✓			
STIFFNESS		✓	✓	✓	✓	✓	✓
STRAIN		✓	✓	✓		✓	
STRESS		✓	✓	✓		✓	
TPRESSURE				✓			
TRIM				✓			
VELO						✓	✓

This page is intentionally blank.

Solution Control Command: **$**

Description: Allows commentary data to be placed in the Solution Control packet.

Hierarchy Level: Various

Format:

```
$ THIS IS A COMMENT
```

Solution Control Command: **ANALYZE**

Description: The first command in the ANALYZE subpacket

Hierarchy Level: Type of run

Format:

```
ANALYZE
```

Solution Control Command: **BOUNDARY**

Description: Specifies the displacement sets and related data used in a particular boundary condition.

Hierarchy Level: Boundary condition

Format and Examples:

```
BOUNDARY [BCID = bcid,] MPC = i, SPC = j, REDUCE = k, SUPPORT = l,
         METHOD = m, CMETHOD = t,
         AUTOSPC [ ( [ PRINT / NOPRINT ] [,PUNCH USING s] [,EPS = x] ) ] [ = YES / NO ]
         DYNRED = n, INERTIA = o, ESET = p, K2GG = q, M2GG = r
BOUNDARY SPC = 6
BOUNDARY SPC = 10, REDUCE = 20, SUPPORT = 30
BOUNDARY SPC = 12, DYNRED = 100, INERTIA = 100, K2GG = FUSSTIFF
BOUNDARY AUTOSPC(NOPRINT, PUNCH USING 1001) = YES, SUPPORT = 101
```

Option	Meaning
bcid	Boundary condition identification number. (Integer>0)
i	Set identification of a multipoint constraint set. Invokes **MPC** and **MPCADD** bulk data entries. (Integer>0)
j	Set identification of a single point constraint set. Invokes **SPC**, **SPC1** and **SPCADD** bulk data entries. (Integer>0)
k	Set identification of a static condensation set. Invokes **ASET**, **ASET1**, **OMIT** and **OMIT1** bulk data entries. (Integer>0)
l	Set identification of the free body support. Invokes **SUPORT** bulk data entries. (Integer>0)
m	Set identification of the **EIGR** bulk data entry to be used. (Integer>0)
n	Selects the dynamic reduction parameters from the **DYNRED** bulk data entry (Integer>0)
o	Selects the **JSETi** bulk data entries identifying inertia relief degrees of freedom for performing dynamic reduction (Integer>0)
p	Set identification of the extra degrees of freedom for the boundary condition. Invokes **EPOINT** bulk data entries. (Integer>0)
q	Selects the direct input stiffness matrix in the g-set. This matrix will be added to **KGG** for this boundary condition. Refers to a **DMI** or **DMIG** Bulk Data entry.

r Selects the direct input mass matrix in the g-set. This matrix will be added to **MGG** for this boundary condition. Refers to a **DMI** or **DMIG** Bulk Data entry.

s Specifies a set identification number to be used for punching the **SPC** Bulk Data entries generated by the **AUTOSPC** option. (Integer>0, less than 9 digits)

t Defines a default **EIGC** set identification to be used by the **CEIG** module if it is passed a zero value in its call sequence.

x Defines the **AUTOSPC** threshhold. Singularities with values less than x are automatically constrained. (Real, Default=10^{-8})

Remarks:

1. If any **BOUNDARY** has a **bcid**, then all boundaries must have a **bcid**. All **bcid** values must be unique, but they need not be in any particular order. Boundaries are implicitly numbered from 1 to n if no **bcid** values are specified. The **bcid** is only used as a reference from user defined functions in the Function Packet.
2. Note that the **REDUCE** and **ESET** set specifications are innovative relative to NASTRAN.
3. The bulk data entries will not be used in ASTROS unless selected in Solution Control.
4. None of the options are required but at least one must appear.
5. **K2GG** and **M2GG** affect the system stiffness and mass matrices, respectively, for all disciplines within the boundary condition.
6. **K2GG** and **M2GG** names will typically refer to **DMI** or **DMIG** entries but may refer to any data base matrix entity of the proper dimension.
7. The **AUTOSPC** command:

   AUTOSPC(PRINT,EPS=1.0E-8) = YES

   is the default value. To disable the feature, use:

   AUTOSPC = NO

Solution Control Command: **END**

Description: Indicates the end of a subpacket.

Hierarchy Level: End

Format:

```
END
```

Remarks:

1. The **ANALYZE** and **OPTIMIZE** subpackets each require an **END** command.

Solution Control Command: **FLUTTER**

Description: Invokes the flutter analysis discipline

Hierarchy Level: Discipline

Format and Examples:

```
FLUTTER [caseid] (FLCOND = i, DCONSTRAINT = j, DCFUNCTION = q, CONTROL = k,
        K2PP = l, M2PP = m, B2PP = n , TFL = o, DAMPING = p,
        DCFUNCTION = q)

FLUTTER (FLCOND = 100)

FLUTTER (FLCOND = 100, CONTROL = AILERON, K2PP = KAIL, TFL = 5)
```

Option	Meaning
caseid	Case identification number. (Integer>0)
i	Set identification of a **FLUTTER** bulk data entry that provides flutter parameters.
j	Set identification of a **DCONFLT** bulk data entry that defines flutter constraint conditions.
k	Selects the input matrix for splining the extra points to the aerodynamic model. Refers to a **DMI** bulk data entry.
l	Selects the direct input stiffness matrix. Refers to a **DMI** or **DMIG** bulk data entry.
m	Selects the direct input mass matrix. Refers to a **DMI** or **DMIG** bulk data entry.
n	Selects the direct input damping matrix. Refers to a **DMI** or **DMIG** bulk data entry.
o	Selects the transfer function set to be added to the input matrices. Refers to **TF** bulk data entries.
p	Set identification of **VSDAMP** and/or **TABDMP** bulk data entries that define damping data.
q	Set identification of **DCONF** constraint functions.

Remarks:

1. If any discipline has a **caseid**, then all disciplines must have a **caseid**. All **caseid** values must be unique, but they need not be in any particular order. Disciplines are implicitly numbered from 1 to n if no **caseid** values are specified. The **caseid** is only used as a reference from user defined functions in the Function Packet.
2. The **FLCOND** option is required, all others are optional.
3. **M2PP**, **B2PP** and **K2PP** and **CONTROL** names will typically refer to **DMI** and **DMIG** entries, but may refer to any existing database entity of the proper dimension.
4. The use of the **CONTROL** matrix requires that extra points be defined in the boundary condition.

Solution Control Command: **FREQUENCY**

Description: Invokes the frequency response analysis discipline

Hierarchy Level: Discipline

Format and Examples:

```
FREQUENCY type [caseid] (DLOAD = i, FSTEP = j, GUST = k, K2PP = l,
          M2PP = m,B2PP = n, TFL = o, DAMPING = p)
FREQUENCY DIRECT (DLOAD = 10, FSTEP = 20)
FREQUENCY MODAL (DLOAD = 100, FSTEP = 30, M2PP = MFREQ, TFL = 5)
FREQUENCY DIRECT (DLOAD = 100, FSTEP = 20 , GUST = 55)
```

Option	Meaning
type	Selects the solution approach from **DIRECT** or **MODAL**.
caseid	Case identification number. (Integer>0)
i	Set identification of a **DLOAD** bulk data entry.
j	Set identification of frequency bulk data entries (**FREQ**, **FREQ1**, or **FREQ2**) that define the frequency steps for the analysis.
k	Set identification of a **GUST** bulk data entry which defines the gust parameters.
l	Selects the direct input stiffness matrix. Refers to a **DMI** or **DMIG** bulk data entry.
m	Selects the direct input mass matrix. Refers to a **DMI** or **DMIG** bulk data entry.
n	Selects the direct input damping matrix. Refers to a **DMI** or **DMIG** bulk data entry.
o	Selects the transfer function set to be added to the input matrices. Refers to **TF** bulk data entries.
p	Set identification of **VSDAMP** and/or **TABDMP** bulk data entries that define damping data.

Remarks:

1. If any discipline has a **caseid**, then all disciplines must have a **caseid**. All **caseid** values must be unique, but they need not be in any particular order. Disciplines are implicitly numbered from 1 to n if no **caseid** values are specified. The **caseid** is only used as a reference from user defined functions in the Function Packet.
2. The **FREQUENCY** discipline does not generate design constraints for optimization.
3. **type, DLOAD** and **FSTEP** are required.
4. No more than one **FREQUENCY** analysis can be done in a single boundary condition.
5. **M2PP**, **B2PP** and **K2PP** names will typically refer to **DMI** and **DMIG** entries, but may refer to any existing database entity of the proper dimension.

Solution Control Command: **K6ROT**

Description: Provides a stiffness value for in-plane stiffnesses for plate elements.

Hierarchy Level: Initial level (above **ANALYZE/OPTIMIZE**)

Format and Examples:

```
K6ROT = val
K6ROT = 1.0
K6ROT = 10.0E3
```

Option	Meaning
`val`	Real value used to compute the stiffness associated with the in-plane rotations of plate elements (Default **K6ROT** = 0.0, **K6ROT** ≥ 0.0)

Solution Control Command: **LABEL**

Description: Provides identifying information on subcase output.

Hierarchy Level: Label information

Format and Examples:

```
LABEL = n
LABEL = SYMMETRIC MANEUVER LOAD
```

Option	Meaning
n	Any descriptive message that the user wishes to use to distinquish output.

Remarks:

1. **LABEL** information is used until it is superseded.
2. The **LABEL** command is optional.
3. Labels are limited to no more than 72 characters.

Solution Control Command: **MODES**

Description: Selects the Normal Modes discipline.

Hierarchy Level: Discipline

Format and Examples:

```
MODES [caseid] (DCONS = n, DCFUNCTION = o)
MODES
MODES (DCONS = 10)
```

Option	Meaning
caseid	Case identification number. (Integer>0)
n	Set identification of **DCONFRQ** bulk data entries which define frequency constraints for the optimization task.
o	Set identification of **DCONF** constraint functions.

Remarks:

1. If any discipline has a **caseid**, then all disciplines must have a **caseid**. All **caseid** values must be unique, but they need not be in any particular order. Disciplines are implicitly numbered from 1 to n if no **caseid** values are specified. The **caseid** is only used as a reference from user defined functions in the Function Packet.
2. Only one modal analysis can be performed in a boundary condition using the **EIGR** bulk data entry selected on the **BOUNDARY** command.

Solution Control Command: **OPTIMIZE**

Description: Invokes the ASTROS design capability

Hierarchy Level: Type of boundary condition

Format and Examples:

```
{ OPTIMIZE  }
{ MINIMIZE  } STRATEGY = ((m1,niter1),(m2,niter2),(m3,niter3)), MAXITER = n,
{ MAXIMIZE  }
                         MOVLIM = o, WINDOW = p, ALPHA = r,
                         CNVRGLIM = s, NRFAC = t, EPS = u, FDSTEP = v,
                         FDSTEP = v, OBJECTIVE = [WEIGHT, w], DCFUNCTION = x
```

```
OPTIMIZE
OPTIMIZE MAXITER = 10, NRFAC = 0.6, EPS = -.05, MOVLIM = 1.3
OPTIMIZE STRATEGY = FSD, ALPHA = 0.8, MAXFSD = 10
OPTIMIZE STRATEGY = (FSD, 3), ALPHA = 0.8, MAXFSD = 10
```

Option	Meaning
m1,m2,m3	The strategy to be used in optimization. Either **MP** for math programming methods or **FSD** for fully stressed design. The order of input on the strategy command is the order that will be used. Each strategy, **MP** or **FSD**, may only appear once. Default for **m1=MP**. (Only **MP** methods will be used)
niter1, niter2, niter3	The number of iterations for **m1**, **m2** and **m3** respectively. The default for each is to use the last named method for those iterations remaining up to **MAXITER**. If **MAXITER** is less than the sum of specified iterations, ASTROS will warn the user but stop at **MAXITER** iterations.

STRATEGY	**MP** for iterations 1 thru **MAXITER**
STRATEGY = MP	**MP** for iterations 1 thru **MAXITER**
STRATEGY = FSD,5	**FSD** for iterations 1 thru 5 **MP** for iterations 6 thru **MAXITER**
STRATEGY = ((FSD,5),MP)	**FSD** for iterations 1 thru 5 **MP** for iterations 6 thru **MAXITER**

n	The maximum number of iterations to be performed using **MP**or **FSD**. Default = 15.
o	The move limit applied to local design variables in **MP**. The local variable after each redesign will lie between **t/MOVLIM** and **t*MOVLIM** where **t** is the initial value. Default = 2.0, must be greater than 1.0.
p	The window around zero in which the **MOVLIM** bound is overridden to allow the local variable to change sign. If **WINDOW=0.0**, the local variable may not change sign. If **WINDOW** is nonzero, the half width of a band around zero, **EPS** is computed

```
EPS = WINDOW/100 * MAX ( ABS(TMIN), ABS(TMAX) )
```

If the local variable falls within the band, the new minimum or maximum for the current iteration is changed to lie on the other side of zero from the local variable. The bandwidth **EPS** is a percentage of the larger of **TMAX** or **TMIN** where **WINDOW** specifies the percentage. Default = 0.0, must be greater than or equal to 0.0.

r Exponential move limit for **FSD**. Numbers less than 1.0 result in a smaller move with smoother convergence. Ignored if **STRAT=MP**, Default = 0.90, must be greater than 0.0

s Convergence limit specifying the maximum percentage change in the objective function that can be considered converged. Default = 1.0, must be greater than 0.0.

t Constraint retention factor for **MP** methods. The number of active constraints will be at least **NRFAC** times the number of design variables. Default = 3.0.

u Constraint retention parameter in which all constraints having a value greater than **EPS** will be considered active. Default = -0.10

v Finite difference step size for nonlinear design variables. The relative design variable increment $\Delta v = \begin{cases} \textbf{FDSTEP} \cdot v \ ; \ v \neq 0.0 \\ \textbf{FDSTEP} \ ; \ v = 0.0 \end{cases}$ for finite difference computation. Default = 0.001 must be greater than zero.

w Objective function selected from **WEIGHT**, the default value, or the set identification of a single scalar **DCONF** function.

x Identification number of **DCONF** Bulk Data entries defining subcase independent functions.

Remarks:

1. None of the options are required.
2. **MAXITER** and **CNVRGLIM** are global parameters that apply to the **MP** and **FSD** strategies.
3. **MOVLIM** and **WINDOW** control the move limits for **MP**. **WINDOW** is only useful for **LOCAL** design variables that need to cross between positive and negative values.
4. **NRFAC** and **EPS** control the constraint deletion algorithm for **MP**, both values are always applied.

Solution Control Command: **PRINT**

Description: Specifies the required output file processing for the print file or CADDB database.

Hierarchy Level: Various

Format and Examples:

```
PRINT       (Form, FREQ = a, ITER = b, MODE = c, TIME = d)
            ACCE (Form, FREQ = a, ITER = b, TIME = d) = e,
            AIRD (Form, ITER = b, MODE = c, TIME = d) = f,
            BUCK (ITER = b) = ag
            CGRA (ITER = b) = h,
            DCON (ITER = b) = i,
            DISP (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = j,
            ENER (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = k,
            FORC (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = l,
            GDES (ITER = b) = m,
            GPFO (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = n,
            GPWG (ITER = b) = n1,
            KSNS (ITER = b) = o,
            LDES (ITER = b) = p,
            LOAD (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = q,
            MASS (ITER = b) = r,
            MSNS (ITER = b) = t,
            OGRA (ITER = b) = u,
            QHH  (ITER = b, MODE = c) = x,
            QHJ  (ITER = b, MODE = c) = y,
            ROOT (Form, ITER = b, MODE = c) = z,
            SPCF (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = aa,
            STIF (ITER = b) = ab,
            STRA (Form, FREQ = a, ITER = b, MODE = c, TIME = d, ah) = ac,
            STRE (Form, FREQ = a, ITER = b, MODE = c, TIME = d, ah) = ad,
            TPRE (ITER = b) = ae,
            VELO (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = af
            TRIM

PRINT DISP = ALL

PRINT (RECT, MODE = 10, ITER = 20) DISP(ITER = LAST) = 6, ENERGY(POLA) = 10

PRINT (MODE = NONE)
```

Options	Meaning
Form	**RECT** or **POLA** requests output in **RECT**angular or **POLA**r format (See Remarks 1 and 2).
a	Set identification of a **FREQLIST** bulk data entry that is used to request the frequencies at which output is to be printed (See Remark 2).
b	Set identification of an **ITERLIST** bulk data entry that is used to request the optimization iterations at which output is to be printed (See Remark 2).

c	Set identification of a **MODELIST** bulk data entry that is used to request the modes at which output is to be printed (See Remark 2).
d	Set identification of a **TIMELIST** bulk data entry that is used to request the times at which output is to be printed (See Remark 2).
e	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which accelerations are to be printed.
f	Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic box elements at which displacements for the aerodynamic model are to be printed.
h	Set identification of an **DCONLIST** bulk data entry that is used to request the the subset of active constraints for which gradients are to be printed (See Remark 2).
i	Set identification of an **DCONLIST** bulk data entry that is used to request the the subset of active constraints which are to be printed (See Remark 2).
j	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which displacements are to be printed.
k	Set identification of an **ELEMLIST** bulk data entry that is used to request the elements for which strain energies are to be printed.
l	Set identification of an **ELEMLIST** bulk data entry that is used to request the elements for which forces are to be printed.
m	Set identification of a **GDVLIST** bulk data entry that is used to request the global design variable IDs for which global design variables are to be printed.
n	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which grid point forces are to be printed.
n1	Either **ALL** or **NONE** depending on whether the **GPWG** is to be computed/printed. If a **GPWG** entry is in the Bulk Data file, it will be used by the algorithm.
o	Set identification of an **LDVLIST** and/or a **GDVLIST** bulk data entry that is used to request the design variables for which stiffness sensitivities are to printed.
p	Set identification of an **LDVLIST** bulk data entry that is used to request the local design variable IDs for which local design variables are to be printed.
q	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which applied loads are to be printed.
r	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points degrees of freedom for which the mass matrix is to be printed.
t	Set identification of an **LDVLIST** and/or a **GDVLIST** bulk data entry that is used to request the design variables for which mass sensitivities are to printed.
u	Set identification of a **GDVLIST** bulk data entry that is used to request the design variables for which objective function gradients are to be printed.
x	Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic elements for which **QHH** is to be printed.
y	Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic elements for which **QHJ** is to be printed.
z	Set identification of an **MODELIST** bulk data entry that is used to request the modes for which flutter and normal modes eigenvalue results are to be printed.

aa Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which **SPC** forces are to be printed.

ab Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points degrees of freedom for which the stiffness matrix is to be printed.

ac Set identification of an **ELEMLIST** bulk data entry that is used to request the elements at which strains are to be printed.

ad Set identification of an **ELEMLIST** bulk data entry that is used to request the elements for which stresses are be printed.

ae Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic elements for which the pressure coefficients at aeroelastic trim are to be printed.

af Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which velocities are to be printed.

ag Specifies the elements for which local buckling results are to be printed, may be **ALL** or **NONE**.

ah Selects the type of stresses or strains to be output for composite elements. The options are: **LAYER**, **LAMINATE** or **BOTH**. The default is **LAYER**.

Remarks:

1. **Form** is an optional parameter for printing complex data. **RECT**angular data outputs complex data with real and imaginary components while **POLA**r outputs complex data using magnitude and phase.
2. If used with the **PRINT** command, all data that are not otherwise specified use the requested **Form**, **FREQ**, **ITER**, **MODE**, and **TIME**, if applicable for that type of data. If used with an option, **Form**, **FREQ**, **ITER**, **MODE**, and **TIME** override the global request. Options a through af can be either **ALL**, **NONE**, or a positive integer, and additionally, option **b** (**ITER**) can be **LAST**, and options **h** (**CGRA**) and **i** (**DCON**) can be **ACTIVE**. **ALL** requests all values. **NONE** turns off a request from a previous hierarchy while an integer value refers to a bulk data entry. **LAST** requests that output be printed for only the final value in a list. For example, **ITER**=**LAST** selects output for the final iteration in an optimization. **ACTIVE** selects the active constraints.
3. **HIST** and **TRIM** are toggles. If they are present, the specified data are printed. **TRIM** indicates that stability derivative data associated with an aeroelastic trim are to be printed. **HIST** indicates that the design iteration history summary is to be printed.
4. Aerodynamic macro elements are selected indirectly. A macro element is chosen by selecting one or more aerodynamic box elements contained within the macro element.
5. See Table 47 for a summary of how the items are printed or written to the CADDB database.

Solution Control Command: **PUNCH**

Description: Specifies the required output file processing for the punch file

Hierarchy Level: Various

Format and Examples:

```
PUNCH       (Form, FREQ = a, ITER = b, MODE = c, TIME = d)
            ACCE (Form, FREQ = a, ITER = b, TIME = d) = e,
            AIRD (Form, ITER = b, MODE = c, TIME = d) = f,
            BUCK (ITER = b) = ah,
            CGRA (ITER = b) = h,
            DCON (ITER = b) = i,
            DISP (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = j,
            ENER (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = k,
            FORC (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = l,
            GDES (ITER = b) = m,
            GPFO (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = n,
            GPWG (ITER = b) = n1,
            KSNS (ITER = b) = o,
            LDES (ITER = b) = p,
            LOAD (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = q,
            MASS (ITER = b) = r,
            MODEL (ITER = ag) = ah
            MSNS (ITER = b) = t,
            OGRA (ITER = b) = u,
            QHH  (ITER = b,MODE=c) = x,
            QHJ  (ITER = b,MODE=c) = y,
            ROOT (Form, ITER = b, MODE = c) = z,
            SPCF (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = aa,
            STIF (ITER = b) = ab,
            STRA (Form, FREQ = a, ITER = b, MODE = c, TIME = d, aj) = ac,
            STRE (Form, FREQ = a, ITER = b, MODE = c, TIME = d, aj) = ad,
            TPRE (ITER = b) = ae,
            VELO (Form, FREQ = a, ITER = b, MODE = c, TIME = d) = af
            TRIM

PUNCH DISP = ALL

PUNCH (RECT, MODE = 10, ITER = 20) DISP(ITER = LAST) = 6, ENERGY(POLA) = 10

PUNCH (MODE=NONE)
```

Options	Meaning
Form	**RECT** or **POLA** requests output in **RECT**angular or **POLA**r format (See Remarks 1 and 2).
a	Set identification of a **FREQLIST** bulk data entry that is used to request the frequencies at which output is to be punched (See Remark 2).
b	Set identification of an **ITERLIST** bulk data entry that is used to request the optimization iterations at which output is to be punched (See Remark 2).

c	Set identification of a **MODELIST** bulk data entry that is used to request the modes at which output is to be punched (See Remark 2).
d	Set identification of a **TIMELIST** bulk data entry that is used to request the times at which output is to be punched (See Remark 2).
e	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which accelerations are to be punched.
f	Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic box elements at which displacements for the aerodynamic model are to be punched.
h	Set identification of an **DCONLIST** bulk data entry that is used to request the the subset of active constraints for which gradients are to be punched (See Remark 2).
i	Set identification of an **DCONLIST** bulk data entry that is used to request the the subset of active constraints which are to be punched (See Remark 2).
j	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which displacements are to be punched.
k	Set identification of an **ELEMLIST** bulk data entry that is used to request the elements for which strain energies are to be punched.
l	Set identification of an **ELEMLIST** bulk data entry that is used to request the elements for which forces are to be punched.
m	Set identification of a **GDVLIST** bulk data entry that is used to request the global design variable IDs for which global design variables are to be punched.
n	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which grid point forces are to be punched.
n1	Either **ALL** or **NONE** depending on whether the **GPWG** is to be computed/punched. If a **GPWG** entry is in the Bulk Data file, it will be used by the algorithm.
o	Set identification of an **LDVLIST** and/or a **GDVLIST** bulk data entry that is used to request the design variables for which stiffness sensitivities are to punched.
p	Set identification of an **LDVLIST** bulk data entry that is used to request the local design variable IDs for which local design variables are to be punched.
q	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which applied loads are to be punched.
r	Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points degrees of freedom for which the mass matrix is to be punched.
t	Set identification of an **LDVLIST** and/or a **GDVLIST** bulk data entry that is used to request the design variables for which mass sensitivities are to punched.
u	Set identification of a **GDVLIST** bulk data entry that is used to request the design variables for which objective function gradients are to be punched.
x	Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic elements for which **QHH** is to be punched.
y	Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic elements for which **QHJ** is to be punched.
z	Set identification of an **MODELIST** bulk data entry that is used to request the modes for which flutter and normal modes eigenvalue results are to be punched.

aa Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which **SPC** forces are to be punched.

ab Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points degrees of freedom for which the stiffness matrix is to be punched.

ac Set identification of an **ELEMLIST** bulk data entry that is used to request the elements at which strains are to be punched.

ad Set identification of an **ELEMLIST** bulk data entry that is used to request the elements for which stresses are be punched.

ae Set identification of an **ELEMLIST** bulk data entry that is used to request the aerodynamic elements for which the pressure coefficients at aeroelastic trim are to be punched.

af Set identification of a **GRIDLIST** bulk data entry that is used to request the grid points at which velocities are to be punched.

ag Specifies the iterations at which the design model will be punched. May be **ALL**, **NONE**, **LAST**, or the set identification of an **ITERLIST** bulk data entry which specifies the iterations at which to punch the model.

ah Specifies the portion of the model which will be punched. May be **ALL** or **NONE**. (Note: an integer value is accepted and treated as **ALL**)

ai Specifies the elements for which local buckling results are to be punched, may be **ALL** or **NONE**.

aj Selects the type of stresses or strains to be output for composite elements. The options are: **LAYER**, **LAMINATE** or **BOTH**. The default is **LAYER**.

Remarks:

1. **Form** is an optional parameter for printing complex data. **RECT**angular data outputs complex data with real and imaginary components while **POLA**r outputs complex data using magnitude and phase.
2. If used with the **PRINT** command, all data that are not otherwise specified use the requested **Form**, **FREQ**, **ITER**, **MODE**, and **TIME**, if applicable for that type of data. If used with an option, **Form**, **FREQ**, **ITER**, **MODE**, and **TIME** override the global request. Options a through af can be either **ALL**, **NONE**, or a positive integer, and additionally, option **b** (**ITER**) can be **LAST**, and options **h** (**CGRA**) and **i** (**DCON**) can be **ACTIVE**. **ALL** requests all values. **NONE** turns off a request from a previous hierarchy while an integer value refers to a bulk data entry. **LAST** requests that output be printed for only the final value in a list. For example, **ITER**=**LAST** selects output for the final iteration in an optimization. **ACTIVE** selects the active constraints.
3. **HIST** and **TRIM** are toggles. If they are present, the specified data are punched. **TRIM** indicates that stability derivative data associated with an aeroelastic trim are to be punched. **HIST** indicates that the design iteration history summary is to be punched.
4. Aerodynamic macro elements are selected indirectly. A macro element is chosen by selecting one or more aerodynamic box elements contained within the macro element.
5. See Table 47 for a summary of how the items are punched or written to the CADDB database.

Solution Control Command: **SAERO**

Description: Invokes the static aerodynamics discipline

Hierarchy Level: Discipline

Format and Examples:

```
SAERO [caseid] [symtype] ( TRIM = k ,DCON = o, STRESS = m,
      STRAIN = n, DCFUNCTION = p)

SAERO (TRIM = 60)

SAERO ANTISYMMETRIC (TRIM = 70, STRESS = 100)
```

Option	Meaning
caseid	Case identification number. (Integer>0)
symtype	Selects the symmetry type for the subcase from **SYMMETRIC** or **ANTISYMMETRIC**. (Default is **SYMMETRIC**)
k	Set identification of a **TRIM** bulk data entry which provides flight condition information.
m	Set identification for stress constraints as defined by **DCONVM**, **DCONVMM**, **DCONVMP**, **DCONTW**, **DCONTWM**, or **DCONTWP** bulk data entries.
n	Set identification for strain constraints as defined by **DCONEP**, **DCONEPM**, **DCONEPP**, **DCONFT**, **DCONFTM**, or **DCONFTP** bulk data entries.
o	Set identification for displacement constraints as defined by **DCONDSP**, **DCONTRM**, **DCONCLA**, **DCONALE**, or **DCONSCF** bulk data entries.
p	Set identification of **DCONF** constraint functions.

Remarks:

1. If any discipline has a **caseid**, then all disciplines must have a **caseid**. All **caseid** values must be unique, but they need not be in any particular order. Disciplines are implicitly numbered from 1 to n if no **caseid** values are specified. The **caseid** is only used as a reference from user defined functions in the Function Packet.
2. **TRIM** is required. Both symtyp and the **CONSTRAINT** section are optional.
3. **SAERO** disciplines may be freely combined with other ASTROS disciplines.
4. For compatibility, the alternate form of constraint specification shown below is also allowed. Its use is, however, discouraged.

   ```
   SAERO [ symtype ] ( TRIM = k ),CONSTRAINT(STRESS=m,STRAIN=n,GENERAL=o)
   ```

Solution Control Command: **SOLUTION**

Description: The first command in the solution control packet.

Hierarchy Level: Beginning of solution

Format:

```
SOLUTION
```

Remarks:

1. One **SOLUTION** command must always appear as the first command of the solution control packet.

Solution Control Command: **STATICS**

Description: Invokes the statics analysis discipline

Hierarchy level: Discipline

Format and Examples:

```
STATICS [caseid] (MECH = i, THERMAL = j, GRAVITY = k, DCON = o,
          STRESS = m, STRAIN = n, DCFUNCTION = p)

STATICS (MECH = 10)

STATICS (MECH = 4, THERMAL = 6, DCFUNCTION = 10)
```

Option	Meaning
caseid	Case identification number. (Integer>0)
i	Set identification for external loads as defined by **LOAD**, **PLOAD**, **FORCE**, **FORCE1**, **MOMENT**, and **MOMENT1** bulk data entries.
j	Set identification for temperatures defined by **TEMP** or **TEMPD** bulk data entries.
k	Set identificaton of **GRAV** bulk data entries which define gravity forces.
m	Set identification for stress constraints defined by **DCONVM**, **DCONVMM**, **DCONVMP**, **DCONTW**, **DCONTWM**, or **DCONTWP** bulk data entries.
n	Set identification for strain constraints defined by **DCONEP**, **DCONEPM**, **DCONEPP**, **DCONFT**, **DCONFTM**, or **DCONFTP** bulk data entries.
o	Set identification of **DCONDSP** bulk data entries which define displacement constraints.
p	Set identification of **DCONF** constraint functions.

Remarks.

1. If any discipline has a **caseid**, then all disciplines must have a **caseid**. All **caseid** values must be unique, but they need not be in any particular order. Disciplines are implicitly numbered from 1 to n if no **caseid** values are specified. The **caseid** is only used as a reference from user defined functions in the Function Packet.
2. The sum of all the loads forms a single right hand side for a statics analysis.
3. At least one of the load types must be present. The **CONSTRAINT** section is optional.
4. Gravity forces may be included indirectly if referenced by the **LOAD** bulk data entry.
5. For compatibility, the alternate form of constraint specification shown below is also allowed. Its use is, however, discouraged.

```
STATICS (MECH = i, THERMAL = j, GRAVITY = k),
     CONSTRAINT(STRESS=m,STRAIN=n,GENERAL=o)
```

Solution Control Command: **SUBTITLE**

Description: Defines a subtitle which will appear in the output.

Hierarchy Level: Label information

Format and Example:

```
SUBTITLE = n
SUBTITLE = SUPERSONIC DESIGN CONDITION
```

Option	Meaning
n	Any descriptive information can be inserted here

Remarks:

1. **SUBTITLE** information is used until it is superseded.
2. The **SUBTITLE** command is optional.
3. Subtitles are limited to 72 characters.

Solution Control Command: **TITLE**

Description: Defines a title which will appear in the output.

Hierarchy Level: Label information

Format and Examples:

```
TITLE = n
TITLE = DESIGN OF A FORWARD SWEPT WING MODEL
```

Option	Meaning
n	Any descriptive information can be inserted here

Remarks:

1. **TITLE** information is used until it is superseded.
2. The **TITLE** command is optional.
3. Titles are limited to no more that 72 characters.

Solution Control Command: **TRANSIENT**

Description: Invokes the transient analysis discipline

Hierarchy Level: Discipline

Format and Examples:

```
TRANSIENT type [caseid] (DLOAD = i, TSTEP = j, FFT = k, IC = l,
          GUST = m, K2PP = n, M2PP = o, B2PP = p, TFL = q, DAMPING = r )

TRANSIENT MODAL (DLOAD = 10, TSTEP = 20)

TRANSIENT DIRECT (DLOAD = 100, TSTEP = 30, K2PP = KFREQ, IC = 45, TFL = 5)

TRANSIENT MODAL (DLOAD = 100, TSTEP = 20 , FFT = 999, GUST = 55)
```

Option	Meaning
caseid	Case identification number. (Integer>0)
type	Selects the solution approach from **DIRECT** or **MODAL**.
i	Set identification of a **DLOAD** bulk data entry.
j	Set identification of **TSTEP** bulk data entries which provide the time step information for the analysis.
k	Set identification of an **FFT** bulk data entry which provides parameters to use the Fast Fourier Transform methods in performing the transient analysis.
l	Set identification of **IC** bulk data entries which define the initial conditions.
m	Set identification of a **GUST** bulk data entry which defines the gust parameters.
n	Selects the direct input stiffness matrix. Refers to a **DMI** or **DMIG** bulk data entry.
o	Selects the direct input mass matrix. Refers to a **DMI** or **DMIG** bulk data entry.
p	Selects the direct input damping matrix. Refers to a **DMI or DMIG** bulk data entry.
q	Selects the transfer function set to be added to the input matrices. Refers to **TF** bulk data entries.
r	Set identification of **VSDAMP** and/or **TABDMP** bulk data entries that define damping data.

Remarks:

1. If any discipline has a **caseid**, then all disciplines must have a **caseid**. All **caseid** values must be unique, but they need not be in any particular order. Disciplines are implicitly numbered from 1 to n if no **caseid** values are specified. The **caseid** is only used as a reference from user defined functions in the Function Packet.

2. The **TRANSIENT** discipline does not generate design constraints for optimization.
3. type, **DLOAD** and **TSTEP** are required.
4. If **GUST** is present, **FFT** must also be used.
5. Initial conditions, **IC**, are only valid for **DIRECT** analyses. **IC** cannot be used with **GUST** or **FFT**.
6. No more than one **TRANSIENT** analysis can be done in a single boundary condition.
7. **M2PP**, **B2PP** and **K2PP** names will typically refer to **DMI** and **DMIG** entries, but may refer to any existing database entity of the proper dimension.

This page is intentionally blank.

Chapter 6.

THE FUNCTION PACKET

6.1. BACKGROUND

The Function Packet allows the user to define one or more functional forms that may be used to define an objective function or ***synthetic design constraints*** beyond those available directly through the Bulk Data packet. The Function Packet consists of functions that define mathematical equations which may reference intrinsic response functions for grid point and element response quantities, such as displacements and stresses. Furthermore, these responses may be selected from any of the optimization boundary conditions or discipline cases.

6.2. THE FUNCTION EVALUATION PROCEDURE

The user references the functions defined in the Function Packet from the Bulk Data Packet. The Bulk Data Packet, in turn, is referenced from the Solution Control Packet. Specifically, the Solution Control Packet references the functional design constraint or objective in the Bulk Data Packet in a manner similar to the way it currently references other design constraints. The Bulk Data Packet then links the design constraint to the functions within the the Function Packet. The Function Packet, in turn, defines the function specifications.

The Function Packet is compiled by ASTROS at run-time. The compiled code, which is stored on the ASTROS CADDB database (see the Programmer's Manual for a detailed description of CADDB), is then used to evaluate functions as necessary during the design process. The Function Packet, while it may look like a Fortran program, is ***non-procedural***. This means that the functional definitions, including any intermediate terms used in the functions, may be specified in any order. When it is necessary to evaluate a function during an ASTROS execution, the evaluation is performed by a process called

instantiation. Instantiation is the process of determining the value of a function by retrieving the components needed to evaluate it.

During instantiation, ASTROS determines the validity of each function both in terms of its syntax and that of the other functions it may use. This process determines that each function is legal, completely defined, and that the supporting Bulk Data, if any, are present on the database. As part of this operation, the actual number of function evaluations (or "instances") is determined. The user may define one function in the function packet but invoke it many times. Each of the invocations must be legal and complete.

The instantiation process results in the creation of data structures that describe each instance (constraint or function evaluation). These data structures are used by ASTROS to control the computation of the constituent responses. For example, if a function calls for the SIGX stress of QUAD4 100, ASTROS will ensure that the stress component is computed.

Following the normal constraint screening process, active synthetic constraints along with these data structures are used to compute required sensitivities. This is done by explicitly differentiating the user functions and using the chain rule to compute constraint gradients from the necessary response derivatives. The response gradients are computed in the normal ASTROS manner.

The following sections describe the relationship between the Solution Control Packet (including the **OPTIMIZE** command), the Bulk Data Packet and the Function Packet.

6.2.1. Solution Control Packet

The Solution Control packet is used to select functions for use as either design constraints or as the objective function. The relevant commands are described in the following sections.

6.2.1.1. Synthetic Objective Function

The ASTROS **OPTIMIZE** command is used to specify the objective function and the type of optimization to be performed. The general form of the command is:

```
{ OPTIMIZE }[ OBJECTIVE = { WEIGHT } ]
{ MINIMIZE }[             { set-id } ]
{ MAXIMIZE }[                        ]
                    [ DCFUNCTION = indep-set-id ] other-options
```

WEIGHT is a keyword that selects the weight as objective function (the original ASTROS objective function) while *set-id* is the identification number of ***ONE*** **DCONF** Bulk Data entry that may be used to define a synthetic objective. The **DCONF** entry ***MUST*** be one that resolves to a single scalar value. If the optional **OBJECTIVE** specifier is omitted, **WEIGHT** is selected by default. The **OPTIMIZE** and **MINIMIZE** options direct ASTROS to minimize the objective function while **MAXIMIZE** directs the opposite. The *indep-set-id* allows the user to specify a single subcase independent functional constraint. (See Chapter 3 for details about the *other-options*.)

To illustrate the use of the **OPTIMIZE** command, consider the following examples. For the standard weight minimization problem, the following are equivalent:

```
OPTIMIZE MAXITER = 10, CNVRGLIM = 1.3
OPTIMIZE OBJECTIVE = WEIGHT, MAXITER = 10, CNVRGLIM = 1.3
MINIMIZE OBJECTIVE = WEIGHT, MAXITER = 10, CNVRGLIM = 1.3
```

To minimize the scalar function defined by **DCONF 101**:

```
MINIMIZE OBJECTIVE = 101, MAXITER = 10, MOVLIM = 1.7
OPTIMIZE OBJECTIVE = 101, MAXITER = 10, MOVLIM = 1.7
```

To maximize the scalar function defined by **DCONF 2001**:

```
MAXIMIZE OBJECTIVE = 2001, MAXITER = 15,
         CNVRGLIM = 1.5, MOVLIM = 1.3
```

6.2.1.2. Synthetic Design Constraints

Subcase independent constraints, such as weight and thickness, may be selected directly using the **DCFUNCTION** option of the **OPTIMIZE** command:

```
OPTIMIZE DCFUNCTION = indep-set-id
```

The **DCFUNCTION** defines a ***Design Constraint Function***. In addition, the user has a mechanism to specify subcase-dependent constraints by using the **DCFUNCTION** option within the four disciplines:

- STATICS— Static structural analysis
- MODES — Normal modes of vibration
- SAERO — Steady-state aeroelastic analysis
- FLUTTER — Aeroelastic stability analysis

When defining functional constraints which are subcase dependent, a similar **DCFUNCTION** option is included within each discipline:

```
<disc> <type> [<case_id>] (DCFUNCTION = set-id)
```

where *set-id* is the identification number of one or more **DCONF** Bulk Data entries. For example:

```
OPTIMIZE DCFUNCTION = 1000  Subcase Independent Functional Contraint
 BOUNDARY SPC = 1
    STATICS (...,DCFUNCTION = 101,...)   Subcase Dependent Functional Contraint
    ...
 BOUNDARY SPC = 2, METHOD = 10
    MODES (...,DCFUNCTION = 201,...)   Subcase Dependent Functional Contraint
    ...
 ...
END
```

An additional option is available for each of the ASTROS discipline commands. Each discipline may include an identification number, *case-id*, which can be used in selecting response quantities for user functions. This identification number simply follows the discipline name:

```
STATICS [case-id] (...)
MODES [case-id] (...)
```

If *case-id*s are not specified, then they are numbered consecutively from 1 to n. If *case-id*s are specified, then they must appear for all discipline commands. In a typical case, the *case-id* associated with a constrained response will be inherited from the discipline that references the function. It is possible, however, to explicitly reference a *case-id* in a user function. You use this feature to create synthetic functions that combine results from many subcases.

6.2.2. Bulk Data Packet

The calling arguments that instantiate functional design constraints are defined using the **DCONF** entry in the Bulk Data packet. The primary use of functions is for synthetic response constraints, and synthetic objective functions that are requested in the Solution Control packet.

In the following example, the Bulk Data Packet defines values for the element identification numbers and the allowable stress resultant for functional design constraint **101**. It points to the function, **SIG**, in the Function Packet. This will be more fully explained in subsequent sections.

BEGIN BULK									
DCONF	101		SIG						+DCN1
+DCN1	EID	10	ALLOW	45000.0					
ENDDATA									

The **DCONF** Bulk Data defines the calling arguments for the named function, **SIG**. Each argument is defined by its name (e.g. **EID**) and the value to be used in this invocation of the function. Notice that, by using name/value pairs, there is no order dependence. While arguments may thus be defined on the **DCONF** entry, it is still **required to define all of the function arguments**.

6.3. FUNCTION SYNTAX

The function packet contains the functional specification equations that are used as either the design constraints or the objective function. This packet has the general form:

```
FUNCTIONS
   ...
   func_def
   ...
ENDFUNC
```

where *func_def* is the definition of a specific function.

Each function, *func_def*, must have a ***single variable*** specified on the left of an equality expression:

```
var_name [ (argi) ] = expression;
```

Where variable names, *var_name*, may be any string of one to eight alphanumeric characters starting with a letter. Arguments, *argi*, may also be passed to the function and used in the expression. These arguments must be referenced in the expression and on the design constraint Bulk Data entry, **DCONF**. Expressions combine arguments, constants and other functions. They may be continued over multiple lines as long as the final line ends with a semicolon (;) character. All of the rules for arithmetic expression evaluation follow the standard rules of FORTRAN. Note that unlike FORTRAN, arguments which are not used may be omitted from the calling list, as is the case with the MAPOL language. Examples of this will be shown in later Subsections.

The following example defines a function which computes the allowable value of the stress resultant for an element.

```
FUNCTIONS
   ...
   SIG(eid,allow)= (SQRT( STRESS(eid,SIGX)**2
                   + STRESS(eid,SIGY)**2 ) / allow ) - 1.0;
   ...
ENDFUNC
```

In this example, the function name is **SIG** which has two arguments, the element identification number, *eid*, and the allowable stress, *allow*. It also references one intrinsic response function, **STRESS**, and one mathematical intrinsic, **SQRT**. Specifically, intrinsic functions are built-in functions which retrieve either standard ***mathematical functions*** such as sine and cosine, or they are the functions which recover the solution results, which are called ***response functions***. There are 20 mathematical functions and 27 response functions which may be used in the Function Packet.

6.3.1. Mathematical Functions

There are 20 intrinsic mathematical functions. The definitions for these functions are shown in Table 6-1.

6.3.2. Response Functions

The 27 available response functions fall into the following categories:

- Design Variables
- Selection
- Geometry
- Grid Point Response
- Element Response
- Natural Frequency
- Flutter
- Static Aero

Each of these is described in the following Sections.

Table 6-1. Mathematical Intrinsics

FUNCTION	DESCRIPTION
ABS(a)	Absolute value: $\mid a \mid$
ACOS(a)	Inverse cosine: $\cos^{-1}(a)$
ASIN(a)	Inverse sine: $\sin^{-1}(a)$
ATAN2(a,b)	Inverse tangent: $\tan^{-1}(a/b)$
CMPLX(a,b)	Convert to complex: $a + bi$
COS(a)	Cosine: $\cos(a)$
DEGS(a)	Convert to degrees: $\frac{a\,\pi}{180}$
EXP(a)	Exponential function e^a
HERTZ(a)	Convert to hertz $\left(\frac{a}{2\pi}\right)$
IMAG(a)	Use the imaginary part of complex *a*
INT(a)	Convert to integer
LOG(a)	Log_e
LOG10(a)	Log_{10}
MOD(a,b)	Remainder
RADS(a)	Convert to radians $\left(\frac{180\,a}{\pi}\right)$
REAL(a)	Use the real part of complex *a*
SIGN(a)	Algebraic sense function, -1 for negative *a*, +1 for positive *a*, and 0 if *a*=0
SIN(a)	Sine: $\sin(a)$
SQRT(a)	Square root
TAN(a)	Tangent: $\tan(a)$
The arguments **a** and **b** represent either constants or expressions.	

6.3.2.1. Design Variable Function

To specify the current value of a design variable, the function:

is used, where the design variable is specified in the Bulk Data Packet as an identification, *dvid*, or as a set, **GDVLIST**.

6.3.2.2. Selection Functions

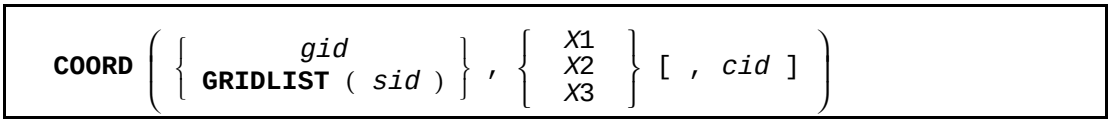
Selection functions are provided to aid in the reference of data items, such as grid or element identifications. The data items may be referenced by either individual data entries through the definition of values passed in the **DCONF** argument list or by data lists through the passing of a list defined in the Bulk Data packet. The functions that represent data lists are shown in Table 6-2 along with the Bulk Data entries which define the set lists. The argument represents a list, *sid*, defined by the parameters in **DCONF** Bulk Data.

6.3.2.3. Geometric Functions

Geometric functions are provided to facilitate the eventual definition of geometry based design variable linking and to define kinematic admissibility constraints. The first function is:

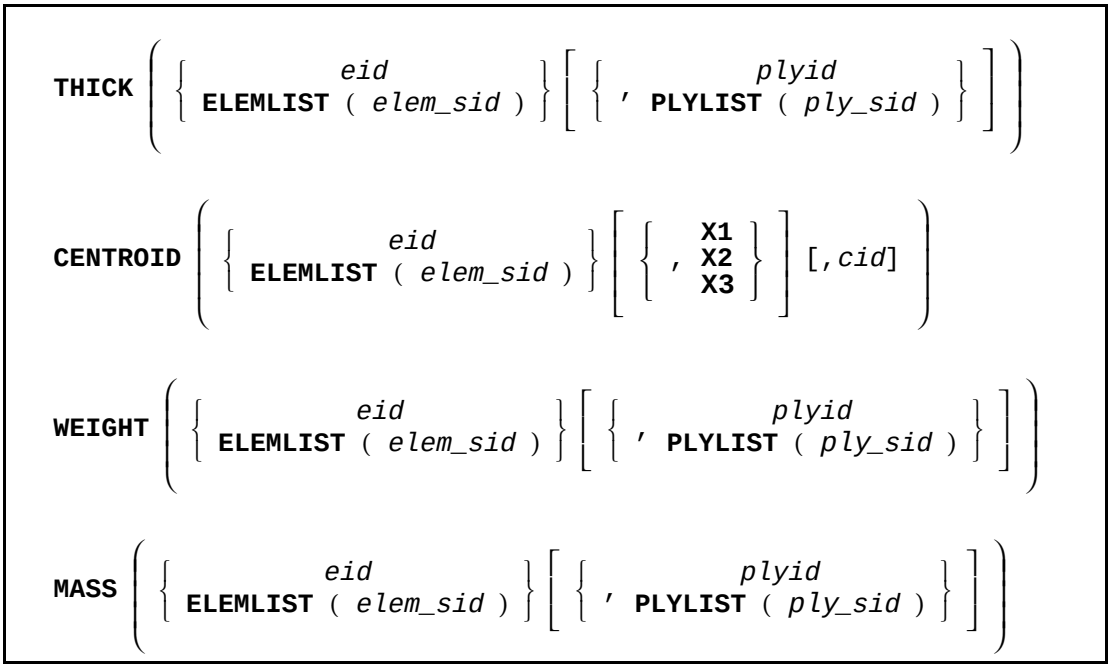
Table 6-2. Selection Functions

SELECTION FUNCTION	BULK DATA ENTRY	DESCRIPTION
CASELIST(*sid***)**	**CASELIST**	Case List
DENSLIST(*sid***)**	**DENSLIST**	Density List
ELEMLIST(*sid***)**	**ELEMLIST**	Element List
GDVLIST(*sid***)**	**GDVLIST**	Global design variable List
GRIDLIST(*sid***)**	**GRIDLIST**	Grid List
ITERLIST(*sid***)**	**ITERLIST**	Iteration List
LDVLIST(*sid***)**	**LDVLIST**	Local design variable List
MACHLIST(*sid***)**	**MACHLIST**	Mach List
MODELIST(*sid***)**	**MODELIST**	Mode List
PLYLIST(*sid***)**	**PLYLIST**	Ply List
VELOLIST(*sid***)**	**VELOLIST**	Velocity List



The **COORD** function retrieves the current value of a geometric coordinate **X1**, **X2**, or **X3** for the requested grid points referenced either as a grid value or a grid list, **GRIDLIST**. The grid point will be retrieved in the selected coordinate system, *cid*. If the *cid* reference is omitted, then the coordinate value is returned in the ***input coordinate system*** of the GRID point, i.e. the **CP** field of the Bulk Data entry **GRID**. A *cid* of 0 requests that the coordinate be returned in the basic coordinate system.

The interpretation of **X1**, **X2**, and **X3** depends on whether the *cid* coordinate system is rectangular, cylindrical, or spherical. In addition to the grid point geometry, there are functions which return information about the specific finite elements. These are:

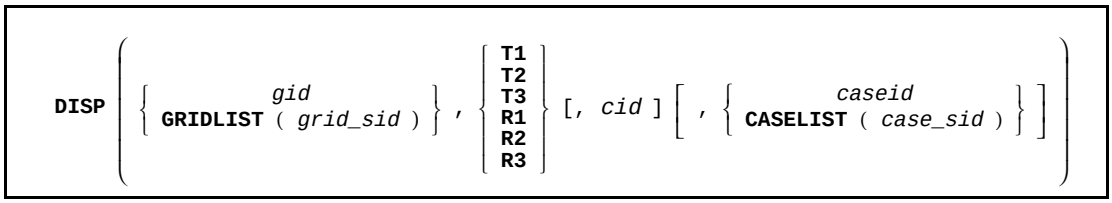


The **THICK** function returns the thickness of the requested two-dimensional element. The **CENTROID** function returns the centroid of the requested one-dimensional, two-dimensional and three-dimensional element in the coordinate system *cid*. The **WEIGHT** function returns the weight of the element selected and the **MASS** function returns the mass of the element selected.

The element is referenced either by an element identification, *eid*, or an element list, **ELEMLIST**. If an element identification number is used, then the *eid* must be unique. If element identification numbers are not unique, then an element list must be used. If the *cid* reference is omitted, or is 0, then the coordinate is returned in the basic coordinate system. Otherwise, it is returned in the specified coordinate system. Composite elements must have their layer numbers specified by a layer number, *plyid*, or a layer list, **PLYLIST**. The **PLYLIST** is not required for non-composite elements, and if present, it is ignored.

6.3.2.4. Grid Point Response Functions

The grid point response function is defined by:



where **DISP** represents displacements. This function allows the current grid result values in the coordinate system, *cid*, of a component **T1**, **T2**, **T3**, **R1**, **R2** or **R3** for the requested grid points, defined either by its value, *gid*, or a grid point list, **GRIDLIST**, to be retrieved in the requested CASE value or list. If the subcase reference is omitted, then the specific discipline request defines the requested subcase. If the *cid* reference is omitted, then the coordinate value is returned in the displacement coordinate system of the grid points. A *cid* of 0 requests that the coordinate be returned in the basic coordinate system.

6.3.2.5. Element Response Functions

The element response functions are defined by:

STRESS (*elemop*,*stress_comp*[,*plyop*][,*caseop*][,*modeop*])

STRAIN (*elemop*,*strain_comp*[,*plyop*][,*caseop*][,*modeop*])

where:

elemop => { *eid* | **ELEMLIST** (*elem_sid*) }

plyop => { *plyid* | **PLYLIST** (*ply_sid*) }

caseop => { *caseid* | **CASELIST** (*case sid*) }

modeop => { *modeid* | **MODELIST** (*mode_sid*) }

These functions allow a component of the requested element results, referenced either by its value, *eid*, or a list, **ELEMLIST**, to be retrieved for the requested **CASE**s and **MODE**s value or list. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used. Composite elements must have their layer numbers specified by a layer number, *plyid*, or a layer list, **PLYLIST**. The element response components, for composite elements, will always be recovered at the center of the layer. The allowable response components for each element type are shown in Table 6-3.

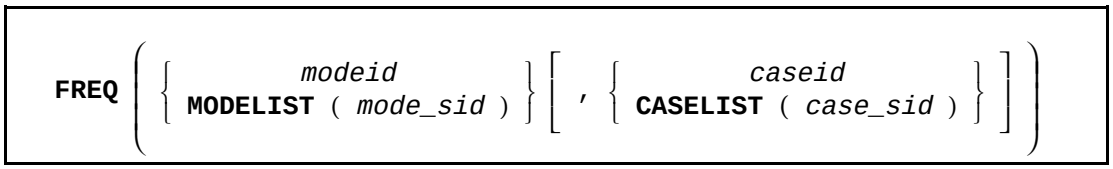
Table 6-3. Element Response Components

ELEMENT		*stress_comp*	*strain_comp*
ROD		SIGAXL SIGTOR SIG1 SIG2 MAXSHEAR	
BAR		SIGAXL SIGCA SIGDA SIGEA SIGFA SIGCB SIGDB SIGEB SIGFB	EPSAXL EPSCA EPSDA EPSEA EPSFA EPSCB EPSDB EPSEB EPSFB
SHEAR		MAXSHEAR	MAXSHEAR
QDMEM1 **TRMEM**		SIGX SIGY TAUXY SIG1 SIG2 MAXSHEAR FIBER TRANSV	EPSX EPSY EPSXY EPS1 EPS2 MAXSHEAR FIBER TRANSV
QUAD4 **TRIA3**	**MIDPLANE**	SIGX SIGY TAUXY SIG1 SIG2 MAXSHEAR FIBER TRANSV	EPSX EPSY EPSXY EPS1 EPS2 MAXSHEAR FIBER TRANSV
	TOP SURFACE	TSIGX TSIGY TTAUXY TSIG1 TSIG2 TMAXSHEAR TFIBER TTRANSV	TEPSX TEPSY TEPSXY TEPS1 TEPS2 TMAXSHEAR TFIBER TTRANSV
	BOTTOM SURFACE	BSIGX BSIGY BTAUXY BSIG1 BSIG2 BMAXSHEAR BFIBER BTRANSV	BEPSX BEPSY BEPSXY BEPS1 BEPS2 BMAXSHEAR BFIBER BTRANSV

The specific discipline request defines whether the case and/or mode is a valid request in the response functions. The mode sequence number is used only if the discipline is MODES. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

6.3.2.6. Natural Frequency Constraints

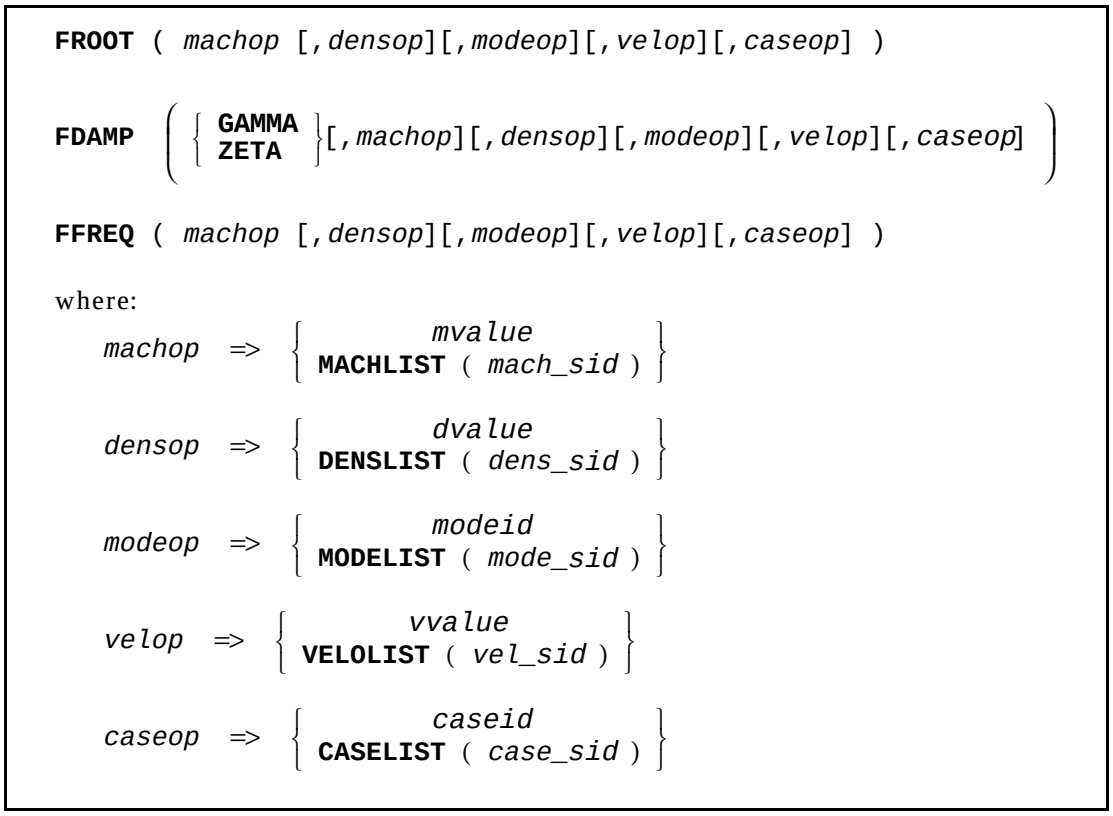
To select a natural frequency computed in a MODES discipline, the function:



is used. Again, a single mode, *modeid*, or a list of modes, **MODELIST**, is selected. The optional *caseid* allows the selection of modes from a specific case.

6.3.2.7. Flutter Response Functions

The flutter response functions are **FROOT**, **FDAMP**, and **FFREQ** which represent the flutter root, flutter damping, and flutter frequency, respectively. These functions are defined by:



The arguments to the first function, **FROOT**, includes a Mach value, *machop*, in either of the two forms shown. It may be an explicit value, *mvalue*, or a Mach list, **MACHLIST**. Similarly, it requires a density ratio value, *dvalue*, or a density list, **DENSLIST**, selected mode index, *modeid*, or a mode list,

MODELIST, for the modes in the flutter set and the analysis velocity value, *vvalue*, or a velocity list, **VELOLIST**. The function **FROOT** then returns a complex number representing the flutter root:

$$p = k(\gamma + i)$$

The arguments to the second function, **FDAMP**, are a component, **GAMMA** or **ZETA**, the Mach value, a density ratio, a selected mode index for the modes in the flutter set and the analysis velocity. The function then returns the specified flutter damping coefficient as defined below.

$$\gamma = \begin{cases} \dfrac{\mathrm{Re}(p)}{\mathrm{Im}(p)} & ; \text{ for complex } p \\ \dfrac{\mathrm{Re}(p)}{\ln 2} & ; \text{ for real } p \end{cases}$$

$$\zeta = \left(\left(\frac{\mathrm{Re}(p)}{\mathrm{Im}(p)} \right)^2 + \mathrm{Re}(p)^2 \right)^{1/2} \quad ; \text{for complex } p$$

The third function, **FFREQ**, has the same arguments as **FROOT** and returns the frequency in radians. Conversion to Hertz may be accomplished by using the **HERTZ** intrinsic function.

6.3.2.8. Static Aero Response Functions

The static aero response functions are defined by:

FLEXCF (*axis* , *trim_param* [, { *caseid* / **CASELIST** (*case_sid*) }])

RIGIDCF (*axis* , *trim_param* [, { *caseid* / **CASELIST** (*case_sid*) }])

TRIM (*trim_param* [, { *caseid* / **CASELIST** (*case_sid*) }])

where **FLEXCF, RIGIDCF** and **TRIM** represent flexible stability coefficient, rigid direct stability coefficient, and trim parameter values, respectively. The flex and rigid functions allow as input the axis, *axis*, and the trim parameters, *trim_param*. The trim function inputs only the trim parameters. When *axis* (see below) is **ROLL**, **PITCH** or **YAW**, these functions return their appropriate results in radians. If degrees are required, the results may be converted using the **DEGS** intrinisic function. The optional *caseid* allows the selection of a specific case.

The allowable values for *axis* are:

axis = { **DRAG** **SIDE** **LIFT** **ROLL** **PITCH** **YAW** }

The allowable control surfaces, *trim_param*, are:

trim_param = { **ALPHA** **BETA** **PRATE** **QRATE** **RRATE** **PACCEL** **QACCEL** **RACCEL** ... ***User Surfaces*** }

where the ***User Surfaces*** are defined using either **AESURF** or **CONLINK** Bulk Data entries.

6.3.3. Ordered Sets

As seen, functions allow the user to define synthetic response constraints and synthetic objective functions. To allow maximum flexibility, a single function may be referenced many times. Because multiple references may become verbose, a special provision has been made to allow the use of sets.

When using a single set in a function, the results are straight-forward. The function is instantiated for every entry in the set. When multiple sets are used, there are several ways to define the resulting values of a function. Specifically, these methods relate to the number of members, or cardinality, and the order of the resulting sets.

An unambiguous definition of multiple set use has been implemented. Each set that appears in the function ***MUST*** have the same cardinality, or, one or more of the sets may have a single member. When the function is evaluated, the members of each set are placed in a one-to-one correspondence with each other. Consider the following example:

```
FUNCTIONS
   ...
   FUN1 = DISP(GRIDLIST(1),T1,,CASELIST(1001));
   ...
ENDFUNC
BEGIN BULK
...
GRIDLIST,1,1,2
CASELIST,1001,3,4
...
ENDDATA
```

This results in two function evaluations:

```
DISP(1,T1,,3)
DISP(2,T1,,4)
```

Other examples of set use are presented in the following Section.

6.4. EXAMPLES

The following examples demonstrate how the definition and linking of the functions with the Solution Control, Bulk Data, and the Function Packet is accomplished. For each of the examples, the Solution Control packet references the functional design constraint in the Bulk Data Packet. The Bulk Data Packet then links the design constraint to the Functional Packet and the Function Packet defines the function specifications.

Example 1: Displaced Coordinate Limit

The following example computes four constraints for the displaced coordinate, **X**, for a set of four grid points, assuming that **XOLD** and **T1** are in the same coordinate system. First, the Solution Control packet references the functional design constraint, **101**, in the Bulk Data Packet for the STATICS discipline of boundary condition 1.

```
OPTIMIZE
   ...
   BOUNDARY SPC = 1
      STATICS (..., DCFUNCTION = 101, ...)
   ...
END
```

The Function Packet defines the function specification for computing the allowable displaced coordinates. The general expression for the Function packet is:

XNEW = XOLD + T1 for grids **5**, **10**, **15**, **20**

This expression is then coded in the Function packet as:

```
FUNCTIONS
   ...
$ Location of the X coordinate for the supplied Grid list
   XOLD(GLIST)= COORD(GLIST, X1);
$ Location of the displaced coordinate
   XNEW(GLIST)= XOLD(GLIST) + DISP(GLIST, T1);
$ Constraint for the displaced coordinate
   CONST(GLIST,ALLOW)= ( XNEW(GRIDLIST(GLIST)) / ALLOW ) - 1.0;
  ...
ENDFUNC
```

The Bulk Data Packet defines the grid list, *glist*, and arguments for constraint set **101** The constraint set links the design constraint, **CONST**, to the Functional Packet, and defines two arguments. The first argument identifies the **GRIDLIST** to use in the function and the second argument defines the allowable upper limit, *allow*, of the constraint. Note that this is the technique used to define a normalized constraint for the optimization step. ***It is highly recommended that functional constraints be normalized in this manner***.

In order for the optimizer to perform properly, it is mandatory that the synthetic constraint be negative when satisfied and positive when violated. It is recommended that the synthetic constraint be normalized such that its values are on the order of unity.

The Bulk Data used to define the functional parameters is then given by:

BEGIN BULK									
...									
GRIDLIST	1	5	10	15	20				
DCONF	101		CONST						+DCN1
+DCN1	GLIST	1	ALLOW	100.0					
ENDDATA									

Example 2: Stress Resultant Limits

The following example computes multiple constraints for the stress resultants of selected QUAD4 elements. The Solution Control packet references the functional design constraint, **101**, in the Bulk Data Packet for the STATICS discipline of boundary condition **1**.

```
OPTIMIZE
   ...
   BOUNDARY SPC = 1
      STATICS (..., DCFUNCTION = 101, ...)
   ...
END
```

The Function Packet defines the function specification for computing the allowable stress resultant. The general expression for this is:

$$\mathbf{RESULT} = \sqrt{\mathbf{SIGX}^2 + \mathbf{SIGY}^2}$$

This is then applied to all QUAD4 elements in the range of **1** through **10000** by the following Function Packet:

```
FUNCTIONS
   ...
$ Alias for the element list selection function
   ELST(ELIST)= ELEMLIST(elist);
$ Stress resultant
   RESULT(ELIST)= SQRT(STRESS(ELST(ELIST),SIGX)**2 +
                       STRESS(ELST(ELIST),SIGY)**2);
$ Constraint for the Stress resultant
   CONST(ELIST,ALLOW)= ( RESULT(ELIST) / ALLOW ) - 1.0;
   ...
ENDFUNC
```

The Bulk Data Packet defines the element list for the QUAD4 elements, defines functions and arguments for design constraint set **101**, which points to the design constraint function, **CONST**. The first argument identifies the **ELEMLIST** to use in the function and the second argument defines the allowable upper limit of each constraint.

BEGIN BULK									
...									
ELEMLIST	1	QUAD4	1	THRU	10000				
DCONF	101		CONST						+DCN1
+DCN1	ELIST	1	ALLOW	100.0+3					
ENDDATA									

Example 3: Noninterference Constraints

This example computes 16 constraints for the relative location between two sets of grid points, G_1 and G_2. The relative location equals the magnitude of the square root of the sum of the squares of the displaced coordinate divided by the sense of the dot product between the points such that a positive number means that the two points are not touching. This algorithm assumes that the geometric locations and the displacements are in the same coordinate system. The equations to be programmed are shown in the following:

$$G_1 = \{ 5 , 6 , 7 , \ldots , 18 , 19 , 20 \}$$

$$G_2 = \{ 105 , 106 , 107 , \ldots , 118 , 119 , 120 \}$$

In the following equations the elements of these sets are denoted by $i \in G_2$, $j \in G_1$:

$$XMAG_{ij} = \left[\left[\left(T1_i + \delta T1_i \right) - \left(T1_j + \delta T1_j \right) \right]^2 + \left[\left(T2_i + \delta T2_i \right) - \left(T2_j + \delta T2_j \right) \right]^2 + \left[\left(T3_i + \delta T3_i \right) - \left(T3_j + \delta T3_j \right) \right]^2 \right]^{1/2} \quad ; \quad i \in G_1 \, , \; j \in G_2$$

$$SDOT_{ij} = sign \left[\left(T1_i - T1_j \right) \left[\left(T1_i + \delta T1_i \right) - \left(T1_j + \delta T1_j \right) \right] + \left(T2_i - T2_j \right) \left[\left(T2_i + \delta T2_i \right) - \left(T2_j + \delta T2_j \right) \right] + \left(T3_i - T3_j \right) \left[\left(T3_i + \delta T3_i \right) - \left(T3_j + \delta T3_j \right) \right] \right] \quad ; \quad i \in G_1 \, , \; j \in G_2$$

$$RDISP_{ij} = \frac{XMAG_{ij}}{SDOT_{ij}}$$

The solution control packet references the functional design constraint, **101**, in the Bulk Data Packet for the STATICS discipline of boundary condition 1.

```
OPTIMIZE
   ...
   BOUNDARY SPC = 1
      STATICS (..., DCFUNCTION = 101, ...)
   ...
END
```

The Function Packet defines the function specifications for computing the relative displacements between two sets of grid points.

```
FUNCTIONS
   ...
$
$  Alias for the grid list
GLST(GLIST) = GRIDLIST(GLIST);
$  XMAG = magnitude
XMAG(GLIST1,GLIST2) =
SQRT(((COORD(GLIST1,X1)+DISP(GLIST1,T1))-(COORD(GLIST2,X1)+DISP(GLIST2,T1)))**2+
     ((COORD(GLIST1,X2)+DISP(GLIST1,T2))-(COORD(GLIST2,X2)+DISP(GLIST2,T2)))**2+
     ((COORD(GLIST1,X3)+DISP(GLIST1,T3))-(COORD(GLIST2,X3)+DISP(GLIST2,T3)))**2);
$  SDOT = Sign of the dot product
```

Continued on following page.

```
SDOT(GLIST1,GLIST2) =
   SIGN((COORD(GLIST2,X1)+DISP(GLIST2,T1)-COORD(GLIST1,X1)-DISP(GLIST1,T1)) *
        (COORD(GLIST2,X1) - COORD(GLIST1,X1)) +
        (COORD(GLIST2,X2)+DISP(GLIST2,T2)-COORD(GLIST1,X2)-DISP(GLIST1,T2)) *
        (COORD(GLIST2,X2) - COORD(GLIST1,X2)) +
        (COORD(GLIST2,X3)+DISP(GLIST2,T3)-COORD(GLIST1,X3)-DISP(GLIST1,T3)) *
        (COORD(GLIST2,X3) - COORD(GLIST1,X3)));
$  Constraint for relative disp
$  RDISP = -(XMAG/SDOT)
RDISP(GLIST1,GLIST2) = -XMAG(GLST(GLIST1),GLST(GLIST2)) /
                       SDOT(GLST(GLIST1),GLST(GLIST2));
ENDFUNC
```

The Bulk Data Packet defines two grid lists, references design constraint 101, which links the design variable, **RDISP**, to the Functional Packet, and defines two arguments. The arguments identify the **GRIDLIST**s to use in the function.

BEGIN BULK									
...									
GRIDLIST	1	5	THRU	20					
GRIDLIST	2	105	THRU	120					
DCONF	101		RDISP						+DCN1
+DCN1	GLIST1	1	GLIST2	2					
ENDDATA									

Example 4: Constraint Instantiation with Explicit Subcases

The following example computes five constraints from subcases defined independently of the analysis discipline. The function evaluates the expression which takes the displacement component **T3** and divides by **2.0** for a set of grid points recovered for a set of unique displacements. The solution

control packet references the functional design constraint, **101**, in the Bulk Data Packet for the STATIC discipline of boundary condition 1.

```
OPTIMIZE
   ...
  DCFUNCTION = 101
  BOUNDARY SPC = 1
      STATICS 1000 (...)
      STATICS 2000 (...)
      STATICS 3000 (...)
      STATICS 4000 (...)
      STATICS 5000 (...)
   ...
END
```

The Function Packet defines the function specification for computing the allowable displacement component **T3**. The general expression for the function is:

$$\mathbf{COMP} = \frac{\mathbf{T3}}{\mathbf{2.0}} \quad \text{for} \begin{cases} \text{grid 5 recovered at subcase 1000} \\ \text{grid 10 recovered at subcase 2000} \\ \text{grid 15 recovered at subcase 3000} \\ \text{grid 20 recovered at subcase 4000} \\ \text{grid 25 recovered at subcase 5000} \end{cases}$$

which is defined by the Function packet:

```
FUNCTIONS
   ...
$ Recover the Normalized Displacement component T3
$ for given grid and subcase list
   COMP(GLIST,CLIST,FACT) =
         DISP(GRIDLIST(GLIST),T3,,CASELIST(CLIST)) / FACT;
$ Constraint for the displacement component
   CONST(GLIST,CLIST,ALLOW)=( COMP(GLIST,CLIST,2.0)/ALLOW )-1.0;
   ...
ENDFUNC
```

The Bulk Data Packet defines a grid list and a subcase list, references design constraint 101, which links the design variable, **CONST**, to the Functional Packet, and defines three arguments. The first argument represents the **GRIDLIST** identification, the second argument is the **CASELIST** identification and the third argument defines the allowable upper limit of the constraint.

BEGIN BULK									
...									
GRIDLIST	1	5	10	15	20	25			
CASELIST	101	1000	2000	3000	4000	5000			
DCONF	101		CONST						+DCN1
+DCN1	GLIST	1	CLIST	101	ALLOW	0.2			
ENDDATA									

Example 5: Multiple Function Evaluations

The following example will compute 25 constraints. The function evaluates the expression which takes two times the displacement component **T3** for a set of grid points recovered for subcases **1**, **2**, **3**, **4**, **5**. The solution control packet references the functional design constraint, 101, in the Bulk Data Packet for the STATICS discipline of boundary condition 1.

```
OPTIMIZE
  ...
  BOUNDARY SPC = 1
    STATICS (..., DCFUNCTION = 101, ...)
  ...
END
```

The Function Packet defines the function specification for computing the allowable displacement component for **T3**. The general expression for the Function packet is:

COMP = 2*T3 for grids 5, 10, 15, 20, 25

which is defined by the Function packet:

```
FUNCTIONS
   ...
$ Recover the Displacement component, T3, times 2.0
   COMP(GLIST,CASEID,MULPT) = MULPT *
          DISP(GRIDLIST(GLIST),T3,,CASEID);
$ Constraint for the component value
   CONST(GLIST,CASEID,MULPT,ALLOW) = ( COMP(GLIST,CASEID,MULPT)
                                       / ALLOW ) - 1.0;
   ...
ENDFUNC
```

The Bulk Data Packet gives the grid list, defines five invocations of design constraint **101**, references the design constraint function, **CONST**, and defines its four arguments. The arguments identify the multiplier used with the displacement component, the **GRIDLIST**, the subcase identification and the allowable upper limit of the constraint.

BEGIN BULK									
...									
GRIDLIST	1	5	10	15	20	25			
DCONF	101	CASE1	CONST						+DCN1
+DCN1	GLIST	1	CASEID	1	MULPT	2	ALLOW	20.0	
DCONF	101	CASE2	CONST						+DCN1
+DCN1	GLIST	1	CASEID	2	MULPT	2	ALLOW	20.0	
DCONF	101	CASE3	CONST						+DCN1
+DCN1	GLIST	1	CASEID	3	MULPT	2	ALLOW	20.0	
DCONF	101	CASE4	CONST						+DCN1
+DCN1	GLIST	1	CASEID	4	MULPT	2	ALLOW	20.0	
DCONF	101	CASE5	CONST						+DCN1
+DCN1	GLIST	1	CASEID	5	MULPT	2	ALLOW	20.0	
ENDDATA									

Example 6: Invalid List Cardinality

The following example demonstrates an invalid request for a set of grid point data recovered for a list of unique subcases. The solution control packet references the functional design constraint 101, in the Bulk Data Packet for the STATICS discipline of boundary condition 1.

```
OPTIMIZE
  ...
  BOUNDARY SPC = 1
    STATICS (..., DCFUNCTION = 101, ...)
  ...
END
```

The Function Packet defines the function specification for computing the allowable displacements component for **T3**. The general expression for the Function packet is:

$$\textbf{COMP = 2*T3} \quad \text{for} \quad \begin{cases} \text{grid 5 recovered at subcase 1} \\ \text{grid 10 recovered at subcase 2} \\ \text{grid 15 recovered at subcase 3} \\ \text{grid 20 recovered at subcase 4} \end{cases}$$

which is defined by the Function packet:

```
FUNCTIONS
  ...
  $ Recover the Displacement component, T3, times 2.0
  COMP(GLIST,CLIST,MULT) =
       MULT * DISP(GRIDLIST(GLIST),T3,,CASELIST(MLIST));
  $ Constraint for the Component Value
  CONST(GLIST,CLIST,MULT,ALLOW)=
                        ( COMP(GLIST,CLIST,MULT) / ALLOW ) - 1.0;
  ...
ENDFUNC
```

The Bulk Data Packet defines the grid list and subcase identification list, defines the design constraints **101**, which references the design constraint function, **CONST**, and defines its four arguments. The arguments identify the **GRIDLIST**, the **CASELIST**, the multiplier used with the displacement component, and the allowable upper limit of the constraint.

BEGIN BULK									
...									
$ Grid list with 4 Grid points identified									
GRIDLIST	1	5	10	15	20				
$ Subcase list with 5 Subcases identified									
CASELIST	101	1	2	3	4	5			
DCONF	101		CONST						+DCN1
+DCN1	GLIST	1	CLIST	101	MULT	2	ALLOW	100	
ENDDATA									

There are no constraints generated because the **GRIDLIST** contains four values and the **CASELIST** contains five values. ASTROS will terminate during the processing of the user input data. ***As indicated earlier, the cardinality of the sets must be equal.***

Example 7: Missing Bulk Data

The following example demonstrates an invalid request for constraints of the normal stress in the element's X direction. The solution control packet references the functional design constraint, 101, in the Bulk Data Packet for the STATICS discipline of boundary condition 1.

```
OPTIMIZE
  ...
  BOUNDARY SPC = 1
    STATICS (..., DCFUNCTION = 101, ...)
  ...
END
```

The Function Packet defines the function specification for computing the allowable normal stress in the element X-direction. The general expression for the Function packet is:

VALUE = SIGX for elements **5**, **10**, **15**, **20**

which is defined by the Function packet:

```
FUNCTIONS
  ...
  $ Constraint for Element Stress
  VALUE(ELIST) = ( STRESS(ELEMLIST(ELIST),SIGX) / 25000.0 ) - 1.0;
  ...
ENDFUNC
```

The Bulk Data Packet defines design constraint **101**, defines the design constraint function, **VALUE**, and defines one argument, the **ELEMLIST** identification (**1**), for the function.

```
BEGIN BULK
...
$ Design Constraint Function
DCONF   101     VALUE                                   +DCN1
+DCN1   ELIST   1
ENDDATA
```

No constraints will be generated because element list 1 is not defined in the Bulk Data packet.

Example 8: Missing Argument Definitions

The following example demonstrates an invalid request to compute the constraints for the normal stress in the element's X direction. The solution control packet references the functional design constraint, 101, in the Bulk Data Packet for the STATICS discipline of boundary condition 1.

```
OPTIMIZE
   ...
   BOUNDARY SPC = 1
      STATICS (..., DCFUNCTION = 101, ...)
   ...
END
```

The Function Packet defines the function specification for computing the allowable normal stress in the element's X direction. The general expression for the Function packet is:

VALUE = SIGX for elements **5**, **10**, **15**, **20**

which is defined by the Function packet:

```
FUNCTIONS
   ...
$ Constraint for Element stress
   VALUE(ELIST) = (STRESS(ELEMLIST(ELIST),SIGX)/45000.0 ) - 1.0;
   ...
ENDFUNC
```

The Bulk Data Packet defines design constraint **101**, which referenced the design constraint function, **VALUE**:

BEGIN BULK									
...									
ELEMLIST	1	5	10	15	20				
DCONF	101		VALUE						+DCN1
+DCN1	GLIST	1							
ENDDATA									

No constraints will be generated because there is no definition in the DCONF bulk data entry for the element list argument.

Example 9: Modified Flutter Damping Constraint

The following example will compute 32 constraints on the critical damping ratio ζ for mach values of **0.8** and **1.2**, density ratio values of **0.8** and **1.0**, mode index list of **1** and **2**, and a velocity list from **600.0** through **1000.0**. The solution control packet references the functional design constraint, **101**, in the Bulk Data Packet for the FLUTTER discipline of boundary condition 1.

```
OPTIMIZE
  ...
  BOUNDARY SPC = 1
    FLUTTER (..., DCFUNCTION = 101, ...)
  ...
END
```

The Function Packet defines the function specification for computing the constraint values for ζ. The general expression for the Function packet is:

$$\zeta = \left(\left(\frac{\mathrm{Re}\,(p)}{\mathrm{Im}\,(p)} \right)^2 + \mathrm{Re}(p)^2 \right)^{1/2}$$; where p is the flutter eigenvalue.

which is defined by the Function packet:

```
FUNCTIONS
  ...
  $ Constraint for ZETA  0.15
  ZETA(MACH, DENS, MODE, VINDX ) =
  1.0 - ( FDAMP(ZETA,MACH, DENS, MODE, VELOLIST(VINDX)) / 0.15);
  ...
ENDFUNC
```

The Bulk Data Packet defines values for the **MACH**, **DENS**, **MODE**, and **VELO** arguments, for function design constraint **101** which points to the function, **ZETA**, in the Functional Packet.

BEGIN BULK									
...									
$ Velocity list									
VELOLIST	4	600.	800.	900.	1000.				
$ Design constraint function request									
DCONF	101	M0P810K	ZETA						+DCN1
+DCN1	MACH	0.8	DENS	0.8	MODE	1	VINDX	4	
DCONF	101	M0P8SL	ZETA						+DCN1
+DCN1	MACH	0.8	DENS	1.0	MODE	1	VINDX	4	
DCONF	101	M1P210K	ZETA						+DCN1
+DCN1	MACH	1.2	DENS	0.8	MODE	1	VINDX	4	
DCONF	101	M1P2SL	ZETA						+DCN1
+DCN1	MACH	1.2	DENS	1.0	MODE	1	VINDX	4	
DCONF	101	M0P810K	ZETA						+DCN1
+DCN1	MACH	0.8	DENS	0.8	MODE	2	VINDX	4	
DCONF	101	M0P8SL	ZETA						+DCN1
+DCN1	MACH	0.8	DENS	1.0	MODE	2	VINDX	4	
DCONF	101	M1P210K	ZETA						+DCN1
+DCN1	MACH	1.2	DENS	0.8	MODE	2	VINDX	4	
DCONF	101	M1P2SL	ZETA						+DCN1
+DCN1	MACH	1.2	DENS	1.0	MODE	2	VINDX	4	
ENDDATA									

6.5. INSTRINSIC RESPONSE COMMANDS

The ASTROS Instrinsic Response Function Commands are described in this section.

Comment: **$**

Purpose:

To insert commentary text into the Function packet.

Usage:

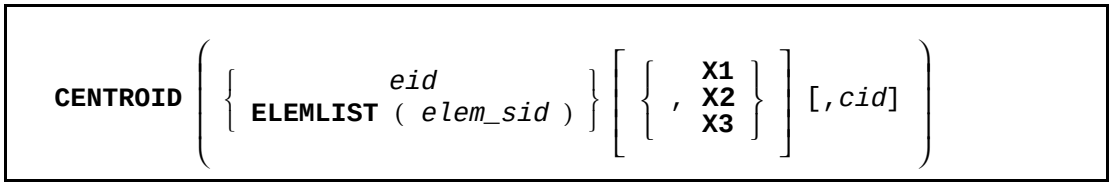
```
$ any text may appear here
```

Intrinsic Function: **CENTROID**

Purpose:

To return the centroidal coordinates of the requested elements.

Usage:



Function Argument:

eid	Identification of an element specified in the Bulk Data Packet.
elem_sid	Set identification of an **ELEMLIST** bulk data entry used to specify an element.
Xi	Component for the geometric coordinate.
cid	Identification of a coordinate system specified in the Bulk Data Packet.

Notes:

1. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used.
2. If the *cid* reference is omitted, then the coordinate value is returned in the input coordinate system of the element.
3. A *cid* of 0 requests that the coordinate be returned in the basic coordinate system.
4. The interpretation of **X1**, **X2**, and **X3** depends on whether the *cid* coordinate system is rectangular, cylindrical, or spherical.

Intrinsic Function: **COORD**

Purpose:

To retrieve the current value of a geometric coordinate.

Usage:

```
COORD ( { gid                  } , { X1 } [ , cid ] )
      ( { GRIDLIST ( grid_sid ) }   { X2 }
                                    { X3 }
```

Function Arguments:

gid	Identification of a grid specified in the Bulk Data Packet.
grid_sid	Set identification of a **GRIDLIST** bulk data entry used to specify the grid.
Xi	Component for the geometric coordinate.
cid	Identification of a coordinate system specified in the Bulk Data Packet.

Notes:

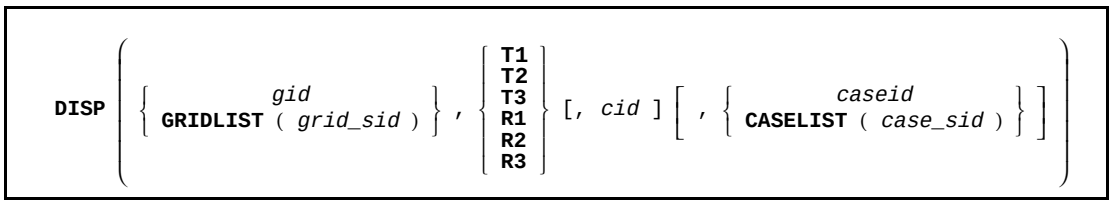
1. If the *cid* reference is omitted, then the coordinate value is returned in the ***input coordinate system*** of the GRID point.
2. A *cid* of 0 requests that the coordinate be returned in the basic coordinate system.
3. The interpretation of **X1**, **X2**, and **X3** depends on whether the *cid* coordinate system is rectangular, cylindrical, or spherical.

Intrinsic Function: **DISP**

Purpose:

To retrieve the current value of a displacement.

Usage:



Function Arguments:

gid	Identification of a grid point specified in the Bulk Data Packet.
grid_sid	Set identification of a **GRIDLIST** bulk data entry used to specify the grid.
Ti,Ri	Displacement component to recover.
cid	Identification of a coordinate system specified in the Bulk Data Packet.
caseid	Identification of a subcase.
case_sid	Set identification of a **CASELIST** bulk data entry used to specify the subcase identification number.

Notes:

1. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.
2. If the *cid* reference is omitted, then the coordinate value is returned in the output coordinate system of the grid points.
3. A *cid* of 0 requests that the coordinate be returned in the basic coordinate system.

Intrinsic Response Function: **DV**

Purpose:

To retrieve the current value of a design variable.

Usage:

```
DV ( { dvid                    } )
   ( { GDVLIST ( gdv_sid ) }   )
```

Function Arguments:

dvid	Identification of a design variable specified in the Bulk Data packet.
gdv_sid	Set identification of a **GDVLIST** Bulk Data entry used to specify the design variable.

Intrinsic Function: **FDAMP**

Purpose:

To retrieve the current value of flutter damping.

Usage:

FDAMP ({ **GAMMA** / **ZETA** } [,*machop*][,*densop*][,*modeop*][,*velop*][,*caseop*])

where:

machop => { *mvalue* / **MACHLIST** (*mach_sid*) }

densop => { *dvalue* / **DENSLIST** (*dens_sid*) }

modeop => { *modeid* / **MODELIST** (*mode_sid*) }

velop => { *vvalue* / **VELOLIST** (*vel_sid*) }

caseop => { *caseid* / **CASELIST** (*case_sid*) }

Fucntion Arguments:

mvalue	Mach value
mach_sid	Set identification of a **MACHLIST** bulk data entry used to specify the mach value.
dvalue	Density ratio value,
dens_sid	Set identification of a **DENSLIST** bulk data entry used to specify the density ratio value.
modeid	Mode index.
mode_sid	Set identification of a **MODELIST** bulk data entry used to specify the mode index.
vvalue	Velocity value.
vel_sid	Set identification of a **VELOLIST** bulk data entry used to specify the velocity value.

caseid Subcase identification.

case_sid Set identification of a **CASELIST** bulk data entry used to specify the subcase number.

Notes:

1. When the component **GAMMA** is specified the following equation is used.

$$\gamma = \begin{cases} \dfrac{\mathrm{Re}(p)}{\mathrm{Im}(p)} & ; \text{ for complex } p \\ \dfrac{\mathrm{Re}(p)}{\ln 2} & ; \text{ for real } p \end{cases}$$

When the component **ZETA** is specified the following equation is used.

$$\zeta = \left(\left(\frac{\mathrm{Re}(p)}{\mathrm{Im}(p)} \right)^2 + \mathrm{Re}(p)^2 \right)^{1/2} \quad ; \text{for complex } p$$

2. The specific discipline request defines whether the case and/or mode is a valid request in the response functions.

3. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **FFREQ**

Purpose:

To retrieve the current value of the flutter frequency.

Usage:

FFREQ (*machop* [,*densop*][,*modeop*][,*velop*][,*caseop*])

where:

machop => { *mvalue* | **MACHLIST** (*mach_sid*) }

densop => { *dvalue* | **DENSLIST** (*dens_sid*) }

modeop => { *modeid* | **MODELIST** (*mode_sid*) }

velop => { *vvalue* | **VELOLIST** (*vel_sid*) }

caseop => { *caseid* | **CASELIST** (*case_sid*) }

Function Arguments:

mvalue	Mach value
mach_sid	Set identification of a **MACHLIST** bulk data entry used to specify the mach value.
dvalue	Density ratio value,
dens_sid	Set identification of a **DENSLIST** bulk data entry used to specify the density ratio value.
modeid	Mode index.
mode_sid	Set identification of a **MODELIST** bulk data entry used to specify the mode index.
vvalue	Velocity value.
vel_sid	Set identification of a **VELOLIST** bulk data entry used to specify the velocity value.
caseid	Subcase identification.

case_sid Set identification of a **CASELIST** bulk data entry used to specify the subcase number.

Notes:

1. The frequency is returned in Radians. Conversion to Hertz may be accomplished by using the **HERTZ** intrinsic function.
2. The specific discipline request defines whether the case and/or mode is a valid request in the response functions.
3. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: FLEXCF

Purpose:

To retrieve flexible stability coefficients for a specific trim parameter from a Static Aerodynamics analysis.

Usage:

FLEXCF (*axis* , *trim_param* [, { *caseid* | **CASELIST** (*case_sid*) }])

Function Arguments:

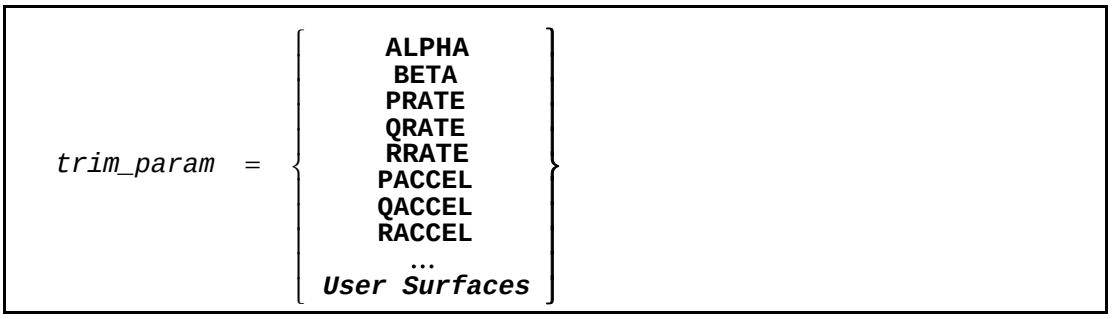
axis	Input axis.
param	Trim parameters.
caseid	Subcase identification.
case_sid	Set identification of a **CASELIST** bulk data entry used to specify the subcase number.

Notes:

1. This function returns its results in radians. If degrees are required, the results may be converted using the **DEGS** intrinisic function.
2. The allowable values for *axis* are:

axis = { **DRAG** | **SIDE** | **LIFT** | **ROLL** | **PITCH** | **YAW** }

3. The allowable control surfaces, *trim_param*, are:



The ***User Surfaces*** are defined using **AESURF** Bulk Data entries.

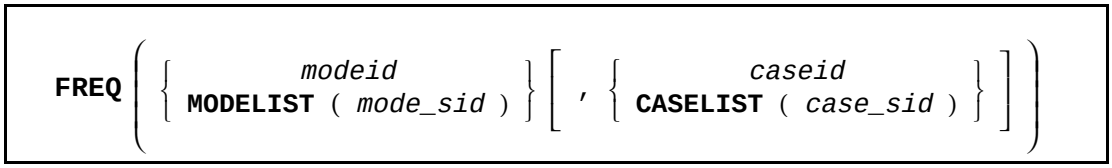
4. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **FREQ**

Purpose:

To retrieve the current value of the natural frequency computed in a Normal Modes analysis.

Usage:



Function Arguments:

modeid	Identification of a mode index.
mode_sid	Set identification of a **MODELIST** bulk data entry used to specify the mode index.
caseid	Subcase identification.
case_sid	Set identification of a **CASELIST** bulk data entry used to specify the subcase identification number.

Notes:

1. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **FROOT**

Purpose:

To retrieve the current value of the flutter root: $p = k(\gamma + i)$

Usage:

FROOT (*machop* [,*densop*][,*modeop*][,*velop*][,*caseop*])

where:

machop => { *mvalue* | **MACHLIST** (*mach_sid*) }

densop => { *dvalue* | **DENSLIST** (*dens_sid*) }

modeop => { *modeid* | **MODELIST** (*mode_sid*) }

velop => { *vvalue* | **VELOLIST** (*vel_sid*) }

caseop => { *caseid* | **CASELIST** (*case_sid*) }

Function Arguments:

mvalue	Mach value
mach_sid	Set identification of a **MACHLIST** bulk data entry used to specify the mach value.
dvalue	Density ratio value.
dens_sid	Set identification of a **DENSLIST** bulk data entry used to specify the density ratio value.
modeid	Mode index.
mode_sid	Set identification of a **MODELIST** bulk data entry used to specify the mode index.
vvalue	Velocity value.
vel_sid	Set identification of a **VELOLIST** bulk data entry used to specify the velocity value.
caseid	Subcase identification.

case_sid Set identification of a **CASELIST** bulk data entry used to specify the subcase number.

Notes:

1. The specific discipline request defines whether the case and/or mode is a valid request in the response functions.
2. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.
3. The function returns the Real part of the flutter root. If the Imaginary part is required, then the **IMAG** intrinsic function must be used.

Intrinsic Function: **MASS**

Purpose:

To return the mass of selected elements.

Usage:

MASS ({ *eid* / **ELEMLIST** (*elem_sid*) } [{ *plyid* / ' **PLYLIST** (*ply_sid*) }])

Function Arguments:

eid	Identification of an element specified in the Bulk Data Packet.
elem_sid	Set identification of an **ELEMLIST** bulk data entry used to specify an element.
plyid	Identification of a layer number for a composite element.
ply_sid	Set identification of a **PLYLIST** bulk data entry used to specify the layer number for a composite element.

Notes:

1. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used.
2. Composite elements must have their layer number specified.

Intrinsic Function: RIGIDCF

Purpose:

To retrieve rigid stability coefficients for a specific trim parameter from a Static Aerodynamics analysis.

Usage:

RIGIDCF (*axis* , *trim_param* [, { *caseid* | **CASELIST** (*case_sid*) }])

Function Arguments:

axis	Input axis.
trim_param	Trim parameters.
caseid	Subcase identification.
sid	Set identification of a **CASELIST** bulk data entry used to specify the subcase number.

Notes:

1. This function returns its results in radians. If degrees are required, the results may be converted using the **DEGS** intrinisic function.
2. The allowable values for *axis* are:

axis = { **DRAG** | **SIDE** | **LIFT** | **ROLL** | **PITCH** | **YAW** }

3. The allowable control surfaces, *trim_param*, are:

trim_param = { **ALPHA** | **BETA** | **PRATE** | **QRATE** | **RRATE** | **PACCEL** | **QACCEL** | **RACCEL** | ... | ***User Surfaces*** }

The ***User Surfaces*** are defined using **AESURF** Bulk Data entries.

4. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **STRAIN**

Purpose:

To retrieve current element STRAIN values.

Usage:

STRAIN (*elemop*,*strain_comp*[,*plyop*][,*caseop*][,*modeop*])

where:

elemop => { *eid* | **ELEMLIST** (*elem_sid*) }

plyop => { *plyid* | **PLYLIST** (*ply_sid*) }

caseop => { *caseid* | **CASELIST** (*case_sid*) }

modeop => { *modeid* | **MODELIST** (*mode_sid*) }

Function Arguments:

eid	Identification of an element specified in the Bulk Data Packet.
elem_sid	Set identification of an **ELEMLIST** bulk data entry used to specify an element.
strain_comp	Element response component.
plyid	Identification of a layer number for a composite element.
ply_sid	Set identification of a **PLYLIST** bulk data entry used to specify the layer number for a composite element.
caseid	Subcase identification.
case_sid	Set identification of a **CASELIST** bulk data entry used to specify the subcase number.
modeid	Identification of a mode index.
mode_sid	Set identification of a **MODELIST** bulk data entry used to specify the mode index.

Notes:

1. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used.
2. The allowable response components for each element type are shown in Table 20.
3. Composite elements must have their layer identification number specified.
4. Strain components will always be recovered at the center of the layer for composite elements.
5. The specific discipline request defines whether the case and/or mode is a valid request in the response functions.
6. The mode sequence number is used only if the discipline is MODES.
7. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **STRESS**

Purpose:

To retrieve current element STRESS values.

Usage:

STRESS (*elemop*,*stress_comp*[,*plyop*][,*caseop*][,*modeop*])

where:

elemop => { *eid* | **ELEMLIST** (*elem_sid*) }

plyop => { *plyid* | **PLYLIST** (*ply_sid*) }

caseop => { *caseid* | **CASELIST** (*case_sid*) }

modeop => { *modeid* | **MODELIST** (*mode_sid*) }

Function Arguments:

eid	Identification of an element specified in the Bulk Data Packet.
elem_sid	Set identification of an **ELEMLIST** bulk data entry used to specify an element.
stress_comp	Element response component.
plyid	Identification of a layer number for a composite element.
ply_sid	Set identification of a **PLYLIST** bulk data entry used to specify the layer number for a composite element.
caseid	Subcase identification.
case_sid	Set identification of a **CASELIST** bulk data entry used to specify the subcase number.
modeid	Identification of a mode index.
mode_sid	Set identification of a **MODELIST** bulk data entry used to specify the mode index.

Notes:

1. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used.
2. The allowable response components for each element type are shown in Table 20.
3. Composite elements must have their layer identification number specified.
4. Stress components will always be recovered at the center of the layer for composite elements.
5. The specific discipline request defines whether the case and/or mode is a valid request in the response functions.
6. The mode sequence number is used only if the discipline is MODES.
7. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **THICK**

Purpose:

To return the thickness of the requested two-dimensional elements.

Usage:

THICK ({ *eid* / **ELEMLIST** (*elem_sid*) } [{ *plyid* / ' **PLYLIST** (*ply_sid*) }])

Function Arguments:

eid	Identification of an element specified in the Bulk Data Packet.
elem_sid	Set identification of an **ELEMLIST** bulk data entry used to specify an element.
plyid	Identification of a layer number for a composite element.
ply_sid	Set identification of a **PLYLIST** bulk data entry used to specify the layer number for a composite element.

Notes:

1. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used.
2. Composite elements must have their layer number specified.

Intrinsic Function: **TRIM**

Purpose:

To retrieve trim parameters from a Static Aerodynamics analysis.

Usage:

TRIM (*trim_param* [, { *caseid* | **CASELIST** (*case_sid*) }])

Function Arguments:

trim_param Trim parameters.

caseid Subcase identification.

case_sid Set identification of a **CASELIST** bulk data entry used to specify the subcase number.

Notes:

1. This function return its results in radians. If degrees are required, the results may be converted using the **DEGS** intrinisic function.
2. The allowable control surfaces, *trim_param,* are:

   trim_param = { **ALPHA** | **BETA** | **PRATE** | **QRATE** | **RRATE** | **PACCEL** | **QACCEL** | **RACCEL** | ... | ***User Surfaces*** }

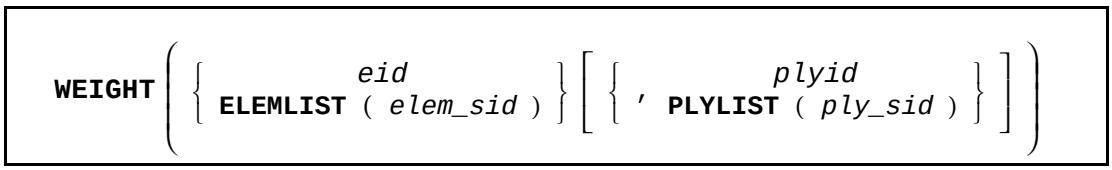
   The ***User Surfaces*** are defined using **AESURF** Bulk Data entries.
3. If the subcase reference is omitted, then the specific discipline request defines the requested subcase.

Intrinsic Function: **WEIGHT**

Purpose:

To return the weight of selected elements.

Usage:



Function Arguments:

eid	Identification of an element specified in the Bulk Data Packet.
elem_sid	Set identification of an **ELEMLIST** bulk data entry used to specify an element.
plyid	Identification of a layer number for a composite element.
ply_sid	Set identification of a **PLYLIST** bulk data entry used to specify the layer number for a composite element.

Notes:

1. When an element identification is used then the *eid* must be unique and if the *eid* is not unique, then an element list must be used.
2. Composite elements must have their layer number specified.

This page is intentionally blank.

Chapter 7.

THE BULK DATA PACKET

The bulk data packet provides the ASTROS system with the engineering data needed to perform the specific tasks requested by the user. It contains the model geometries for the structural model, the aerodynamic model(s) and the design model as well as the pool of data from which the solution control requests are made. Finally, specialized information required by the analysis disciplines (e.g., Mach number and reduced frequency pairs for unsteady aerodynamic analyses) is also provided to the system through the bulk data packet. The basic input item is the bulk data entry which is directly analogous to the NASTRAN bulk data *card*. In fact, NASTRAN compatible formats were chosen for the ASTROS bulk data entries whenever possible because modern structures are often analyzed using large NASTRAN finite element models having tens of thousands of lines of bulk data. Further, these large models are usually prepared using software designed specifically to generate NASTRAN models. Thus, by utilizing NASTRAN bulk data structures where possible and by using the NASTRAN bulk data style for the additional engineering data, ASTROS is highly compatible with existing NASTRAN models and with current finite element model generation methods.

Just as in NASTRAN, the bulk data packet begins with the keyword **BEGIN BULK** (which may be abbreviated **BEGIN**) and is terminated by the optional keyword **ENDDATA** or by the end of the input stream. The intervening bulk data entries can appear in any order. An alphabetically sorted listing of the bulk data input will be echoed to the output file unless suppressed by the user through the **BEGIN BULK** command line options.

All the input entries are interpreted by **IFP** through ***templates*** that are defined as part of the system generation task. The templates provide for basic error checking, establish defaults and direct the placement of the raw data onto the database. The use of templates allows additional entries to be added to the system very simply without software changes. The definition of the templates and the means of adding new entries are documented in the Programmer's Manual. In addition, the complete listing of ASTROS

bulk data templates is included in the output summary generated by the SYSGEN system generation utility during the creation of the system database files.

On restart with a bulk data packet in the input stream, the **IFP** module will append the new data onto the data from the previous run(s). There is no provision for deleting existing bulk data except through MAPOL sequence modifications or direct interaction using the ICE program (Reference 7). This restart feature, while limited, can be useful in many instances; e.g. when additional analysis disciplines are desired or when different output requests are desired. The remainder of this section presents the structure of the bulk data entry for ASTROS and discusses some features of the **IFP** module that are useful to the general user. ASTROS bulk data entries have been carefully designed to be NASTRAN compatible, so the NASTRAN User's Manual (Reference 2) has provided much of the information in the following discussion as well as having directed the design of the **IFP** software. The reader is also referred to the ASTROS Programmer's Manual for more information on the **IFP** module and for information on the addition of new bulk data entries.

7.1. BULK DATA ECHO OPTIONS

There are special options on the **BEGIN BULK** command which allow the user to control the echoing of the Bulk Data. The format of this command is:

The following table describes the actions which are performed for the various options.

ECHO	FILE	ORDER	ACTION
ECHO	PRINT	SORT	Sorted echo to the output file.
		UNSORT	Unsorted echo to the output file.
	PUNCH	SORT	Sorted echo to punch file.
		UNSORT	Unsorted echo to punch file.
	BOTH	SORT	Sorted echo to both output and punch files.
		UNSORT	Unsorted echo to both output and punch files.
NOECHO			No echo to either output or punch file.

7.2. FORMAT OF THE BULK DATA ENTRY

Each bulk data entry consists of a required ***parent*** line followed by a number of optional ***continuation*** lines. Therefore, a single bulk data entry resides on one or more lines. The basic bulk data line has one mnemonic field of eight characters followed by either eight data fields of eight characters or by four data fields of 16 characters and terminates with an eight character continuation field as shown in Figure 7-1. The data field size (either eight or 16 characters) is determined by the presence of the optional ***large field marker*** in the first mnemonic field of each bulk data line. The parent line begins with a character mnemonic identifying the entry followed by 4 or 8 data fields and ending with a continuation field. The continuation lines are identical except that the leading mnemonic field contains a continuation label which is used to link it to its parent line. This structure is identical to that in NASTRAN. One important exception to NASTRAN compatibility is that ASTROS requires that the continuation lines follow continuously from the parent line although the bulk data entries themselves can be in any order. Random placement of continuations in NASTRAN is an artifact from using physical cards that were punched with the bulk data. If the card deck were dropped, the resulting random order still had to be interpretable by the code. This feature no longer needs to be supported in light of modern computer storage methods but NASTRAN compatibility dictated that similar continuation labeling be used.

A continuation line is defined for a bulk data entry that requires more than eight (or four large) data fields. The last field of the parent line is used in conjunction with the first field of the continuation line as

Small Field Entry with a Small Field Continuation

NAME									ABC
+BC									

Small Field Entry with a Large Field Continuation

NAME									ABC
*BC									

Large Field Entry with both Large and Small Field Continuations

NAME*									ABC
*BC									DEF
+EF									

Large Field Entry with a Large Field Continuation

NAME*					ABC
*BC					

Figure 7-1. Bulk Data Entry Formats

an identifier. The parent continuation field can contain any alphanumeric entry while the first field of the continuation line contains a plus (**+**) as a continuation character in column 1 followed by the last 7 characters from the parent continuation label. For the parent line, the large field marker is an asterisk (*****) following the name of the entry which signifies that large data fields are to be used. For continuation lines, the asterisk used as the continuation character plays the role of the large field marker as shown below. Each bulk data line must be either all narrow field or all large field, although separate lines of a single bulk data entry can have different field widths simply by using the proper field marker. This means that the same bulk data entry in wide and narrow formats are functionally identical with no need for separate templates. Unlike NASTRAN, the continuation mnemonics need not be unique among all the bulk data entries in the bulk data packet since there is no provision for randomly sorted continuations.

The input on a bulk data line can either be in fixed format, in which each item must reside within the field to which it belongs, or in free format, in which fields are separated by commas and can be positioned anywhere to the left of the column in which the fixed field would normally start. Free format input is indicated by the appearance of a comma in the first 10 characters of the input line. ASTROS requires that each line (not each bulk data entry) be either all fixed or all free format and that each free format field be separated by a comma. The NASTRAN use of a blank character as a field separator is not supported. When free format input is used, the continuation lines can reside on the same physical line of input with the continuation labels either included or not as in the following equivalent examples:

```
MKAERO1, 1, , 0.3, 0.5, , , , , ABC , +BC, 0.01, 0.05, 0.1, 0.2
```

```
MKAERO1, 1, , 0.3, 0.5, , , , , 0.01, 0.05, 0.1, 0.2
```

In the latter case, ASTROS will automatically generate the missing continuation mnemonics. Care must be taken, however, that the first ***two*** data fields of the continuation line be non-blank. If not, there is an ambiguity as to whether the first continuation field constitutes a continuation label or a data field. This ambiguity causes the **IFP** to terminate execution with an error indicating that there is a missing continuation line. Free format input in which the parent and continuation lines are broken into separate physical lines or which explicitly include the continuation mnemonics do not suffer this limitation. Free format input is further restricted in that the break between physical lines, if needed, must occur at a break in the logical line, that is, the split must occur between the ending continuation field on the current logical line and the continuation field of the next logical line. This means, for the preceding example, that the first example entry could be broken into two lines between the **ABC** and **+BC** fields but nowhere else. When an entry is broken into multiple physical lines, the continuation mnemonics must be supplied. Obviously, fixed format input requires continuation mnemonics for any bulk data entries having continuation lines.

7.3. DATA FIELD FORMATS

The interior fields of a bulk data line can contain either integer data, real data, character data or certain combinations (e.g. either integer or real data). The template for each entry defines which types of data are acceptable in each field. Each data item is limited to the number of characters that fit in the length of the field. For narrow width fields no more than eight characters can be used in the data item. Unlike NASTRAN, any extra characters will spill to the next field and will result in **IFP** errors, there is no provision for rounding real data to fit the field size.

In order to be considered valid, the data item must first satisfy the data type requirement as specified on the template. Real numbers, including zero, must contain a decimal point, although there are a number of formats supported. For example, the real number **3.1** may be encoded as shown or as **3.1E0**, **+3.1D00**, **0.31E1**, or **3.1+0**. Unlike NASTRAN, however, there ***cannot*** be embedded blanks anywhere in the real number and a **D** edit descriptor is treated as a single precision number until actually loaded to a double precision relational attribute. Blank fields that do not have other defaults specified on the template, will be interpreted as blank characters, an integer zero or a real zero as required. Integer values must be formed from the ten decimal digits with an optional leading plus or minus sign. Character data consist of any combination of alphanumeric characters including any digits, decimal points, etc., with no restriction that the first character be alphabetic.

7.4. ERROR CHECKING IN THE INPUT FILE PROCESSOR

As mentioned in the preceding subsection, the **IFP** module performs basic error checking to ensure that the input data is of the correct type. In addition, the templates provide for error checks that enable the **IFP** to check that the data satisfy particular requirements. For example, the **IFP** can be directed to require that a particular value be greater than zero or be one of a finite number of selections. At its most complex, the bulk data processor checks to ensure specific relationships among data on a single bulk data entry. It is important to understand, however, that no error checks occur in the **IFP** to ensure that references to, and interrelationships among, multiple bulk data entries are satisfied. These more complex checks occur in subsequent engineering modules. A complete description of the available template error checks and the mechanism provided to add additional error checks is presented in the Programmer's Manual. The reader may find it helpful to study this documentation since the bulk data packet and the bulk data entries are closely linked to the software in both the SYSGEN utility and the **IFP** module.

7.5. BULK DATA ENTRY SUMMARY

This section contains a summary of all the bulk data entries in the ASTROS system separated into logically related groups. The groups are composed of either model definition entries, subcase definition entries or general list entries. This is followed by a detailed description of each of the entries listed in this section. Section 7.6 discusses the differences between NASTRAN and ASTROS for those entries that have been changed or are completely different than in NASTRAN but that use the same mnemonic and serve a similar purpose. Entries indicated by * are unchanged from NASTRAN.

7.5.1. Aerodynamic Load Transfer

ATTACH	Rigid load transfer definition.
SET1*	A structural grid point list for spline interpolation or a mode list for omitting normal modes in flutter analysis.
SET2*	Structural grid point list in term of aerodynamic macroelements.
SPLINE1*	Surface spline definition for out-of-plane motion.
SPLINE2*	Beam spline definition for interpolating panels and bodies.

7.5.2. Applied Dynamic Loads

DLAGS	Time and phase lag definition for a spatial load.
DLOAD*	Linear combination of dynamic load sets.
DLONLY	Direct definition of dynamic spatial load.
GUST	Stationary vertical gust definition.
RLOAD1	Frequency dependent dynamic load definition.
RLOAD2	Frequency dependent dynamic load definition.
TABLED1	Tabular function definition for dynamic load generation.
TLOAD1	Time dependent dynamic load definition.
TLOAD2	Time dependent dynamic load definition.

7.5.3. Applied Static Loads

FORCE*	Definition of a concentrated load at a grid point.
FORCE1*	Definition of a concentrated load at a grid point.
GRAV*	Definition of an acceleration vector for gravity loads.
LOAD*	Definition of linear load combinations.
MOMENT*	Definition of a moment at a grid point.
MOMENT1*	Definition of a moment at a grid point.
PLOAD*	Definition of a pressure load over an area.
PLOAD2*	Definition of a pressure load on plate elements.
PLOAD4*	Definition of a pressure load on plate elements in a specified direction.
TEMP*	Definition of a temperature at a structural node.
TEMPD*	Definition of default nodal temperatures.

7.5.4. Boundary Condition Constraints

ASET*	Analysis set definition.
ASET1*	Analysis set definition.
DYNRED	Dynamic reduction parameters.
JSET	Inertia relief mode shape parameter definition.
JSET1	Inertia relief mode shape parameter definition.
MPC*	Multipoint constraint definition.
MPCADD*	Definition of combinations of **MPC** sets.
OMIT	Omit set definition.
OMIT1	Omit set definition.
RBAR	Rigid bar element
RBE1	Rigid body element
RBE2	Rigid body element
RBE3	Rigid body element
RROD	Rigid rod element

SPC*	Single point constraint/enforced displacement definition.
SPC1*	Single point constraint definition.
SPCADD*	Definition of combinations of SPC sets.
SUPORT	Definition of coordinates for determinate reactions.

7.5.5. Design Constraints

DCONALE	Aileron effectiveness constraint definition.
DCONBK	Buckling constraint definition.
DCONBKE	Euler buckling constraint definition.
DCONCLA	Lift effectiveness constraint definition.
DCONDSP	Displacement constraint definition.
DCONEP	Principal strain constraint definition.
DCONEPM	Principal strain constraint definition.
DCONEPP	Principal strain constraint definition.
DCONF	Functional constraint definition.
DCONFLT	Flutter constraint definition.
DCONFRQ	Modal frequency constraint definition.
DCONFT	Fiber/transverse strain constraint definition.
DCONFTM	Fiber/transverse strain constraint definition.
DCONFTP	Fiber/transverse strain constraint definition.
DCONLAM	Composite laminate constraint definition.
DCONLMN	Composite laminate minimum gauge constraint definition.
DCONPMN	Composite element ply minimum gauge constraint definition.
DCONSCF	Flexible stability coefficient constraint definition.
DCONSDE	BAR element cross-sectional parameter side constraint definition.
DCONSDL	BAR element cross-sectional parameter side constraint definition.
DCONTH2	Composite layer thickness constraint definition for shape linking.
DCONTH3	BAR element cross-sectional parameter definition for shpae linking.
DCONTHK	Thickness constraint definition for use with shape function design variable linking.
DCONTRM	Aeroelastic trim parameter constraint definition.
DCONTW	Tsai-Wu stress constraint definition.
DCONTWM	Tsai-Wu stress constraint definition.
DCONTWP	Tsai-Wu stress constraint definition.
DCONVM	Von-Mises stress constraint definition.
DCONVMM	Von-Mises stress constraint definition.
DCONVMP	Von-Mises stress constraint definition.

7.5.6. Design Variables, Linking and Optimization Parameters

DESELM	Unique physical design variable definition.
DESVARP	Linked physical design variable definition
DESVARS	Linked shape function design variable definition.
DVTOPTE	Thickness variation type definition for bending plate element design.
DVTOPTL	Thickness variation type definition for an element list.
DVTOPTP	Thickness variation type definition based on element properties.
ELIST	Element list for physical linking.
ELISTM	Element list for physical linking of different local design variables.
MPPARM	Mathematical programming default parameter override.
PLIST	Physical design variable linking definition.
PLISTM	Physical design variable linking of different local design variables.
SHAPE	Definition of element linking factors to define a shape variable.
SHAPEM	Definition of element linking factors of different local design variables to define a shape variable.
SHPGEN	Definition of design variables using the SHAPE generation utility.

7.5.7. Geometry

CORD1C*	Cylindrical coordinate system definition.
CORD1R*	Rectangular coordinate system definition.
CORD1S*	Spherical coordinate system definition.
CORD2C*	Cylindrical coordinate system definition.
CORD2R*	Rectangular coordinate system definition.
CORD2S*	Spherical coordinate system definition.
EPOINT*	Extra point definition for dynamics.
GRDSET*	Default parameters for fields on the GRID entry.
GRID*	Grid point location and coordinate system selection.
SPOINT*	Scalar point definition.

7.5.8. Material Properties

MAT1*	Isotropic elastic properties definition.
MAT2*	Two-dimensional anisotropic properties definition.
MAT8*	Orthotropic properties definition.
MAT9*	Anisotropic properties definition for isoparametric hexahedral elements.

7.5.9. Miscellaneous Inputs

$*	Commentary data.
CONVERT	Conversion factor definitions.
DMI	Direct matrix input.

DMIG	Direct matrix input at structural nodes.
MFORM	Mass matrix form (LUMPED or COUPLED).
SAVE	List of database entities not to be purged.
SEQGP*	Structural set resequencing definition.

7.5.10. Selection Lists

CASELIST	List of subcase identification numbers.
DCONLIST	List of design constraint identification numbers.
DENSLIST	List of density ratio values.
ELEMLIST	List of element identification numbers.
FREQLIST	List of frequency step values.
GDVLIST	List of global design variable identification numbers.
GPWG	Definition of the location to perform grid point weight generation
GRIDLIST	List of GRID point identification numbers.
ITERLIST	List of iteration step identification numbers.
LDVLIST	List of local design variable identification numbers.
MACHLIST	List of Mach number values.
MODELIST	List of normal mode identification numbers.
PLYLIST	List of GRID point identification numbers.
TIMELIST	List of time step values.
VELOLIST	List of velocity values.

7.5.11. Steady Aerodynamics

AEROS	Reference parameters
AEFACT	List of real parameters.
AESURF	Aerodynamic control surface definition.
AIRFOIL	Airfoil property definition.
AXSTA	Body axial station parameter definition.
BODY	Body configuration definition.
CAERO6	Macroelement (panel) definition.
CONEFFS	Definition of static aerodnamic control effectiveness
CONLINK	Definition of linked control surfaces.
PAERO6	Body parameter definition.

7.5.12. Structural Element Connection

BAROR	Definition of default parameters for the CBAR bar element.
CBAR	Prismatic beam element.
CELAS1	Scalar elastic spring element.
CELAS2	Scalar elastic spring element.

CIHEX1*	Linear isoparametric hexahedral element.
CIHEX2*	Quadratic isoparametric hexahedral element.
CIHEX3*	Cubic isoparametric hexahedral element.
CMASS1	Scalar mass element.
CMASS2	Scalar mass element.
CONM1*	Direct 6 x 6 mass matrix definition at a structural node.
CONM2	Concentrated mass at a structural node.
CONROD	Rod element.
CQDMEM1	Isoparametric quadrilateral membrane element.
CQUAD4	Isoparametric quadrilateral element with bending and membrane stiffness.
CROD	Rod element.
CSHEAR	Shear panel.
CTRIA3	Isoparametric triangular element with bending and membrane stiffness.
CTRMEM	Constant strain triangular membrane element.
GENEL*	General element.

7.5.13. Structural Element Properties

PBAR	Prismatic beam element.
PBAR1	Prosmatic beam element defined with standard cross-sectional parameters.
PCOMP	Composite laminate definition for **CQDMEM1**, **CQUAD4**, **CTRIA3**, and **CTRMEM** elements.
PCOMP1	Composite laminate definition for **CQUAD4** and **CTRIA3** elements.
PCOMP2	Composite laminate definition for **CQUAD4** and **CTRIA3** elements.
PELAS	Scalar elastic spring element.
PIHEX*	Linear, quadratic and cubic isoparametric hexahedral element.
PMASS	Scalar mass element
PQDMEM1	Isoparametric quadrilateral membrane element.
PROD	Rod element.
PSHEAR	Shear panel.
PSHELL	Definition of shell element properties for **CQUAD4** and **CTRIA3** elements.
PTRMEM	Constant strain triangular membrane element.

7.5.14. Unsteady Aerodynamics

AERO	Reference parameters
CAERO1*	Aerodynamic macroelement (panel) definition.
CAERO2*	Body configuration definition.
CONEFFF	Definition of flutter aerodynamic control effectiveness
FLFACT*	Parameter definition for flutter analysis.

MKAERO1	Table of symmetries, Mach numbers, and reduced frequencies.
MKAERO2	Table of symmetries, Mach numbers, and reduced frequencies.
PAERO1*	Association between bodies and macroelements.
PAERO2*	Body cross-section property definition.

7.5.15. Discipline Dependent Problem Control

The following bulk data entries are the controlling entries referenced by Solution Control in selecting specific disciplines and subcases. In each case, many of these inputs can appear in the bulk data packet with the particular input to be used for the subcase referenced in the Solution Control Packet.

FLUTTER	Basic parameters for flutter analyses.
TRIM	Flight condition for steady aeroelastic trim analyses.
EIGC	Complex eigenvalue extraction parameters
EIGR*	Real eigenvalue extraction parameters.
FFT	Fast Fourier Transform parameter definition.
FREQ*	Frequency step definition for frequency response.
FREQ1*	Frequency step definition for frequency response.
FREQ2*	Frequency step definition for frequency response.
IC	Initial condition definition for direct transient response (same as NASTRAN **TIC** entry).
TABDMP1	Modal damping table for modal dynamic response.
TF*	Dynamic transfer function definition.
TSTEP*	Time step definition for transient response
VSDAMP	Definition of viscous damping based on equivalent structural damping.

7.6. DIFFERENCES BETWEEN ASTROS AND NASTRAN BULK DATA

Some of the bulk data entries listed in the preceding Section do not exist in the NASTRAN versions that guided the definition of the bulk data entries. Some of them do exist in other NASTRAN systems, however; the **DYNRED, JSET, JSET1, PCOMP1**, and **PCOMP2** entries are examples. Others take the place of the NASTRAN **PARAM** entry which was felt to have been overused to the point where it had lost all utility. Examples of these inputs are the **CONVERT, MFORM** and **VSDAMP** entries. The steady aeroelastic model is completely new to ASTROS since NASTRAN uses the same modeling for both steady and unsteady analysis. Also, it was felt that the NASTRAN mechanism for defining dynamic loads was needlessly complicated. Working from the NASTRAN inputs, a simpler, but equally general set of entries was developed. This resulted in the generation of a number of new entries and the modification of others. The definition of the design variables, design variable linking and the design constraints is, of course, completely new for ASTROS.

The majority of the changed entries have been modified to accommodate the design task. In these cases, the bulk data entry is often identical to the NASTRAN version for use in analysis with optional additional fields to specify the design data. The element connectivity and property entries are all examples of this type of change in that additional field(s) have been added to specify the maximum and minimum

allowable physical design variable value if shape function design variable linking is used. In cases where data from NASTRAN preprocessors are used, there are no changes required unless shape function linking is desired.

A more subtle set of changes was required to perform multidisciplinary analysis. In NASTRAN, as was mentioned in the discussion of the Solution Control packet, many parameters were specified as part of the model definition or discipline specification because the code was limited to performing a single analysis of the given discipline. In order to remove these artificial restrictions, these data have been moved to the proper discipline's subcase definition. Examples of this form of modification are the addition of symmetry options to the **MKAEROi, GUST, FLUTTER**, and **TRIM** entries and the removal of subcase dependent data from the **AERO** entry. Further, the rigid elements, **ASETi, OMITi** and **EPOINT** entries were modified to include a set identification number to enable multiple boundary conditions and multiple control systems to be analyzed simultaneously.

The last type of modification came about because of the nature of the ASTROS database management system. These were limited to the **DMI** and **DMIG** entries for direct loading of database entities. The NASTRAN inputs were not compatible with the ASTROS database and so had to be modified. In fact, these entries, while having the same name as a NASTRAN entry, are completely new entries for ASTROS. A minor additional modification to the input definitions was made for the **TABDMP1** entry to make it more compact and to remove the spurious **ENDT** table termination symbol. In ASTROS, all tabular input entries are terminated when no more data appears and require no specific declaration of the table end.

While a seemingly large number of bulk data entries have been changed relative to their NASTRAN counterparts, in fact only a few have been changed in such a way that the NASTRAN version will not work in analysis. By far, the majority of the modeling bulk data entries are completely unchanged except for certain design variable linking options. In unsteady and steady aerodynamic disciplines care must be taken to account for the subcase dependencies that NASTRAN defined implicitly or with **PARAM** entries. Finally, the use of **ASET** and **OMIT** entries will cause minor problems in that ASTROS requires a set identification for these entries. While this latter restriction can require some effort to fix, the gain in capability simply required that the bulk data entry be modified.

The most serious potential problem using NASTRAN models in ASTROS is that the set of bulk data entries is more limited in ASTROS than in NASTRAN. The ASTROS system has been developed primarily as a multidisciplinary preliminary design tool and does not yet contain the wide range of options supported by a mature code like NASTRAN. The many NASTRAN input entries supporting these options, therefore, have not been defined to the ASTROS system because they are not supported by any ASTROS code. Thus, there will be instances where a NASTRAN input deck will have to be modified to remove these entries which serve no purpose in ASTROS. The majority of these bulk data entries deal with unsupported elements, plotting options, output options, etc., which are not felt to present a major problem. More important is the support for NASTRAN's model definitions, most of which have already been adopted by ASTROS.

7.7. BULK DATA DESCRIPTIONS

This Section contains a complete description of each of the ASTROS Bulk Data entries.

This page is intentionally blank.

Input Data Entry: **$** Comment

Description: Allows commentary text to be inserted into the unsorted echo of the input Bulk Data Deck. The **$** entry is otherwise ignored by the program. These entries do not appear in a sorted echo.

Format and Example:

```
   1       2       3       4       5       6       7       8       9       10
$ Followed by any legitimate characters in columns 2-80
```

```
$ THIS (*,,"$$)--/
```

Remarks:

1. The comment entry may also be used in the Solution Control packet.

Input Data Entry: **AEFACT** Aerodynamic Lists

Description: Used to specify lists of real numbers for aeroelastic analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
AEFACT	SID	D1	D2	D3	D4	D5	D6	D7	CONT
CONT	D8	D9	-etc-						

AEFACT	97	0.3	0.7	1.0					

Field	Contents
SID	Set identification number (Unique Integer > 0).
Di	Number (Real).

Remarks:

1. These factors must be selected by an **AIRFOIL**, **AXSTA**, **CAEROi** or **PAEROi** data entry.
2. All contiguous **Di** fields must contain data — embedded blank fields are not allowed.
3. If this entry is used to specify division points, there will be one more division point than the number of divisions.

Input Data Entry: **AERO** Aerodynamic Physical Data

Description: Gives basic aerodynamic parameters for unsteady aerodynamic disciplines.

Format and Example:

1	2	3	4	5	6	7	8	9	10
AERO	ACSID	REFC	RHOREF						
AERO	100	300.0	1.1E-7						

Field	Contents
ACSID	Aerodynamic coordinate system identification (Integer ≥ 0 or Blank). See Remark 2.
REFC	Reference length (for reduced frequency) (Real ≥ 0).
RHOREF	Reference density (Real ≥ 0).

Remarks:

1. This entry is required for unsteady aerodynamic disciplines. Only one **AERO** entry is allowed.
2. The **ACSID** must be a rectangular coordinate system. Flow is in the positive x-direction. If blank, the basic coordinate system is used.

Input Data Entry: **AEROS** Steady Aero Physical Data

Description: Gives basic aerodynamic parameters for the steady aerodynamic discipline.

Format and Example:

1	2	3	4	5	6	7	8	9	10
AEROS	ACSID	RCSID	REFC	REFB	REFS	GREF	REFD	REFL	
AEROS	10	20	10.	100.	1000.	1			

Field	Contents
ACSID	Aerodynamic coordinate system identification (Integer > 0) or blank. See Remark 2.
RCSID	Reference coordinate system identification for rigid body motions. (Integer > 0, or blank)
REFC	Reference chord length (Real > 0.0) (D = 1.0)
REFB	Reference span (Real > 0.0) (D = 1.0)
REFS	Reference wing area (Real > 0.0) (D = 1.0)
GREF	Reference grid point for stability derivative calculations (Integer > 0).
REFD	Fuselage reference diameter (Real > 0) or blank (D = 1.0)
REFL	Fuselage reference length (Real > 0) or blank (D = 1.0)

Remarks:

1. This entry is required for static aeroelasticity problems. Only one **AEROS** entry is allowed.
2. The **ACSID** must be a rectangular coordinate system. Flow is in the positive x-direction. If **ACSID** is blank, the Basic Coordinate system is used.
3. The **RCSID** must be a rectangular coordinate system. All degrees of freedom defining trim variables will be defined in this coordinate system. If **RCSID** is blank, the Basic Coordinate system is used.

Input Data Entry **AESURF** Aerodynamic Control Surface

Description: Specifies an Aerodynamic Control Surface.

Format and Example:

1	2	3	4	5	6	7	8	9	10
AESURF	LABEL	TYPE	ACID	CID	FBOXID	LBOXID			
AESURF	ELEV	SYM	6000		6010	6030			

Field	Contents
LABEL	Unique alphanumeric string of up to eight characters used to identify the control surface
TYPE	Surface type (Character) (Remark 2) **SYM** symmetric surface **ANTISYM** antisymmetric surface **ASYM** Asymmetric surface
ACID	Identification number of the aircraft component (**CAERO6**) on which the surface lies. (Integer > 0)
CID	Identification number of a rectangular coordinate system whose y-axis defines the hinge line of the control surface. (Integer > 0 or blank)
FBOXID	First aero box on the control surface relative to **ACID**. (Integer > 0)
LBOXID	Last aero box on the control surface relative to **ACID**. (Integer > 0)

Remarks:

1. The **LABEL** is arbitrary, but all labels must be unique.
2. The asymmetric surface, **TYPE=ASYM**, is not currently available. Pitch controllers are **TYPE=SYM** while yaw and roll controllers are **TYPE=ANTISYM**.
3. The aerodynamic box numbering scheme is illustrated on the **CAERO1** Bulk Data entry.

Input Data Entry **AIRFOIL** Airfoil Definition

Description: Defines airfoil properties for the static aerodynamic model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
AIRFOIL	ACID	CMPNT	CP	CHORD	USO/THK	LSO	CAM	RADIUS	CONT
CONT	X1	Y1	Z1	X12	IPANEL				

AIRFOIL	1	WING	1	10	20		30		ABC
+BC	0.0	0.0	0.0	50.0					

Field	Contents
ACID	Associated aircraft component identification number referenced by a matching **CAERO6** bulk data entry. (Integer > 0)
CMPNT	Type of aircraft component (Character) selected from: (See Remark 3) **WING** **FIN** **CANARD**
CP	Coordinate system for airfoil. (Integer > 0, or blank) (See Remark 4)
CHORD	Identification number of an **AEFACT** data entry containing a list of division points (in terms of percent chord) at which airfoil thickness and camber data are specified. (Integer > 0)
USO/THK	Identification number of an **AEFACT** data entry defining either the upper surface ordinates in percent chord if **LSO** is not blank, or the half thicknesses about the camber ordinates if **CAM** is not blank. (Integer > 0, or blank) (See Remark 3)
LSO	Identification number of an **AEFACT** data entry defining the lower surface ordinates in percent chord. Must be used in conjunction with **USO**. (Integer > 0, or blank) (See Remark 3)
CAM	Identification number of an **AEFACT** data entry defining the mean line (camber line) ordinates in percent chord. (Integer) (See Remark 3)
RADIUS	Radius of leading edge in percent chord. (Real ≥ 0.0)
X1,Y1,Z1	Location of the airfoil leading edge in coordinate system **CP**. (Real, **Y1** ≥ 0.0)
X12	Airfoil chord length in x-axis coordinate of system **CP**. (Real > 0 or blank)
IPANEL	Identification number of an **AEFACT** data entry containing a list of chord wise cuts in percent chord for wing paneling. (Integer > 0, or blank)

Remarks:

1. If the **RADIUS** field is blank, a round leading edge of radius zero is used.
2. **IPANEL** is optional and is used when different chord-wise cuts on each end of the panel are desired.

3.

For **WING** components, the options for **USO**, **LSO**, **THK** and **CAM** are:	
All Blank	Default flat plat airfoil generated automatically
USO alone	Lower and upper surface ordinates of airfoil are defined with effectively **LSO**=**USO** internally generated
USO/**LSO**	Lower and upper surface ordinates of airfoil are defined; **CAM** ***Must*** be blank
THK/**CAM**	Half thicknesses about the camber line are defined. **LSO** ***Must*** be blank
USO/**LSO**/**CAM**	***Illegal*** over-specification of data
LSO/**CAM**	***Illegal***, must use **THK** field for half thickness
CAM alone	***Illegal*** under-specification of data
For **CANARD** components, the options are as above except that camber is not allowed so **CAM** ***Must*** be blank	
All Blank	Default flat plat airfoil generated automatically
USO alone	Lower and upper surface ordinates of airfoil are defined with effectively **LSO**=**USO** internally generated
USO/**LSO**	Lower and upper surface ordinates of airfoil are defined; **CAM** ***Must*** be blank
THK/**CAM**	***Illegal*** specification, **CAM** must be blank
USO/**LSO**/**CAM**	***Illegal*** over-specification of data
LSO/**CAM**	***Illegal***, **CAM** must be blank
CAM alone	***Illegal*** under-specification of data and **CAM** must be blank for CANARD
For **FIN** components, the options are very limited: only symmetric airfoils are allowed and they must be entered as an upper surface ordinate (the lower surface ordinates are then defaulted)	
All Blank	Default flat plat airfoil generated automatically
USO alone	Lower and upper surface ordinates of airfoil are defined with effectively **LSO**=**USO** internally generated. ***Only Legal Nonblank Fin Option***

4. The basic coordinate system must be used (CP blank). This field exists to allow the addition of user defined coordinate systems in the future.

Input Data Entry: **ASET** Selected Coordinates for the a-set

Description: Defines degrees of freedom that the user desires to place in the analysis set. Used to define the number of independent degrees of freedom.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
ASET	SETID	ID	C	ID	C	ID	C		
ASET	16	2	23	3516					

Field	Contents
SETID	The set identification number of the **REDUCE** set. (Integer > 0)
ID	Grid or scalar point identification number (Integer > 0)
C	Component number, zero or blank for scalar points, any unique combinations of the digits 1 through 6 for grid points.

Remarks:

1. When **ASET** and/or **ASET1** entries are present, all degrees of freedom not otherwise constrained will be placed on the o-set. The o-set is a mutually exclusive set. Degrees of freedom may not be specified on other entries that define mutually exclusive sets.
2. **ASET** entries must be selected in Solution Control (**REDUCE=SETID**) to be used.

Input Data Entry: **ASET1** Selected Coordinates for the a-set, Alternate Form

Description: Defines degrees of freedom that the user desires to place in the analysis set. Used to define the number of independent degrees of freedom.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
ASET1	SETID	C	G	G	G	G	G	G	CONT
CONT	G	G	G	-etc-					

ASET1	345	2	1	3	10	9	6	15	ABC
+bc	7	8							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
ASET1	SETI	C	ID1	"THRU"	ID2				

Field	Contents
SETID	The **REDUCE** set identification number (Integer > 0)
C	Component number (any unique combination of the digits 1 through 6 with no embedded blanks) when point identification numbers are grid points; must be null or zero if point identification numbers are scalar points.
G, ID1, ID2	Grid or scalar point identification numbers (Integer > 0, **ID2** > **ID1**)

Remarks:

1. When **ASET** and/or **ASET1** entries are present, all degrees of freedom not otherwise constrained will be placed in the o-set. The o-set is a mutually exclusive set. Degrees of freedom may not be specified on other entries that define mutually exclusive sets.
2. If the alternate form is used, all points in the sequence **ID1** through **ID2** are required to exist.
3. **ASET1** entries must be selected in Solution Control (**REDUCE=SETID**) to be used.

Input Data Entry: **ATTACH**

Description: Defines the aerodynamic control points to be attached to a reference grid for load transfer.

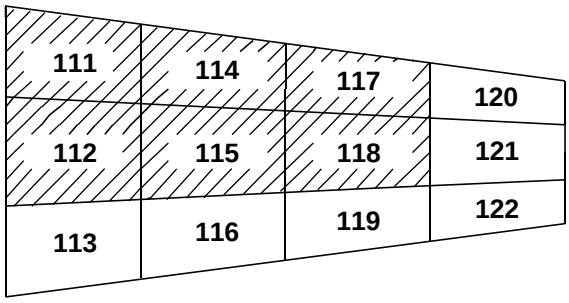
Format and Example:

1	2	3	4	5	6	7	8	9	10
ATTACH	EID	MACROID	BOX1	BOX2	RGRID				
ATTACH	100	111	111	118	1				

Field	Contents
EID	Element identification number (Integer > 0)
MACROID	Element identification of a **CAEROi** or **PAEROi** element which contains the specified aerodynamic control points (Integer > 0)
BOX1, BOX2	Starting and final box whose force is to be transferred to the referenced grid (Integer > 0, **BOX2** > **BOX1**)
RGRID	Grid point identification of reference grid point (Integer > 0)

Remarks:

1. The **EID** is used only for error messages.
2. This entry applies to both the steady and unsteady aerodynamic models.
3. The attached aerodynamic boxes are selected as shown below:



Input Data Entry: **AXSTA**

Description: Defines body axial station parameters. There is one AXSTA for each axial station at which the surface points are defined.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
AXSTA	BCID	XSTA	CBOD	ABOD	LYRAD	LZRAD			
AXSTA	10	10.00	0.5		10	20			

Field	Contents
BCID	Body component identification number (Integer > 0)
XSTA	Value of the x-ordinate of the body station (Real)
CBOD	Value of the z-ordinate of the center line at this station. This defines the body camber (Real).
ABOD	Cross sectional area of the body at this station (Real ≥ 0.0).
LYRAD, LZRAD	Identification number of an **AEFACT** data entry containing a list of the y-ordinates (z-ordinates) of the body section. (Integer ≥ 0.0)

Remarks:

1. If **ABOD** is present, the body is assumed to be circular and the radial ordinates are computed at **NRAD** (cf. the **BODY** bulk data entry) equal intervals. No **LYRAD** and **LZRAD** data are allowed when **ABOD** is present.
2. If **ABOD** is blank, **LYRAD** and **LZRAD** data must be present.
3. For Pods, **CBOD**, **LYRAD** and **LZRAD** data are not permitted.
4. For the fuselage, **XSTA** is actual x location; for pods, **XSTA** is relative to the **XLOC** value given on the **BODY** bulk data entry.

Input Data Entry: **BAROR** Simple Beam (BAR) Orientation Default Values

Description: Defines default values for fields 3 and 6 - 8 of the **CBAR** entry.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1,GO	X2	X3		
BAROR		39			0.6	2.9	-5.87		

Field	Contents
PID	Identification number of **PBAR** property entry (Integer > 0 or blank)
Xi	Vector components measured in displacement coordinate system at GA to determine (with the vector from end A to end B) the orientation of the element coordinate system for the bar element (Real or blank)
GO	Grid point identification number (Integer > 0)

Remarks:

1. The contents of fields on this entry are used for any **CBAR** entry whose corresponding fields are blank.
2. Only one **BAROR** entry may appear in the Bulk Data Packet.

Input Data Entry: **BODY**

Description: Defines body configuration parameters for steady aeroelasticity.

Format and Example:

1	2	3	4	5	6	7	8	9	10
BODY	BCID	CMPNT	CP	NRAD	XLOC	YLOC	ZLOC		
BODY	10	FUSEL	0	3					

Field	Contents
BCID	Body component identification number (Integer > 0)
CMPNT	Component type (**FUSEL** for the fuselage and **POD** for a pod)
CP	Coordinate system of the geometry input (Integer ≥ 0, or blank)
NRAD	Number of equal radial cuts used to define the body (Integer ≥ 0, or blank)
XLOC, YLOC, ZLOC	Ordinates of the nose of the pod in the **CP** coordinate system (Real)

Remarks:

1. **NRAD** is input if equally spaced radial cuts are desired. Arbitrary radial cuts are specified using the **AXSTA** and **AEFACT** data entries.
2. The geometry given with the **XLOC**, **YLOC**, **ZLOC** entries is used only with **POD** components.

Input Data Entry: **CAERO1** Aerodynamic Panel Element Connection

Description: Defines an aerodynamic macroelement (panel) in terms of two leading edge locations and side chords. This is used for Doublet-Lattice theory.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CAERO1	EID	PID	CP	NSPAN	NCHORD	LSPAN	LCHORD	IGID	CONT
+BC	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

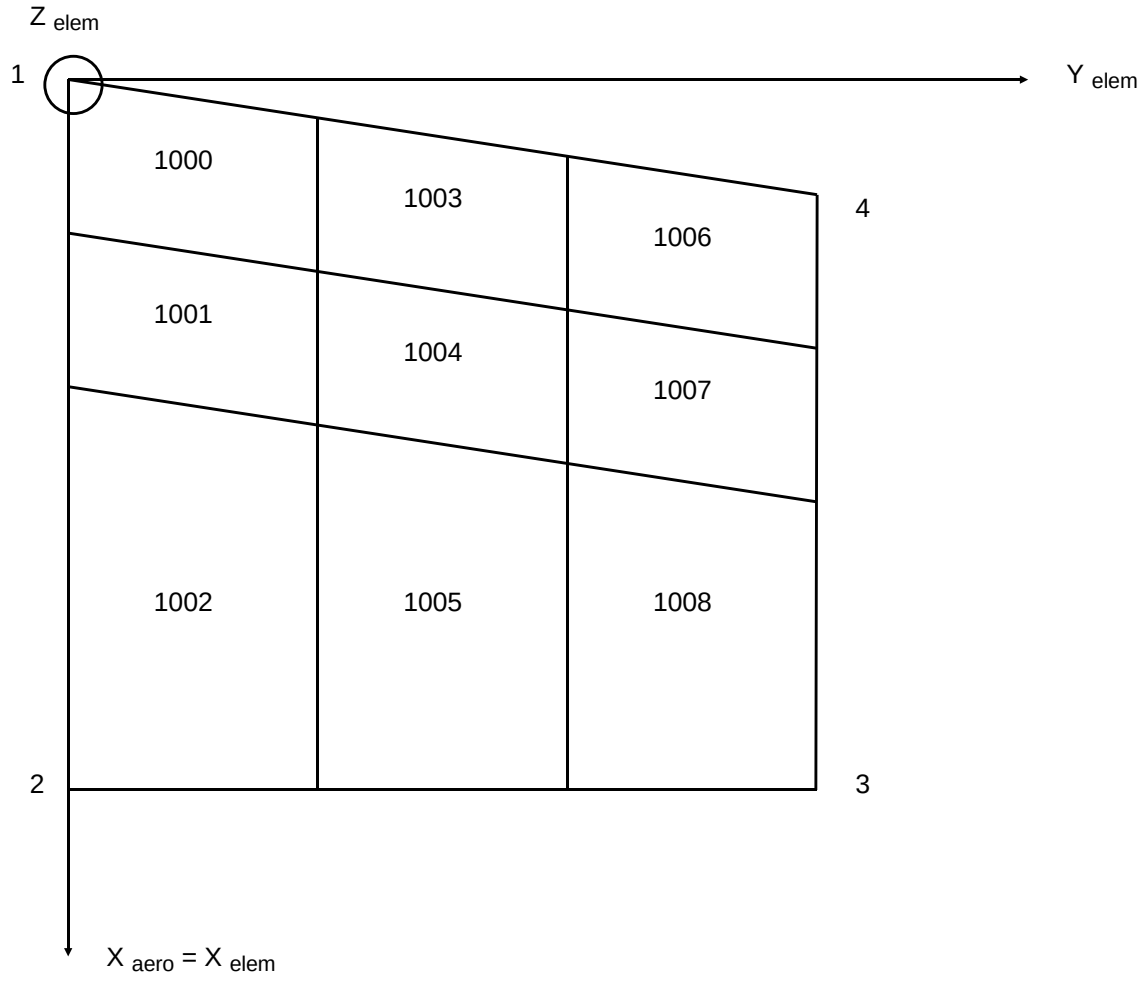
CAERO1	1000	1		3			2	1	ABC
+BC	0.0								

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of property entry (Integer > 0, or blank). Used to specify associated bodies
CP	Coordinate system for locating points 1 and 4 (Integer ≥ 0 or blank)
NSPAN	Number of span-wise boxes; if a positive value is given **NSPAN**, equal divisions are assumed; if zero or blank, a list of division points is given at **LSPAN** (Integer ≥ 0 or blank)
NCHORD	Number of chord-wise boxes; if a positive value is given **NCHORD**, equal divisions are assumed; if zero of blank, a list of division points is given at **LCHORD** (Integer ≥ 0 or blank)
LSPAN	Identification number of an **AEFACT** data entry containing a list of division points for span-wise boxes. Used only if **NSPAN** is zero or blank (Integer ≥ 0 or blank)
LCHORD	Identification number of an **AEFACT** data entry containing a list of division points for chord-wise boxes. Used only if **NCHORD** is zero or blank (Integer ≥ 0 or blank)
IGID	Interference group identification (aerodynamic elements with different **IGID**s are uncoupled) (Integer > 0)
X1,Y1,Z1; X4,Y4,Z4	Location of points 1 and 4, in coordinate system **CP** (Real)
X12, X43	Edge chord lengths (in aerodynamic coordinate system) (Real ≥ 0, and not both zero)

Remarks:

1. The boxes are numbered sequentially, beginning with **EID**.
2. The continuation entry is required.
3. The number of division points is one greater than the number of boxes. Thus, if **NSPAN** = 3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0. and 1. (in which the corners to the panel would not be at the reference points).
4. A triangular element is formed if **X12** or **X43** = 0.0.

5. The element coordinate system (right-handed) is shown in the sketch below.



Input Data Entry: **CAERO2** Unsteady Aerodynamic Body Connection

Description: Defines an aerodynamic body for Doublet-Lattice aerodynamics.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
CAERO2	EID	PID	CP	NSB	NINT	LSB	LINT	IGID	CONT
+BC	X1	Y1	Z1	X12					

CAERO2	1500	2	100		4	99		1	ABC
+BC	-1.0	100	-30	175					

Field	Contents
EID	Element identification number (Integer > 0)
PID	Property identification number (Integer > 0)
CP	Coordinate system for locating point 1 (Integer ≥ 0, or blank)
NSB	Grid point identification number of connection points (Integer > 0)
NINT	Number of interference elements; if a positive number is given, **NSB** equal divisions are assumed; if zero or blank, see **LSB** for a list of divisions (Integer ≥ 0, or blank)
LSB	Identification number of an **AEFACT** data entry for slender body division points; used only if **NSB** is zero or blank (Integer ≥ ,0 or blank)
LINT	Identification number of an **AEFACT** data entry containing a list of division points for interference elements; used only if **NINT** is zero or blank (Integer ≥ 0, or blank
IGID	Interference group identification (aerodynamic elements with different **IGID**'s are uncoupled) (Integer > 0)
X1, Y1, Z1	Location of points 1 and 4, in coordinate system **CP** (Real)
X12	Edge chord lengths (in aerodynamic coordinate system) (Real ≥ 0, and not both zero)

Remarks:

1. Point 1 is the leading point of the body.
2. All **CAERO1** (panels) and **CAERO2** (bodies) in the same group (**IGID**) will have aerodynamic interaction.
3. At least one interference element is required for each aerodynamic body specified by this entry.
4. Element identification numbers on the aerodynamic bodies must have the following sequence:
   (A) Panels first
   (B) Z bodies (see **PAERO2** orientation flag)
   (C) ZY bodies
   (D) Y bodies

Input Data Entry: **CAERO6**

Description: Defines an aerodynamic macroelement (panel) for USSAERO.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CAERO6	ACID	CMPNT	CP	IGRP	LCHORD	LSPAN			
CAERO6	1	WING		1	20	30			

Field	Contents
ACID	Component identification number (Integer > 0)
CMPNT	Aircraft component (Character) selected from: **WING** **FIN** **CANARD**
CP	Coordinate system (Integer ≥ 0, or blank) (See Remark 4)
IGRP	Group number for this component (Integer > 0)
LCHORD	Identification number of an **AEFACT** Bulk Data entry containing a list of division points in percent chord for chord-wise boxes for the aerodynamic surface. If **LCHORD** is zero, the chord-wise divisions are identified by the **IPANEL** entry on the **AIRFOIL** Bulk Data entry (Integer ≥ ,0 or blank)
LSPAN	Identification number of an **AEFACT** Bulk Data entry containing a list of division points for spanwise boxes. For **WING**s and **CANARD**s use the y (lateral) dimensional coordinates of the stations, and for **FIN**s, use the z (vertical) dimensional coordinates. If **LSPAN** is zero or blank, the **y/z** locations from the **AIRFOIL** Bulk Data entries for the component **ACID** are used (Integer ≥ ,0 or blank)

Remarks:

1. The **IGRP** field allows related components to be processed together for interference effects; e.g., one group could be a wing/body/tail combination while a second group could be a pod/fin combination.
2. Note that the chord-wise cuts are in percent while the span-wise cuts require physical coordinates. For span-wise cuts, y-coordinates are input for wings and canards while z-coordinates are input for fins.
3. Only the right half-plane can be modeled in USSAERO. As such, all y-coordinates specified by **LSPAN** must be positive.
4. The basic coordinate system must be used (**CP** blank). This field exists for the addition of a user defined coordinate system in the future.

Input Data Entry: **CASELIST**

Description: Defines a list of Subcase identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CASELIST	SID	CASE1	CASE2	CASE3	CASE4	CASE5	CASE6	CASE7	CONT
CONT	CASE8	CASE9	-etc-						

CASELIST	101	1	THRU	6					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
CASELIST	SID	CASE1	THRU	CASE2					

Field	Contents
SID	Subcase set identification number (Integer > 0)
CASEi	Subcase identification number (Integer > 0)

Remarks:

1. **CASELIST** Bulk Data entries are selected in the Function Packet.
2. Refer to the Solution Control discipline commands for details on assigning subcase identification numbers.

Input Data Entry: **CBAR** Simple Beam Element Connection

Description: Defines a simple beam element (BAR) of the structural model.

Format and Example:

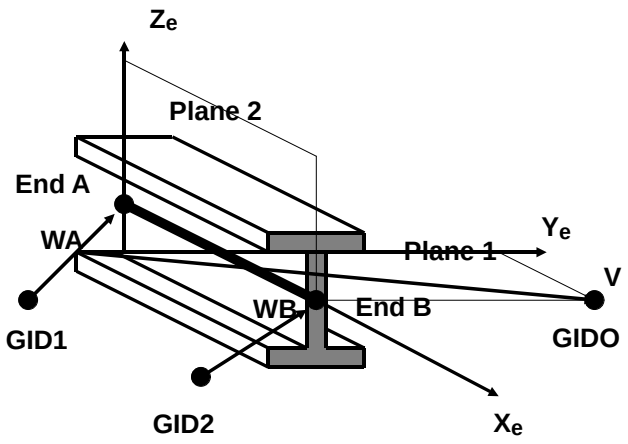
1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1,GO	X2	X3	TMAX	CONT
CONT	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

CBAR	2	39	7	3	13				
+23	123								

Field	Contents
EID	Unique element identification number (Integer > 0).
PID	Identification number of a **PBAR** property entry (Default is **EID** unless **BAROR** entry has nonzero entry in Field 3) (Integer > 0)
GA,GB	Grid point identification numbers of connection points (Integer > 0).
Xi	Components of vector {v}, at end A, measured at end A, parallel to the components of the displacement coordinate system for **GA**, to determine (with the vector from end A to end B) the orientation of the element coordinate system for the BAR element (Real)
GO	Grid point identification number to optionally supply **Xi** (Integer > 0). Direction of orientation vector is **GA** to **GO**
TMAX	Maximum allowable cross-sectional area in design (Real > 0.0, or blank). Default = 10^4.
PA,PB	Pin flags for bar ends A and B, respectively (up to 5 of the unique digits 1 through 6 anywhere in the fields with no embedded blanks; Integer > 0 or blank). Used to remove connections between the grid point and selected degrees of freedom of the bar. The degrees of freedom are defined in the **element's** coordinate system. The bar must have stiffness associated with the pin flag. For example, if PA=4 is specified, the **PBAR** entry must have a value for J, the torsional stiffness.
W1A,W2A,W3A W1B,W2B,W3B	Components of offset vectors wa and wb, respectively, in displacement coordinate systems at points **GA** and **GB**, respectively (Real or blank).

Remarks:

1. The element coordinate system is shown in the following figure:



2. If there are no pin flags or offsets, the continuation entry may be omitted.
3. The **TMAX** value is used only for shape function design variable linking.
4. See the **BAROR** entry for default options for Fields 3 and 6 through 8.

Input Data Entry: **CELAS1** Scalar Spring Connection

Description: Defines a scalar spring element of the structural model

Format and Example:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2	TMAX		
CELAS1	2	6			8	1			

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of a **PELAS** property entry (Default is **EID**) (Integer > 0)
Gi	Geometric grid point identification number (Integer ≥ 0)
Ci	Component number (6 ≥ Integer ≥ 0)
TMAX	Maximum value for design (Real, Default = 1.0 E4)

Remarks:

1. Scalar points may be used for **G1** and/or **G2** in which case the corresponding **C1** and/or **C2** must be zero or blank. Zero or blank may be used to indicate a grounded terminal **G1** or **G2** with a corresponding blank or zero **C1** or **C2**. A grounded terminal is a point whose displacement is constrained to zero.
2. The two connection points (**G1**, **C1**) and (**G2**, **C2**) must be distinct.
3. **TMAX** is ignored unless the element is designed using shape function linking.

Input Data Entry: **CELAS2** Scalar Spring Property and Connection

Description: Defines a scalar spring element of the structural model without reference to a property entry.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	CONT
CONT	TMIN	TMAX							

CELAS2	28	6.2+3	32		19	4			

Field	Contents
EID	Element identification number (Integer > 0)
K	The value of the scalar spring (Real > 0.0)
Gi	Geometric grid point identification number (Integer ≥ 0)
Ci	Component number (6 ≥ Integer ≥ 0)
GE	Damping coefficient (Real ≥ 0.0)
S	Stress coefficient (Real ≥ 0.0)
TMIN, TMAX	Minimum and maximum values for design (Real)

Remarks:

1. Scalar points may be used for **G1** and/or **G2** in which case the corresponding **C1** and/or **C2** must be zero or blank. Zero or blank may be used to indicate a grounded terminal **G1** or **G2** with a corresponding blank or zero **C1** or **C2**. A grounded terminal is a point whose displacement is constrained to zero.
2. This single entry completely defines the element since no material or geometric properties are required.
3. The two connection points (**G1**, **C1**) and (**G2**, **C2**) must be distinct.
4. The **TMIN** and **TMAX** values are ignored unless shape function design variable linking is used.

Input Data Entry: **CIHEX1** Linear Isoparametric Hexahedron Element Connection

Description: Defines a linear isoparametric hexahedron element of the structural model.

Format and Example:

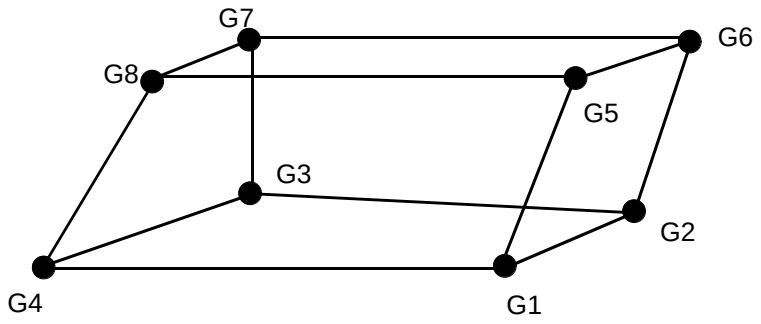
1	2	3	4	5	6	7	8	9	10
CIHEX1	EID	PID	G1	G2	G3	G4	G5	G6	CONT
CONT	G7	G8							

CIHEX1	137	5	3	8	5	4	9	14	ABC
+BC	88	602							

Field	Contents
EID	Element identification number (Integer > 0).
PID	Identification number of a **PIHEX** property entry (Integer > 0). (Default is **EID**)
Gi	Grid point identification numbers of connection points (Integer > 0, **G1** ≠ **G2** ≠...**G8**).

Remarks:

1. Grid points **G1**, **G2**, **G3**, and **G4** must be given in counterclockwise order about one quadrilateral face when viewed from within the element. Grid points **G5**, **G6**, **G7**, and **G8** must also be given in counter-clockwise order, and **G1** and **G5** must be along the same edge as shown in the figure below:



2. There is no nonstructural mass.
3. The quadrilateral faces need not be planar.
4. Stresses are given in the basic coordinate system.
5. The continuation is required.
6. No physical property in this element can be used as a local design variable for automated design.

Input Data Entry: **CIHEX2** Quadratic Isoparametric Hexahedron Element Connection

Description: Defines a quadratic isoparametric hexahedron element of the structural model.

Format and Example:

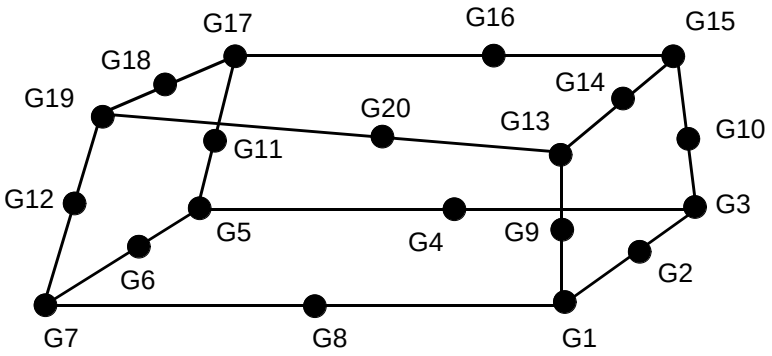
1	2	3	4	5	6	7	8	9	10
CIHEX1	EID	PID	G1	G2	G3	G4	G5	G6	CONT
CONT	G7	G8	G9	G10	G11	G12	G13	G14	CONT
CONT	G15	G16	G17	G18	G19	G20			

CIHEX1	110	7	3	8	12	13	14	9	ABC
+BC	5	4	16	19	20	17	23	27	DEF
+EF	31	32	33	28	25	24			

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of a **PIHEX** property entry (Integer > 0) (Default is **EID**)
Gi	Grid point identification numbers of connection points (Integer > 0, **G1** ≠ **G2** ≠ ....≠ **G20**).

Remarks:

1. Grid points **G1**,...,**G8** must be given in counterclockwise order about one quadrilateral face when viewed from within the element. **G9**,...,**G12** and **G13**,...,**G20** must also be in a counterclockwise direction with **G1**, **G9** and **G13** along the same edge as shown in the figure below:



2. There is no nonstructural mass.
3. The quadrilateral faces need not be planar.
4. Stresses are given in the basic coordinate system.
5. The continuations are required.
6. No physical property in this element can be used as a local design variable for automated design.

Input Data Entry: **CIHEX3** Cubic Isoparametric Hexahedron Element Connection

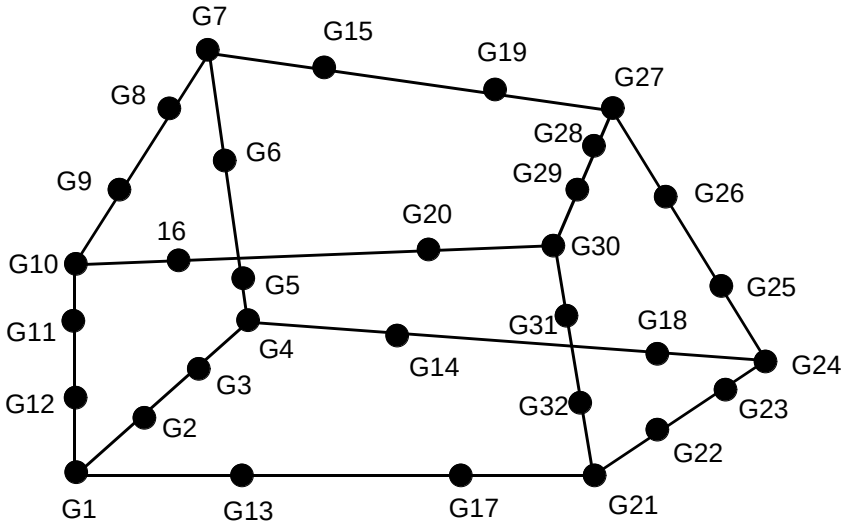
Description: Defines a cubic isoparametric hexahedron element of the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CIHEX1	EID	PID	G1	G2	G3	G4	G5	G6	CONT
CONT	G7	G8	G9	G10	G11	G12	G13	G14	CONT
CONT	G15	G16	G17	G18	G19	G20	G21	G22	CONT
CONT	G23	G24	G25	G26	G27	G28	G29	G30	CONT
CONT	G31	G32							

CIHEX1	15	3	4	9	12	17	18	19	ABC
+BC	20	13	10	7	6	5	22	25	DEF
+EF	31	32	33	28	25	24	108	214	GHI
+HI	106	213	413	95	67	40	45	90	+KL
+KL	38	37							

Field	Contents
EID	Element identification number (Integer > 0).
PID	Identification number of a **PIHEX** property entry (Integer > 0) (Default is **EID**)
Gi	Grid point identification number of connection points (Integer > 0, **G1** ≠ **G2** ≠ ... ≠ **G32**).



Remarks:

1. Grid points **G1**,...,**G12** must be given in counterclockwise order about one quadrilateral face when viewed from inside the element. **G13**,...,**G16**; **G17**,...,**G20;** and **G21**,..., **G32** must also be in a counter-clockwise direction with **G1**, **G13**, **G17**, and **G21** along the same edge as shown in the previous figure.
2. There is no nonstructural mass.
3. The quadrilateral faces need not be planar.
4. Stresses are given in the basic coordinate system.
5. The continuations are required.
6. No physical property in this element can be used as a local design for automated design.

Input Data Entry: **CMASS1** Scalar Mass Connection

Description: Defines a scalar mass element of the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2	TMAX		
CMASS1	32	6	2	1					

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of a **PMASS** property entry (Default is **EID**) (Integer > 0)
Gi	Geometric grid point identification number (Integer > 0)
Ci	Component number (6 ≥ Integer ≥ 0)
TMAX	The maximum mass value allowed in design (Real, Default = 10^4)

Remarks:

1. Scalar points may be used for **G1** and/or **G2** in which case the corresponding **C1** and/or **C2** must be zero or blank. Zero or blank may be used to indicate a grounded terminal **G1** or **G2** with a corresponding blank or zero **C1** or **C2**. A grounded terminal is a point whose displacement is constrained to zero.
2. The two connection points (**G1**, **C1**) and (**G2**, **C2**), must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for **G** and **C**.
3. The **TMAX** value is used only for shape function design variable linking.

Input Data Entry: **CMASS2** Scalar Mass Property and Connection

Description: Defines a scalar mass element of the structural model without reference to a property entry.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	M	G1	C1	G2	C2	TMIN	TMAX	
CMASS2	32	9.25	6	1					

Field	Contents
EID	Element identification number (Integer > 0)
M	The value of the scalar mass (Real)
Gi	Geometric grid point identification number (Integer > 0)
Ci	Component number 6 ≥ Integer ≥ 0)
TMIN,TMAX	The minimum and maximum mass values in design (Real)

Remarks:

1. Scalar points may be used for **G1** and/or **G2** in which case the corresponding **C1** and/or **C2** must be zero or blank. Zero or blank may be used to indicate a grounded terminal **G1** or **G2** with a corresponding blank or zero **C1** or **C2**. A grounded terminal is a point whose displacement is constrained to zero.
2. This single entry completely defines the element since no material or geometric properties are required.
3. The two connection points (**G1**, **C1**) and (**G2**, **C2**), must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for G and C.
4. The **TMIN** and **TMAX** values are used only for shape function design variable linking.

Input Data Entry: **CONEFFF** Flutter aerodynamic control effectiveness data

Description: Defines adjustment factors of control surface effectiveness values for use in flutter analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONEFFF	EFFID	EFF	MODE	MACROID	BOX1	BOX2			
CONEFFF	10	0.60	6	1001	1007	1021			

Field	Contents
EFFID	Effectiveness identification number (Integer > 0)
EFF	Effectiveness value (Real)
MODE	Structural mode to which the effectiveness is to be applied (Integer > 0)
MACROID	Aerodynamic component (macroelement) on which the control surface lies
BOX1, BOX2	First and last box whose effectiveness is to be altered (Integer > 0, **BOX2** > **BOX1**)

Remarks:

1. The **EFFID** is referenced by the **FLUTTER** bulk data entry.
2. The **EFFID** need not be unique.
3. The pressures for the referenced mode and all the referenced boxes will be modified by the **EFF** parameter. For example, **EFF** = 0.60 indicates a 40 percent reduction in the effectiveness for the affected boxes.
4. Refer to the **SPLINE1** bulk data entry for the interpretation of **BOX1** and **BOX2**.

Input Data Entry: **CONEFFS** Static aerodynamic control effectiveness data

Description: Defines adjustment factors for control surface effectiveness values for use in static aeroelastic analysis and nonplanar aerodynamic analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONEFFS	EFFID	LABEL1	EFF1	LABEL2	EFF2	LABEL3	EFF3		CONT
CONT	LABEL4	EFF4	-etc-						

CONEFFS	10	AIL1	0.65	INBORD	0.55				

Field	Contents
EFFID	A unique identification number identifying the set
LABELi	A unique alphanumeric string of up to eight characters to identify a control surface defined by an **AESURF** entry
EFFi	Effectiveness value for the associated surface (Real)

Remarks:

1. The set identification number is referenced by the **TRIM** bulk data entry.
2. All aerodynamic forces created by the control surface will be reduced to the reference amount. For example, **EFF1** = 0.70 indicates a 30 percent reduction in the forces.

Input Data Entry: **CONLINK** Linked Control Surfaces

Description: Causes control surfaces to vary in a prescribed fashion relative to one another.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONLINK	LABEL		LABEL1	VAL1	LABEL2	VAL2	LABEL3	VAL3	CONT
CONT	LABEL4	VAL4	-etc-						

CONLINK	ROLL1		AIL	1.0	LEFLAP	1.0			

Field	Contents
LABEL	A unique alphanumeric string of up to eight characters to identify the control surface taht is composed of other control surfaces.
LABELi	A unique alphanumeric string of up to eight characters to identify a control surface defined by an **AESURF** entry
VALi	Participation factor (Real)

Remarks:

1. All of the **LABEL** surfaces must be of the same **TYPE**, e.g. **SYM**. See the **AESURF** entry for additional information.
2. An arbitrary number of entries are allowed.
3. The **CONLINK** entry may not reference the **LABEL** of another **CONLINK** entry.

Input Data Entry: **CONM1** Concentrated Mass Element Connection, General Form

Description: Defines a 6 x 6 symmetric matrix at a geometric grid point of the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	CONT
CONT	M33	M41	M42	M43	M44	M51	M52	M53	CONT
CONT	M54	M55	M61	M62	M63	M64	M65	M66	

CONM1	2	22	2	2.9	6.3	+1			+1
+1	4.8				28.6				+2
+2		28.6						28.6	

Field	Contents
EID	Element identification number (Integer > 0).
G	Grid point identification number (Integer > 0).
CID	Coordinate system identification number for the mass matrix (Integer ≥ 0 or blank)
Mij	Mass matrix values (Real).

Remarks:

1. For a less general means of defining concentrated mass at grid points, see **CONM2**.
2. No physical property in this element can be used as a local design variable for automated design.

Input Data Entry: **CONM2** Concentrated Mass Element Connection, Rigid Body Form

Description: Defines a concentrated mass at a grid point of the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		CONT
CONT	I11	I21	I22	I31	I32	I33	TMIN	TMAX	

CONM2	2	15	6	49.7					123
+23	16.2		16.2			7.8			

Field	Contents
EID	Element identification number (Integer > 0).
G	Grid point identification number (Integer > 0).
CID	Coordinate system identification number (Integer ≥ -1). A **CID** of –1 (integer) allows the user to input **Xi** as the center of gravity location in the basic coordinate system. A **CID** of **0** implies the basic coordinate system
M	Mass value (Real).
Xi	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in Field 4, unless **CID** = –1, in which case **Xi** are the coordinates of the center of gravity of the mass in the basic coordinate system (Real).
Iij	Mass moments of inertia measured at the mass c.g., in coordinate system defined by Field 4 (Real). If **CID** = –1, the basic coordinate system is implied.
TMIN, TMAX	The minimum and maximum mass values for design (Real)

Remarks:

1. The continuation entry may be omitted.
2. If **CID** = –1, offsets are internally computed as the difference between the grid point location and **Xi**. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.

3. The form of the inertia matrix about its c.g. is taken as:

$$\boldsymbol{M} = \begin{bmatrix} M & & & & & \\ & M & & & \text{SYM} & \\ & & M & & & \\ & & & I_{11} & & \\ & & & -I_{21} & I_{22} & \\ & & & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

where:

$$\mathbf{M} = \int \rho \, dv$$

$$\mathbf{I11} = \int \rho \, (x_2^2 + x_3^2) \, dv$$

$$\mathbf{I22} = \int \rho \, (x_1^2 + x_3^2) \, dv$$

$$\mathbf{I33} = \int \rho \, (x_1^2 + x_2^2) \, dv$$

$$\mathbf{I21} = \int \rho \, x_1 \, x_2 \, dv$$

$$\mathbf{I31} = \int \rho \, x_1 \, x_3 \, dv$$

$$\mathbf{I32} = \int \rho \, x_2 \, x_3 \, dv$$

and x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in Field 4. The negative signs for the off-diagonal terms are supplied by the program. A warning message is issued if the inertia matrix is non-positive definite, as this may cause fatal errors in dynamic analysis modules.

4. For design, the mass moments of inertia must be zero.

5. The **TMIN** and **TMAX** values are used only for shape function design variable linking.

Input Data Entry: **CONROD** Rod Element Property and Connection

Description: Defines a rod element of the structural model without reference to a property entry.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	CONT
CONT	TMIN	TMAX							

CONROD	2	16	17	23	2.69				

Field	Contents
EID	Element identification number (Integer > 0).
G1,G2	Grid point identification numbers of connection points (Integer > 0)
MID	Material identification number (Integer > 0).
A	Area of rod (Real ≥ 0.0).
J	Torsional constant (Real ≥ 0.0).
C	Coefficient for torsional stress determination (Real).
NSM	Nonstructural mass per unit length (Real).
TMIN,TMAX	Minimum and maximum allowable cross-sectional areas in design (Real > 0.0, or blank)

Remarks:

1. For structural problems, **CONROD** entries may only reference **MAT1** material entries.
2. The continuation entry is optional.
3. **TMAX** and **TMIN** are ignored unless element is linked to global design variable through a **SHAPE** entry.

Input Data Entry: **CONVERT**

Description: Defines conversion factors for various physical quantities.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CONVERT	QUANT1	FACTOR	QUANT2	FACTOR	QUANT5	FACTOR	QUANT4	FACTOR	CONT
CONT	QUANT	FACTOR	QUANT	FACTOR	-etc-				

CONVERT	MASS	0.00259							

Field	Contents
QUANTi	A character string identifying the physical quantity to be converted = **MASS**, or **VELOCITY**
FACTOR	The conversion factor (Real ≠ 0.0)

Remarks:

1. Any number of valid quantity-factor combinations can be entered on a single entry.
2. Only **MASS** and **VELOCITY** are currently valid quantity entries.
3. Input mass values will be multiplied by the input factor. Input velocities will be multiplied by the factor.

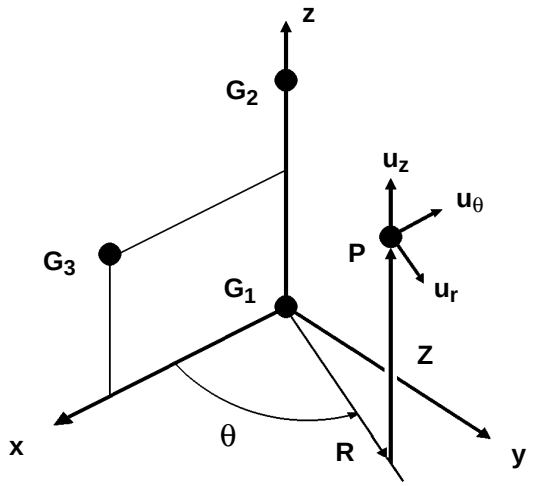
Input Data Entry: **CORD1C** Cylindrical Coordinate System Definition, Form 1

Description: Defines a cylindrical coordinate system by reference to three grid points. These points must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CORD1C	CID	G1	G2	G3	CID	G1	G2	G3	
CORD1C	3	16	32	19					

Field	Contents
CID	Coordinate system identification number (Integer > 0)
Gi	Grid point identification number (Integer > 0; **G1** ≠ **G2** ≠ **G3**).



Remarks:

1. Coordinate system identification numbers on all **CORD1R**, **CORD1C**, **CORD1S**, **CORD2R**, **CORD2C**, and **CORD2S** entries must be unique.
2. The three points **G1**, **G2**, and **G3** must be noncollinear.
3. The location of a grid point (**P** in the sketch) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
4. The displacement coordinate directions at **P** are dependent on the location of **P** as shown above by (u_r, u_θ, u_z,).
5. Points on the z-axis may not have their displacement directions defined in this coordinate system since an ambiguity results.
6. One or two coordinate systems may be defined on a single entry.

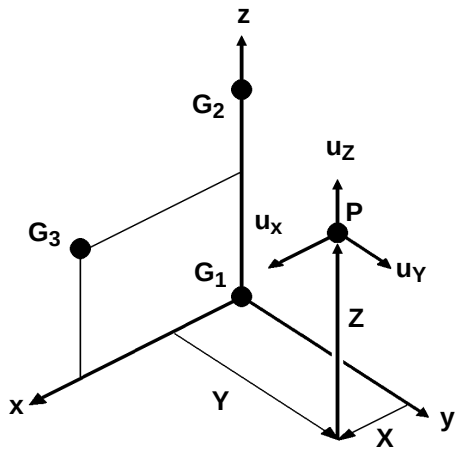
Input Data Entry: **CORD1R** Rectangular Coordinate System Definition, Form 1

Description: Defines a rectangular coordinate system by reference to three grid points. These points must be defined in coordinate systems whose definition does not involve the coordinate systems defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CORD1R	CID	G1	G2	G3	CID	G1	G2	G3	
CORD1R	3	16	32	19					

Field	Contents
CID	Coordinate system identification number (Integer > 0)
Gi	Grid point identification number (Integer > 0; **G1** ≠ **G2** ≠ **G3**).



Remarks:

1. Coordinate system identification numbers on all **CORD1R**, **CORD1C**, **CORD1S**, **CORD2R**, **CORD2C**, and **CORD2S** entries must be unique.
2. The three points **G1**, **G2**, and **G3** must be noncollinear.
3. The location of a grid point (**P** in the sketch) in this coordinate system is given by (X, Y, Z).
4. The displacement coordinate directions at **P** are shown above by (u_x, u_y, u_z)
5. One or two coordinate systems may be defined on a single entry.

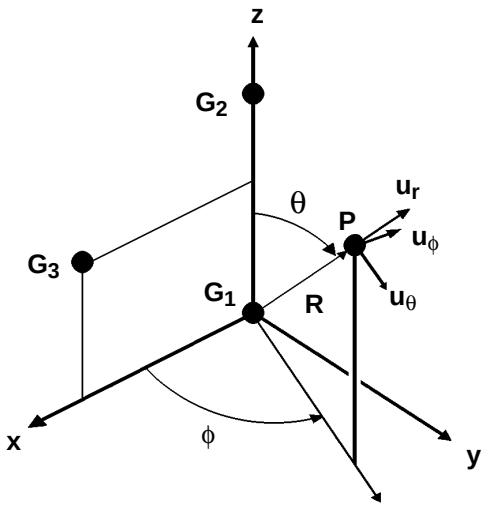
Input Data Entry: **CORD1S** Spherical Coordinate System Definition, Form 1

Description: Defines a spherical coordinate system by reference to three grid points. These points must be defined in coordinate systems whose definition does not involve the coordinate systems defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
CORD1S	CID	G1	G2	G3	CID	G1	G2	G3	
CORD1S	3	16	32	19					

Field	Contents
CID	Coordinate system identification number (Integer > 0)
Gi	Grid point identification number (Integer > 0; **G1** ≠ **G2** ≠ **G3**).



Remarks:

1. Coordinate system identification numbers on all **CORD1R**, **CORD1C**, **CORD1S**, **CORD2R**, **CORD2C**, and **CORD2S** entries must be unique.
2. The three points **G1**, **G2**, and **G3** must be noncollinear.
3. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, ϕ) where θ and φ are measured in degrees.
4. The displacement coordinate directions at **P** are dependent on the locations of **P** as shown above by (u_r, u_θ, u_φ).
5. Points in the polar axis may not have their displacement direction defined in this coordinate system since an ambiguity results.
6. One or two coordinate systems may be defined on a single entry.

Input Data Entry: **CORD2C** Cylindrical Coordinate System Definition, Form 2

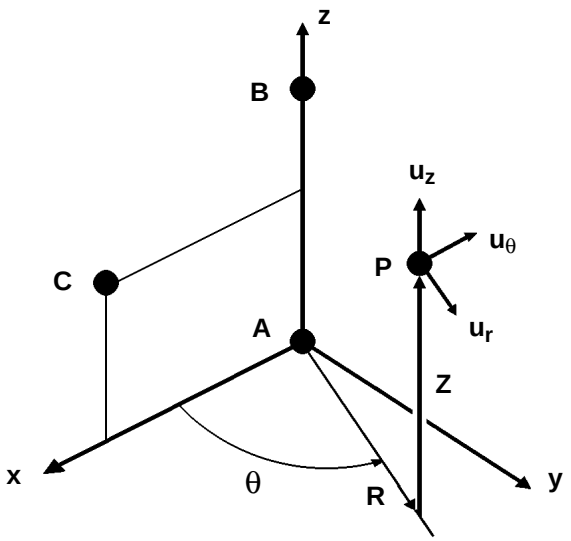
Description: Defines a cylindrical coordinate system by reference to the coordinates of three grid points. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	CONT
CONT	C1	C2	C3						

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	123
+23	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number (Integer > 0)
RID	Reference to a coordinate system which is defined independently of new coordinate system (Integer ≥ 0, or blank)
Ai,Bi,Ci	Coordinates of three points in coordinate system defined by **RID** (Real)



Remarks:

1. Continuation entry must be present.
2. The three points (**A1**, **A2**, **A3**), (**B1**, **B2**, **B3**), (**C1**, **C2**, **C3**) must be unique and noncollinear, Noncollinearity is checked by the geometry processor.
3. Coordinate system identification numbers on all **CORD1R**, **CORD1C**, **CORD1S**, **CORD2R**, **CORD2C**, and **CORD2S** entries must all be unique.

4. An **RID** of zero references the basic ordinate system.
5. The location of a grid point (P in the sketch) in this coordinate is given by (R, θ, Z) where θ is measured in degrees.
6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).
7. Points on the z-axis may not have their displacement direction defined in this coordinate system since an ambiguity results.

Input Data Entry: **CORD2R** Rectangular Coordinate System Definition, Form 2

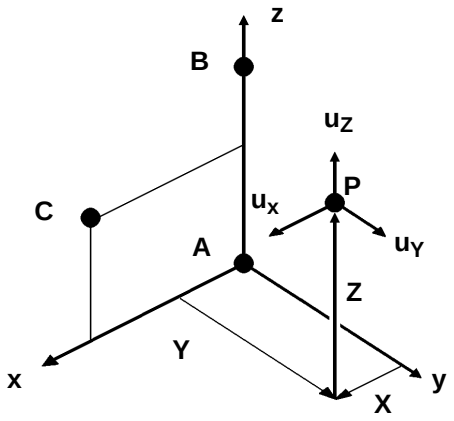
Description: Defines a rectangular coordinate system by reference to coordinates of three points. The first point defines the origin. The second defines the direction of the z-axis. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	CONT
CONT	C1	C2	C3						

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	123
+23	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number (Integer > 0)
RID	Reference to a coordinate system which is defined independently of new coordinate system (Integer ≥ 0, or blank)
Ai,Bi,Ci	Coordinates of three points in coordinate system defined by **RID** (Real)



Remarks:

1. The continuation entry must be present.
2. The three points (**A1**, **A2**, **A3**), (**B1**, **B2**, **B3**), (**C1**, **C2**, **C3**) must be unique and noncollinear. Noncollinearity is checked by the geometry processor.
3. Coordinate system identification numbers on all **CORD1R**, **CORD1C**, **CORD1S**, **CORD2R**, **CORD2C**, and **CORD2S** entries must all be unique.
4. An **RID** of zero references the basic coordinate system.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (X, Y, Z)
6. The displacement coordinate directions at P are shown by (u_x, u_y, u_z)

Input Data Entry: **CORD2S** Spherical Coordinate System Definition, Form 2

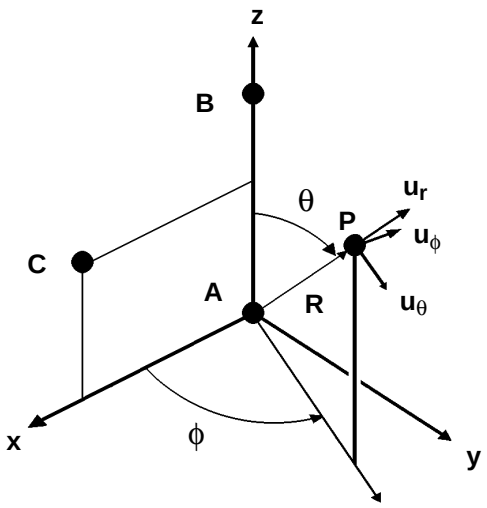
Description: Defines a spherical coordinate system by reference to the coordinates of three points. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	CONT
CONT	C1	C2	C3						

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	123
+23	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number (Integer > 0)
RID	Reference to a coordinate system which is defined independently of of new coordinate system (Integer ≥ 0, or blank)
Ai,Bi,Ci	Coordinates of three points in coordinate system defined by **RID** (Real)



Remarks:

1. The continuation entry must be present.
2. The three points (**A1**, **A2**, **A3**), (**B1**, **B2**, **B3**), (**C1**, **C2**, **C3**) must be unique and noncollinear.
3. Coordinate system identification numbers on all **CORD1R**, **CORD1C**, **CORD1S**, **CORD2R**, **CORD2C** and **CORD2S** entries must all be unique.
4. An **RID** of zero references the basic coordinate system.

5. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, φ) where θ, and φ are measured in degrees.
6. The displacement coordinate directions at P are shown above by (u_r, u_θ, u_φ).
7. Points on the polar axis may not have their displacement directions defined in this coordinate system since an ambiguity results.

Input Data Entry: **CQDMEM1** Isoparametric Quadrilateral Element Connection

Description: Defines the isoparametric quadrilateral membrane element.

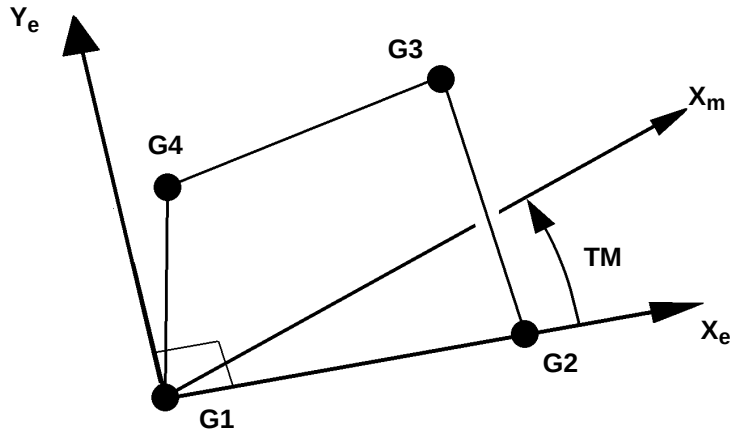
Format and Example:

1	2	3	4	5	6	7	8	9	10
CQDMEM1	EID	PID	G1	G2	G3	G4	TM	TMAX	
CQDMEM1	72	13	13	14	15	16	29.2		

Field	Contents
EID	Element identification number (Integer > 0).
PID	Identification number of a **PQDMEM1** or **PCOMP** property entry (Default is **EID**) (Integer > 0).
Gi	Grid point identification numbers of connection points (Integer > 0)
TM	Material property orientation angle. If **TM** is real, the sketch below gives the sign convention for **TM**. If **TM** is an integer, the material x-axis is along the projection onto the plane of the element of the x-axis of coordinate system identified by the integer.
TMAX	Maximum allowable element thickness in design (Real > 0.0 or blank). (Default=10^4)

Remarks:

1. Grid points **G1** through **G4** must be ordered consecutively around the perimeter of the element as shown in the figure below.



2. All interior angles must be less than 180°.
3. **TMAX** is ignored unless element is linked to global design variable by a **SHAPE** entry.

Input Data Entry: **CQUAD4** Quadrilateral Element Connection

Description: Quadrilateral plate element (QUAD4) of the structural model. This is an isoparametric membrane-bending element.

Format and Example:

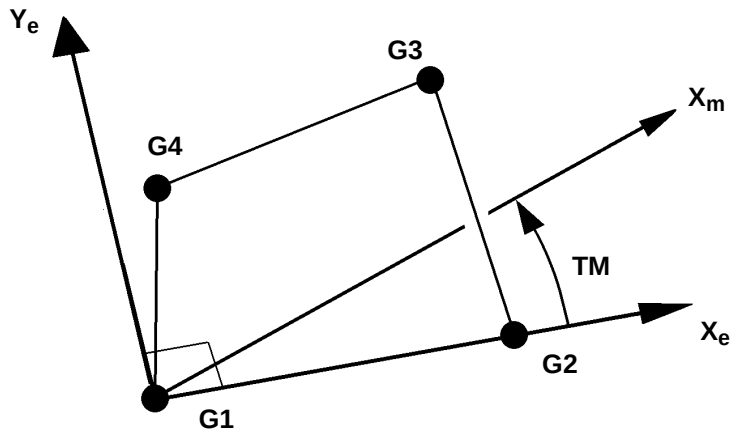
1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	TM	ZOFF	CONT
CONT		TMAX	T1	T2	T3	T4			

CQUAD4	101	17	1001	1005	1010	1024	45.0	0.01	ABC
+BC			0.03	0.125	0.05	0.04			

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of a **PSHELL** or **PCOMPi** entry (Default is **EID**) (Integer > 0).
Gi	Grid point identification numbers of connection points (Integer > 0).
TM	Material property orientation specification (Real or blank; or 0 ≤ Integer < 1,000,000). If Real or blank, specifies the material property orientation angle in degrees. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value.
ZOFF	Offset of the element reference plane from the plane of grid points. A positive value means the $+z_e$ direction. (Real or blank, see Remark 2 for default).
TMAX	Maximum allowable element thickness in design (Real > 0.0).
Ti	Membrane thickness of element at grid points Gi (Real or blank, see Remark 3 for default).

Remarks:

1. The **QUAD4** geometry, coordinate systems and numbering are shown in the figure below:



2. The material coordinate system (**TM**) and the offset (**ZOFF**) may also be provided on the **PSHELL** entry. The property data will be used if the corresponding field on the **CQUAD4** entry is blank. The element reference plane is located at the mid-thickness of the element parallel to the element mean plane.
3. The **Ti** are optional, if not supplied they will be set to the value of T specified on the **PSHELL** entry. In such cases, the continuation entry is not required.
4. **TMAX** is ignored unless the element is linked to the global design variables by a **SHAPE** entry.

Input Data Entry: **CROD** Rod Element Connection

Description: Defines a tension-compression-torsion element (ROD) of the structural model.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2	TMAX				
CROD	12	13	21	23					

Field	Contents
EID	Element identification number (Integer > 0).
PID	Identification number of a **PROD** property entry (Default is **EID**) (Integer > 0).
Gi	Grid point identification numbers of connection points (Integer > 0)
TMAX	Maximum allowable rod area in design (Real > 0.0 or blank)

Remarks:

1. See **CONROD** for alternative method of rod definition.
2. Only one **ROD** element may be defined on a single entry.
3. **TMAX** is ignored unless the element is linked to global design variables by a **SHAPE** entry.

Input Data Entry: **CSHEAR** Shear Panel Element Connection

Description: Defines a shear panel element (SHEAR) of the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4	TMAX		
CSHEAR	3	6	1	5	3	7			

Field	Contents
EID	Element identification number (Integer > 0).
PID	Identification number of a **PSHEAR** property entry (Default is **EID**) (Integer > 0).
Gi	Grid point identification numbers of connection points (Integer > 0)
TMAX	Maximum allowable thickness in design (Real > 0.0, or blank).

Remarks:

1. Grid points **G1** through **G4** must be ordered consecutively around the perimeter of the element.
2. All interior angles must be less than 180°.
3. **TMAX** is ignored unless element is linked to global design variable by a **SHAPE** entry.

Input Data Entry: **CTRIA3** Triangular Element Connection

Description: Defines a triangular shell element (TRIA3) of the structural model.

Format and Example:

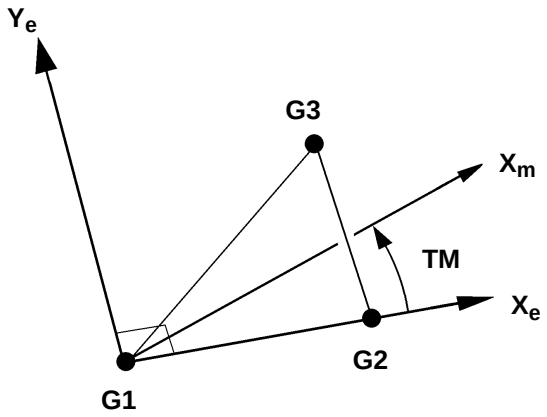
1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	TM	ZOFF		CONT
CONT		TMAX	T1	T2	T3				

CTRIA3	101	17	1001	1005	1010	45.0	0.01		ABC
+BC			0.03	0.125	0.05				

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of a **PSHELL** or **PCOMPi** property entry (Default is **EID**) (Integer > 0).
Gi	Grid point identification numbers of connection points (Integer > 0).
TM	Material property orientation specification (Real or blank; or 0 ≤ Integer < 1,000,000). If Real or blank, specifies the material property orientation angle in degrees. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value.
ZOFF	Offset of the element reference plane from the plane of grid points. A positive value means the $+z_e$ direction. (Real or blank, see Remark 2 for default).
TMAX	Maximum allowable element thickness in design (Real > 0.0) (Default = 10^4)
Ti	Membrane thickness of element at grid points **Gi** (Real or blank, see Remark 3 for default).

Remarks:

1. The **TRIA3** geometry, coordinate systems and numbering are shown in the figure below:



2. The material coordinate system (**TM**) and the offset (**ZOFF**) may also be provided on the **PSHELL** entry. The property data will be used if the corresponding field on the **CTRIA3** entry is blank. The element reference plane is located at the mid-thickness of the element parallel to the element mean plane.
3. The **Ti** are optional, if not supplied they will be set to the value of **T** specified on the **PSHELL** entry. In such cases, the continuation entry is not required.
4. **TMAX** is ignored unless the element is linked to the global design variables by a **SHAPE** entry.

Input Data Entry: **CTRMEM**

Description: Defines a triangular membrane element.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
CTRMEM	EID	PID	G1	G2	G3	TM	TMAX		
CTRMEM	100	500	1	7	12				

Field	Contents
EID	Element identification number (Integer > 0).
PID	Identification of **PTRMEM** or **PCOMP** entry (Integer > 0) Default = **EID**.
Gi	Grid point identifications of connection points (Integer > 0).
TM	Material orientation angle (Real) **or** 0 < Integer < 1,000,000. If integer, then material x-axis lies along the projection onto the plane of the element of the x-axis of coordinate system identified by the integer.
TMAX	Maximum allowable thickness in design. (Real ≥ 0., Default = 10^4)

Remarks:

1. The **TMAX** value is used only for shape function design variable linking.

Input Data Entry: **DCONALE**

Description: Defines an aileron effectiveness constraint of the form:

$AE \leq$ **AEREQ** (upper bound) or $AE \geq$ **AEREQ** (lower bound)

where,

$$AE = \frac{-C_{l_{\delta_a}}}{C_{l_{\frac{pb}{2v}}}}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONALE	SID	LABEL	CTYPE	AEREQ					
DCONALE	25	OUTBDAIL	LOWER	0.4					

Field	Contents
SID	Aerodynamic set identification for the imposed constraint (Integer > 0)
LABEL	A string of up to eight characters to identify the **AESURF** or **CONLINK** control surface. **LABEL**s must be unique.
CTYPE	Constraint type: either **UPPER** for upper bound or **LOWER** for lower bound (Character, Default = **LOWER**)
AEREQ	Required aileron effectiveness (Real ≠ 0.0)

Remarks:

1. This constraint constraint will only be applied if selected by the Solution Control discipline option **DCON=SID** and if an antisymmetric aeroelastic trim analysis is being performed.
2. A **LOWER** bound constraint excludes all values to the left of **AEREQ** on a real number line, while an **UPPER** bound constraint excludes all values to the right, irrespective of the sign of **AEREQ**.
3. The effectiveness in roll of multiple control surfaces may be specified using multiple **DCONALE** entries with one constraint generated for each **LABEL**/**CTYPE** combination.

Input Data Entry: **DCONBK** Buckling Constraint Definition

Description: Defines a local panel buckling constraint of the form:

$$\text{Lower Bound: } g_{lower} = \left(\frac{\lambda_{REQ}}{\lambda}\right)^{1/3} - 1.0 \leq 0.0 \quad \text{for } \lambda \geq \lambda_{REQ}$$

or:

$$\text{Upper Bound: } g_{upper} = 1.0 - \left(\frac{\lambda_{REQ}}{\lambda}\right)^{1/3} \leq 0.0 \quad \text{for } \lambda < \lambda_{REQ}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONBK	SID	ETYPE	EID	LENGTH	WIDTH		CTYPE	λ_{REQ}	
DCONBK	25	QUAD4	101	1.5			LOWER	3.65	

Field	Contents
SID	Plate buckling panel constraint set identification. (Integer > 0)
ETYPE	Plate buckling control element type. May be **QUAD4** or **TRIA3** (Character)
EID	Element identification number. (Integer > 0)
LENGTH	Plate buckling panel length in consistant length units. (Real > 0.0 or blank) (See Remark 3)
WIDTH	Plate buckling panel width in consistant length units. (Real > 0.0 or blank) (See Remark 3)
CTYPE	Constraint type: either **LOWER** for lower bound or **UPPER** for upper bound. (Character)
λ_{REQ}	Buckling eigenvalue limit. (Real, Default = 1.0)

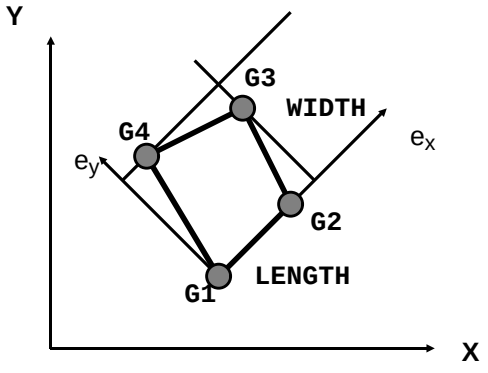
Remarks:

1. Buckling constraints are selected in Solution Control with the discipline option:

   DCON = *sid*

2. The buckling control element (which must be a designed element) supplies the running loads Nx, Ny and Nxy and material properties to the rectangular pseudo-panel of dimension **LENGTH x WIDTH**.

3. If **LENGTH** or **WIDTH** are omitted, the corresponding value will be computed from the rectangle that circumscribes the control element. **LENGTH** is defined as the side most closely associated with the element x-axis as shown in the adjoining figure.



Input Data Entry: **DCONBKE** Euler Buckling Constraint Definition

Description: Defines an Euler buckling constraint of the form:

$$\text{Lower Bound: } g_{lower} = \left(\frac{\lambda_{REQ}}{\lambda}\right) - 1.0 \leq 0.0 \quad \text{for } \lambda \geq \lambda_{REQ}$$

or:

$$\text{Upper Bound: } g_{upper} = 1.0 - \left(\frac{\lambda_{REQ}}{\lambda}\right) \leq 0.0 \quad \text{for } \lambda < \lambda_{REQ}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONBKE	SID	ETYPE	EID	LENGTH	BCTYPE	CTYPE	λ_{REQ}		CONT
CONT	RSQR	ALPHA							

DCONBKE	25	BAR	101	1.5	FIX-FIX	LOWER	3.65		

Field	Contents
SID	Euler buckling constraint set identification. (Integer > 0)
ETYPE	Euler buckling control element type (Character) selected from: **BAR** **ROD**
EID	Control element identification number. (Integer > 0)
LENGTH	Rod buckling length in consistant length units. (Real > 0.0 or blank) (See Remark 2)
BCTYPE	Boundary conditions for control element. (Character) (See Remark 3)
CTYPE	Constraint type: either **LOWER** for lower bound or **UPPER** for upper bound. (Character)
λ_{REQ}	Buckling constraint value. (Real, Default = 1.0)
RSQR ALPHA	Parameters which define inertia linking when **ETYPE** is the **ROD** element. (Real or blank) (See Remark 4)

Remarks:

1. Buckling constraints are selected in Solution Control with the discipline option:

   DCON = *sid*

2. If the **LENGTH** is omitted, the corresponding value will be computed from the length of the control element.

3. The boundary condition types are defined in the following table:

BCTYPE	**Boundary Condition**
PIN-PIN	Pin connected at both ends.
PIN-FIX	Pinned at one end, fixed at the other.
FIX-FIX	Fixed at both ends.
FREE-COL	A free column: one end fixed, the other free.

Only **PIN-PIN** may be used for a **ROD** element, while all types may apply to the **BAR**.

4. The inertia is computed from the relation:

$$\mathbf{I} = \mathbf{RSQR} \times \mathbf{AREA}^{\mathbf{ALPHA}}$$

where **AREA** is the area of the control element. If not specified, a solid circular cross-section is assumed.

Input Data Entry: **DCONCLA**

Description: Defines a flexible lift curve slope constraint of the form:

$CLA \leq$ **CLAREQ** or $CLA \geq$ **CLAREQ**

where,

$$CLA = \frac{(C_{l_\alpha})_f}{(C_{l_\alpha})_r}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONCLA	SID	CTYPE	CLAREQ						
DCONCLA	25	UPPER	0.8						

Field	Contents
SID	Aerodynamic set identification for the imposed constraint (Integer > 0)
CTYPE	Constraint type: either **UPPER** for upper bound or **LOWER** for lower bound (Character, Default = **LOWER**)
CLAREQ	Required flexible-to-rigid lift curve slope (Real ≠ 0.0)

Remarks:

1. Displacement constraints are selected in Solution Control with the discipline option:

   DCON=SID

2. A **LOWER** bound constraint excludes all values to the left of **CLAREQ** on a real number line, while an **UPPER** bound constraint excludes all values to the right, irrespective of the sign of **CLAREQ**.

Input Data Entry: **DCONDSP**

Description: Defines a deflection constraint of the form:

$$\sum_j A_j u_j \leq \delta \text{ all (}\textbf{UPPER BOUND}\text{) or } \sum_j A_j u_j \geq \delta \text{ (}\textbf{LOWER BOUND}\text{)}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONDSP	CTSET	DCID	CTYPE	DALL	LABEL	G	C	A	CONT
CONT		G	C	A	G	C	A	–etc–	

DCONDSP	1	10	LOWER	-2.3	TIP	32	3	2.0	ABC
+BC		7	3	-4.0					

Field	Contents
CTSET	Constraint set identification number (Integer > 0)
DCID	Constraint identification number (Integer > 0)
CTYPE	Constraint type, either **UPPER** or **LOWER** bound (Character, Default = **UPPER**)
DALL	Allowable displacement (Real)
LABEL	User specified label to identify constraint (Character)
G	Grid identification (Integer > 0)
C	Component number—any one of digits 1 through 6
A	Real coefficient (Real ≠ 0.0)

Remarks:

1. Displacement constraints are selected in Solution Control with the discipline option:

   DCON=CTSET

   The **CTSET** is the constraint set identification number and **DCID** is an arbitrary constraint identifier supplied by the user. All **DCONDSP** that share the same **CTSET** and **DCID** will form one constraint equation.

2. Both upper and lower bounds on the deflections can be specified by this entry. For example, if constraints of the form $|u| \leq 2.0$ are to be imposed, one **DCONDSP** entry would use **CTYPE** = **UPPER**, **DALL** = 2.0, G = 32, C = 3, A = 1.0 while a second entry would use **CTYPE** = **LOWER**, **DALL** = -2.0, G = 32, C = 3, A = 1.0.

3. Twist constraints can be specified by differencing two displacements while camber constraints can be expressed as a weighted sum of three displacements.

4. Any number of continuation entries are permitted.

5. A **LOWER** bound constraint excludes all values to the left of **DALL** on a real number line, while an **UPPER** bound constraint excludes all values to the right, irrespective of the sign of **DALL**.

Input Data Entry **DCONEP** Principal Strain Constraint Definition

Description: Defines a principal strain constraint by specifying the identification numbers of constrained elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONEP	SID	ST	SC	SS	ETYPE	LAYRNUM	EID1	EID2	CONT
CONT	EID3	EID4	-etc-						

DCONEP	100	1.-2	-1.-2	1.-2	BAR		101	102	ABC
+BC	107	108	142						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONEP	SID	ST	SC	SS	ETYPE	LAYRNUM	EID1	THRU	CONT
CONT	EID2								

Field	Contents
SID	Strain constraint set identification (Integer > 0)
ST	Principal strain limit in tension (Real > 0.0)
SC	Principal strain limit in compression (Real, Default = **ST**)
SS	Principal strain limit in shear (Real > 0.0)
ETYPE	Element type (Character) selected from: **BAR ROD** **SHEAR QDMEM1 TRMEM QUAD4 TRIA3**
LAYRNUM	Layer number of a composite element (Integer > 0 or Blank)
EIDi	Element identification numbers (Integer > 0)

Remarks:

1. Strain constraints are selected in Solution Control with the discipline option:

   STRAIN=sid

2. If the alternate form is used, **EID2** must be greater than or equal to **EID1**. Elements in the range which do not exist are ignored.

3. The shear strain limit, **SS**, is used only with the **SHEAR** element.

4. The strain limit for compression, **SC**, is always treated as a negative value regardless of the sign of the input value.

5. **LAYRNUM** is only used if the element is composed of a composite material defined with **PCOMP** Bulk Data entries.

Input Data Entry **DCONEPM** Principal Strain Constraint Definition

Description: Defines a principal strain constraint by specifying material identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONEPM	SID	ST	SC	SS	MID1	MID2	MID3	MID4	CONT
CONT	MID5	MID6	-etc-						

DCONEPM	100	1.-2	-1.-2	1.-2	8888	9999	1	99	ABC
+BC	111	123							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONEPM	SID	ST	SC	SS	MAT1	THRU	MAT2		

Field	Contents
SID	Strain constraint set identification (Integer > 0)
ST	Principal strain limit in tension (Real > 0.0)
SC	Principal strain limit in compression (Real, Default = **ST**)
SS	Principal strain limit in shear (Real > 0.0)
MIDi	Material identification numbers (Integer > 0)

Remarks:

1. Strain constraints are selected in Solution Control with the discipline option:

   STRAIN=sid

2. If the alternate form is used, **MID2** must be greater than or equal to **MID1**. Material properties in the range which do not exist are ignored.
3. The shear strain limit, **SS**, is used only with the **SHEAR** element.
4. The strain limit for compression, **SC**, is always treated as a negative value regardless of the sign of the input value.

Input Data Entry **DCONEPP** Principal Strain Constraint Definition

Description: Defines a principal strain constraint by specifying element property identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONEPP	SID	ST	SC	SS	PTYPE	LAYRNUM	PID1	PID2	CONT
CONT	PID3	PID4	-etc-						

DCONEPP	100	1.-2	-1.-2	1.-2	PBAR		100	200	ABC
+BC	300	400	500						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONEPP	SID	ST	SC	SS	PTYPE	LAYRNUM	PID1	THRU	ABC
+BC	PID2								

Field	Contents
SID	Strain constraint set identification (Integer > 0)
ST	Principal strain limit in tension (Real > 0.0)
SC	Principal strain limit in compression (Real, Default = **ST**)
SS	Principal strain limit in shear (Real > 0.0)
PTYPE	Property type (Character) selected from: **PBAR** **PROD** **PSHEAR** **PQDMEM1** **PTRMEM** **PSHELL** **PCOMP** **PCOMP1** **PCOMP2**
LAYRNUM	Layer number of a composite element (Integer > 0 or Blank)
PIDi	Property identification numbers (Integer > 0)

Remarks:

1. Strain constraints are selected in Solution Control with the discipline option:

   STRAIN=sid

2. If the alternate form is used, **PID2** must be greater than or equal to **PID1**. Property identification numbers in the range which do not exist are ignored.

3. The shear strain limit, **SS**, is used only with the **SHEAR** element.

4. The strain limit for compression, **SC**, is always treated as a negative value regardless of the sign of the input value.

5. **LAYRNUM** is only used if the element is composed of a composite material defined with **PCOMP** Bulk Data entries.

Input Data Entry: **DCONF** Functional Design Constraint

Description: Define one or more synthetic response constraints or a synthetic objective function.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONF	SID	LNAME	FNAME						CONT
CONT	ARG1	VAL1	ARG2	VAL2	ARG3	VAL3	-etc-		

DCONF	101		ZETA						+DCN1
+DCN1	MACH	0.8	DENS	0.8	MODE	1	VELO	600.0	

Field	Contents
SID	Set Identification number selected by Solution Control (See Remark 1). (Integer > 0)
LNAME	Optional User-defined label for the design constraint function. (Character or blank)
FNAME	The name of a function defined in the Functions packet. (Character)
ARG_i	The name of an argument, as given in the Functions packet, defined in the named function, **FNAME**. (Character)
VAL_i	The value of the parameter **ARGi** to be used in the named function, **FNAME**. (Integer or Real)

Remarks:

1. The **DCONF** entry is selected in Solution Control with one of the two options:

   DCFUNCTION = *sid*
   or
   OBJECT = *sid*

   of the **OPTIMIZE** command, and/or by the option:

   DCFUNCTION = *sid*

   on the discipline commands **STATICS**, **MODES**, **SAERO** and **FLUTTER**. The following example computes ζ for a mach value of 0.8, a density value of 0.8, a mode index of 1 and a velocity of 600.0.

$$\zeta = \left[\left(\frac{\mathrm{Re}\,(p)}{\mathrm{Im}\,(p)} \right)^2 + \mathrm{Re}\,(p)^2 \right]^{1/2}$$

; where p is the complex flutter eigenvalue.

The solution control packet references the functional design constraint, 101, in the Bulk Data Packet for the FLUTTER discipline of boundary condition 1.

```
ANALYZE
   ...
   BOUNDARY SPC = 1
      FLUTTER (..., DCFUNCTION=101, ...)
   ...
END
```

The Function Packet defines the function specification for computing the design constraint $\zeta > 0.15$.

```
FUNCTIONS
   ...
   ZETA(mach, dens, mode, velo )=
                1.0 - ( FDAMP( ZETA ,mach, dens, mode, velo ) / 0.15)
   ...
ENDFUNC
```

The Bulk Data Packet defines values for the **MACH**, **DENS**, **MODE**, and **VELO** arguments, for function design constraint 101 which points to the function, **ZETA**, in the Functional Packet.

```
BEGIN BULK
...
DCONF,101,,ZETA,,,,,,+DCN1
+DCN1,MACH,0.8,DENS,0.8,MODE,1,VELO,600.0
...
ENDDATA
```

2. **ARGi** and **VALi** must be defined together. They represent, by name, the substitution parameters for the function **FNAME**. The following example computes the normal stress in the element's X direction for element 1. The Function Packet defines the function specification for recovering the allowable normal stress in the element's X direction.

```
FUNCTIONS
   ...
   VALUE(eid,allow)= ( STRESS(eid,SIGX) / allow ) - 1.0
   ...
ENDFUNC
```

The Bulk Data Packet defines the element identification and references design constraint 101 which links the design constraint, **VALUE**, to the Functional Packet.

```
BEGIN BULK
...
DCONF,101,EID1,VALUE,,,,,,+DCN1
+DCN1,EID,1,ALLOW,57.0+3
...
ENDDATA
```

3. The **DCONF** entry must uniquely define each argument to the named function and constitutes one or more references to the function **FNAME**. The following example computes the normal stress in the element's X direction for elements 1 and 2. The Function Packet defines the function specification for recovering the allowable normal stress in the element's X direction.

```
FUNCTIONS
   ...
   VALUE(eid,allow)= ( STRESS(eid,SIGX) / allow ) - 1.0
   ...
ENDFUNC
```

The Bulk Data Packet defines two design constraint function requests for the elements 1 and 2, and references design constraint 101 which links the design constraint, **VALUE**, to the Functional Packet.

```
BEGIN BULK
...
DCONF,101,EID1,VALUE,,,,,+DCN1
+DCN1,EID,1,ALLOW,60.+3
DCONF,101,EID2,VALUE,,,,,+DCN1
+DCN1,EID,2,ALLOW,60.+3
...
ENDDATA
```

More than one constraint can be created by a single DCONF entry if list identification arguments are used:

```
FUNCTIONS
   ...
   VALUE2(list,allow)= ( STRESS(ELEMLIST(list),SIGX) / allow ) - 1.0
   ...
ENDFUNC
BEGIN BULK
...
DCONF,101,EID1,VALUE2,,,,,+DCN1
+DCN1,LIST,101,ALLOW,60.+3
...
ELEMLIST,101,QUAD4,1,2
ENDDATA
```

4. **VALi** must be of the type, either integer or real required by the function **FNAME**.

Input Data Entry: **DCONFLT** Flutter Constraint Definition

Description: Defines a flutter constraint in the form of a table:

$$\frac{\gamma - \gamma_{REQ}}{GFACT} \leq 0.0$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONFLT	SID	VTYPE	GFACT	V1	GAM1	V2	GAM2		CONT
CONT	V3	GAM3	V4	GAM4	-etc-				

DCONFLT	100	EQUIV	0.1	0.0	0.0	35.	0.05		

Field	Contents
SID	Constraint set identification, the constraints are referenced by the design constraint id in Solution Control (Integer > 0)
VTYPE	Nature of the velocity referred to in the table. Either **TRUE** for true velocity or **EQUIV** for equivalent air speed. Default = **TRUE**.
GFACT	Constraint scaling factor (Real > 0.0, Default = 0.10)
Vi	Velocity value (Real ≥ 0.0)
GAMi	Required damping value (Real)

Remarks:

1. Flutter constraints are selected in Solution Control with the discipline option:

   DCON=SID

2. A negative value of **GAMi** refers to a stable system.
3. The **Vi** must be in either ascending or descending order.
4. Linear interpolation is used to determine **GAMA** for a given velocity.
5. At least two pairs must be entered.
6. Jumps between two points (**Vi** = **Vi+1**) are allowed, but not at the end points. If the jump point is used, the average of the two **GAMi** will be returned.

Input Data Entry: **DCONFRQ**

Description: Defines a frequency constraint of the form:

$$f \leq f_{all} \text{ or } f \geq f_{all}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONFRQ	SID	MODE	CTYPE	FRQALL					
DCONFRQ	3	1	LOWER	6.0					

Field	Contents
SID	Constraint set identification (Integer > 0)
MODE	Modal number of the frequency to be constrained (Integer > 0)
CTYPE	Constraint type: either **UPPER** for upper bound or **LOWER** for lower bound (Character, Default = **LOWER**)
FRQALL	Frequency constraint (in Hz.).

Remarks:

1. More than one constraint can be placed on a mode allowing specification of pseudo-equality constraints.

Input Data Entry **DCONFT** Fiber/Transverse Strain Constraint Definition

Description: Defines fiber/transverse strain constraints for composite elements by specifying the identification numbers of constrained elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONFT	SID	EFT	EFC	ETT	ETC	ETYPE	LAYRNUM	EID1	CONT
CONT	EID2	EID3	-etc-						

DCONFT	100	1.-2	-1.-2	1.-3	-1.-3	QUAD4	1	101	ABC
+BC	102	110							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONFT	SID	EFT	EFC	ETT	ETC	ETYPE	LAYRNUM	EID1	CONT
CONT	THRU	EID2							

Field	Contents
SID	Strain constraint set identification (Integer > 0)
EFT	Tensile strain limit in the fiber direction (Real > 0.0)
EFC	Compressive strain limit in the fiber direction (Real, Default = **EFT**)
ETT	Tensile strain limit in the transverse direction (Real > 0.0)
ETC	Compressive strain limit in the transverse direction (Real, Default = **ETT**)
ETYPE	Element type (Character) selected from: **QDMEM1** **TRMEM** **QUAD4** **TRIA3**
LAYRNUM	The layer number of a composite element (Integer > 0, or blank)
EIDi	Element identification numbers (Integer > 0)

Remarks:

1. Strain constraints are selected in Solution Control with the discipline option:

   STRAIN=sid

2. Fiber/transverse strain constraints may only be applied to elements defined using composite materials.
3. If the alternate form is used, **EID2** must be greater than or equal to **EID1**. Elements in the range which do not exist are ignored.
4. The strain limits for compression, **EFC** and **ETC**, are always treated as negative values regardless of the signs of the input values.

Input Data Entry **DCONFTM** Fiber/Transverse Strain Constraint Definition

Description: Defines fiber/transverse strain constraints for composite elements by specifying material identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONFTM	SID	EFT	EFC	ETT	ETC	MID1	MID2	MID3	CONT
CONT	MID4	MID5	-etc-						

DCONFTM	100	1.-2	-1.-2	1.-3	-1.-3	11	16	101	ABC
+BC	19	14							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONFTM	SID	EFT	EFC	ETT	ETC	MID1	THRU	MID2	

Field	Contents
SID	Strain constraint set identification (Integer > 0)
EFT	Tensile strain limit in the fiber direction (Real > 0.0).
EFC	Compressive strain limit in the fiber direction (Real, Default = **EFT**)
ETT	Tensile strain limit in the transverse direction (Real > 0.0)
ETC	Compressive strain limit in the transverse direction (Real, Default = **ETT**).
MIDi	Material identification numbers (Integer > 0)

Remarks:

1. Strain constraints are selected in Solution Control with the discipline option:

   STRAIN=sid

2. Fiber/transverse strain constraints may only be applied to elements defined using composite materials.
3. If the alternate form is used, **MID2** must be greater than or equal to **MID1**. Material properties in the range which do not exist are ignored.
4. The strain limits for compression, **EFC** and **ETC**, are always treated as negative values regardless of the signs of the input values.

Input Data Entry **DCONFTP** Fiber/Transverse Strain Constraint Definition

Description: Defines fiber/transverse strain constraints for composite elements by specifying property identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONFTP	SID	EFT	EFC	ETT	ETC	PTYPE	LAYRNUM	PID1	CONT
CONT	PID2	PID3	-etc-						

DCONFTP	100	1.-2	1.-2	2.-3	3.-3	PCOMP	2	100	CONT
CONT	110	120							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONFTP	SID	EFT	EFC	ETT	ETC	ETYPE	LAYRNUM	PID1	CONT
CONT	THRU	PID2							

Field	Contents
SID	Strain constraint set identification (Integer > 0).
EFT	Tensile strain limit in the fiber direction (Real > 0.0)
EFC	Compressive strain limit in the fiber direction (Real, Default = **EFT**)
ETT	Tensile strain limit in the transverse direction (Real > 0.0).
ETC	Compressive strain limit in the transverse direction (Real, Default = **ETT**)
PTYPE	Property type (Character) selected from: **PCOMP** **PCOMP1** **PCOMP2**.
LAYRNUM	The layer number of a composite element (Integer > 0 or blank)
PIDi	Property identification numbers (Integer > 0)

Remarks:

1. Strain constraints are selected in Solution Control with the discipline option:

   STRAIN=sid

2. Fiber/transverse strain constraints may only be applied to elements defined using composite materials.
3. If the alternate form is used, **PID2** must be greater than or equal to **PID1**. Properties in the range which do not exist are ignored.
4. The strain limits for compression, **EFC** and **ETC**, are always treated as negative values regardless of the signs of the input values.

Input Data Entry: **DCONLAM** Composite laminate composition constraint.

Description: Defines a constraint on the relative thickness of a *ply* that is part of a *laminate*. The constraint is of the form:

$$\frac{\%_{req}}{100} - \frac{t_{ply}}{t_{lam}} \leq 0 \quad \text{(lower bound)}$$

$$\frac{t_{ply}}{t_{lam}} - \frac{\%_{req}}{100} \leq 0 \quad \text{(upper bound)}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONLAM	CTYPE	%REQ	PLYNUM	PLYSET	LAM	SID	SID	SID	CONT
CONT	SID	SID	-etc-						

DCONLAM	UPPER	40.0		100	ALL	1000	1001		

Field	Contents
CTYPE	Constraint type: either **UPPER** for upper bound or **LOWER** for lower bound. (Character, Default = **UPPER**)
%REQ	Minimum (lower bound) or maximum (upper bound) **PERCENTAGE** (0.0 to 100.0) of the total laminate thickness that is to be made up of the ply thickness. (see Remark 2) (Real > 0.0)
PLYNUM	Single ply number (numbered in the order used on the **PCOMPi**) that constitutes the ply thickness. Only one of **PLYNUM** or **PLYSET** may be used. (Integer > 0 or blank)
PLYSET	Set identification number of one or more **PLYLIST** bulk data entries naming a set of plies whose ***summed*** thicknesses constitute the *ply* thickness in the constraint. Only one of **PLYNUM** or **PLYSET** may be used. (Integer > 0 or blank)
LAM	The character string **ALL** ***or*** the set identification number of one or more **PLYLIST** entries naming a set of plies whose ***summed*** thicknesses constitute the *laminate* thickness in the constraint. If **ALL**, the *laminate* is defined to be all the layers on the **PCOMP**s of the elements selected by **SIDi**. (Character = **ALL** or Integer > 0, Default = **ALL**)
SID	Set identification of one or more **ELEMLIST** entries that define the set of composite elements to which this composition constraint will be applied. (Integer > 0 or blank)

Remarks:

1. One and only one of either **PLYNUM** or **PLYSET** must be given.
2. The definition of *ply* and *laminate* thickness can vary from entry to entry. ***If*** **PLYNUM** is used to define t_{ply} that one layer constitutes a ply; otherwise t_{ply} is the ***sum*** of the layer thicknesses of all the layers listed in **PLYSET.**

Similar rules are applied for tlam. If **ALL** is used, every layer of the element is used to compute tlam (including undesigned layers-see Remark 3); otherwise the ***summed*** thicknesses of the layers specified by the **PLYLIST** set will be used. As a result, there is no real distinction between a *ply* thickness and a *laminate* thickness. Typically, the *ply* will be a subset of the layers that define the *laminate*, but that is not a requirement.

3. If this constraint is applied to a composite element with undesigned layers, these layers may be freely included in the layer(s) composing the *ply* and/or the layer(s) composing the *laminate*. The only restriction is that at least one layer in the *ply* must be a local design variable ***and*** at least one layer in the *laminate* must be a local design variable.

Input Data Entry **DCONLIST** Design Constraint List

Description: Defines a list of design constraints for which constraint value output and/or constraint gradient output are desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONLIST	SID	TYPE	NRFAC	EPS					
DCONLIST	1000	DISP	0.6	-.05					

Field	Contents
SID	Set identification number (Integer > 0)
TYPE	The design constraint type. One of the following: **FREQ** frequency **FLUT** flutter **DISP** displacement **VMISES** Von Mises **TSAIWU** Tsai-Wu **STRAIN** strain **THICK** thickness **EFF** aeroelastic effectiveness **SCF** stability coefficient **TRIM** trim **ALL** all of the above **OTHER** all EXCEPT the above The Default value is **ALL**
NRFAC	Constraint retention factor for math programming methods. At least **NRFAC** * (number of design variables) constraints will be considered active. (Real > 0.0, Default = 3.0)
EPS	Constraint retention parameter in which all constraints having a value greater than **EPS** will be considered active. (Real, Default = – 0.1)

Remarks:

1. **NRFAC** and **EPS** control the number of constraints that are selected for print and punch output. For constraint gradients, only those considered active by the global constraint screening algorithm (**NRFAC** and **EPS** from the **OPTIMIZE** command in Solution control) are available to be selected.
2. More than one **DCONLIST** with the same set identification number may be used to select subsets of different constraint types.

Input Data Entry: **DCONLMN** Composite laminate minimum gauge constraint.

Description: Defines a lower bound constraint on the total thickness of all or part of the layers of a composite element. The constraint is of the form:

$$1.0 - \frac{t_{lam}}{t_{\min}} \leq 0$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONLMN	MINTHK	LAM	SID	SID	SID	SID	SID	SID	CONT
CONT	SID	SID	-etc-						

DCONLMN	0.20	ALL	1001	1002				,	

Field	Contents
MINTHK	Minimum *laminate* thickness. (Real > 0.0, Default = 10^{-4})
LAM	The character string **ALL *or*** the set identification number of one or more **PLYLIST** entries naming a set of plies whose ***summed*** thicknesses constitute the *laminate* thickness in the constraint. If **ALL**, the *laminate* is defined to be all the layers on the **PCOMP**s of the elements selected by **SIDi**. (Character = **ALL** or Integer > 0, Default = **ALL**)
SID	Set identification of one or more **ELEMLIST** entries that define the set of composite elements to which this composition constraint will be applied. (Integer > 0 or blank)

Remarks:

1. Because of the generality of the definition of the *laminate*, there is no real distinction between the **DCONLMN** and the **DCONPMN** constraints. Only the defaults are different to allow simple definitions of the common laminate in **DCONLMN (ALL)** or ply (**PLYNUM**) in **DCONPMN**.
2. The definition of *laminate* thickness can vary from entry to entry. If **ALL** is used, every layer of the element is used to compute tlam (including undesigned layers-see Remark 3); otherwise the ***summed*** thicknesses of the layers specified by the **PLYLIST** set will be used.
3. If this constraint is applied to a composite element with undesigned layers, these layers may be freely included in the layer(s) composing the *ply* and/or the layer(s) composing the *laminate*. The only restriction is that at least one layer in the *laminate* must be a local design variable.
4. If the *laminate* is composed of a single layer, this constraint becomes redundant with the **TMIN** entered on the **PCOMPi** field (for shape function linking) or the **VMIN** entered on the **DESELM** or **DESVARP** entry (for physical linking). In this case, the most critical limit will be determined from among all sources (**DCONPMN, DCONLMN, TMIN/VMIN**) and will be used to update the local variable side constraint. The **DCONxxx** entry will then be automatically removed since it will no longer be necessary. A summary of this action will be echoed to the print file.

Input Data Entry: **DCONPMN** Composite element ply minimum gauge constraint.

Description: Defines a lower bound constraint on the total thickness of all or part of the layers of a composite element. The constraint is of the form:

$$1.0 - \frac{t_{ply}}{t_{\min}} \leq 0$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONPMN	MINTHK	PLYNUM	PLYSET	SID	SID	SID	SID	SID	CONT
CONT	SID	SID	-etc-						

DCONPMN	0.010	3		1001	1002			,	

Field	Contents
MINTHK	Minimum *ply* thickness. (Real > 0.0, Default = 10^{-4})
PLYNUM	Single ply number (numbered in the order used on the **PCOMPi**) that constitutes the ply thickness. Only one of **PLYNUM or PLYSET** may be used. (Integer > 0 or blank)
PLYSET	Set identification number of one or more **PLYLIST** bulk data entries naming a set of plies whose ***summed*** thicknesses constitute the *ply* thickness in the constraint. Only one of **PLYNUM or PLYSET** may be used. (Integer > 0 or blank)
SID	Set identification of one or more **ELEMLIST** entries that define the set of composite elements to which this composition constraint will be applied. (Integer > 0 or blank)

Remarks:

1. One and only one of either **PLYNUM or PLYSET** must be given.
2. Because of the generality of the definition of the *ply*, there is no real distinction between the **DCONLMN** and the **DCONPMN** constraints. Only the defaults are different to allow simple definitions of the common laminate in **DCONLMN (ALL)** or ply (**PLYNUM**) in **DCONPMN**.
3. The definition of *ply* thickness can vary from entry to entry. If **PLYNUM** is used to define t_{ply}, that one layer constitutes a ply; otherwise t_{ply} is the ***sum*** of the layer thicknesses of all the layers listed in **PLYSET**.
4. If this constraint is applied to a composite element with undesigned layers, these layers may be freely included in the layer(s) composing the *ply*. The only restriction is that at least one layer in the *ply* must be a local design variable.
5. If the *ply* is composed of a single layer, this constraint becomes redundant with the **TMIN** entered on the **PCOMPi** field (for shape function linking) or the **VMIN** entered on the **DESELM** or **DESVARP** entry (for physical linking). In this case, the most critical limit will be determined from among all sources (**DCONPMN**, **DCONLMN**, **TMIN/VMIN**) and will be used to update the local variable side constraint. The **DCONxxx** entry will then be automatically removed since it will no longer be necessary. A summary of this action will be echoed to the print file.

Input Data Entry **DCONSCF** Stability Derivative Constraint

Description: Defines a constraint on the flexible stability derivative at the reference grid point associated with the force or moment due to a trim parameter or control surface deflection of the form:

$$\left[\frac{\partial C_F}{\partial \delta_{trim}}\right]_{lower} \leq \frac{\partial C_F}{\partial \delta_{trim}} \leq \left[\frac{\partial C_F}{\partial \delta_{trim}}\right]_{upper}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONSCF	SETID	ACCLAB	PRMLAB	CTYPE	PRMREQ	UNITS			
DCONSCF	999	PACCEL	AILERON	LOWER	1.0	RADIANS			

Field	Contents
SETID	Set identification number referenced by the **DCONSTRAINT** Solution Control option of the **SAERO** command. (Integer > 0)
ACCLAB	Alphanumeric string identifying the aerodynamic force or moment by naming the corresponding structural acceleration in a manner consistent with the **TRIM** entry. See Remarks 2 and 4.
PRMLAB	Alphanumeric string identifying a constrained control surface or aeroelastic trim parameter (e.g. **ALPHA** or **PRATE**). See Remarks 3 and 4.
CTYPE	Constraint type; either **UPPER**, for upper bound, or **LOWER** for lower bound. (Character, default=**UPPER**)
PRMREQ	Bound for the stability coefficient. For units, see Remarks 5 and 6. (Real)
UNITS	Units for the stability coefficient. Either **RADIANS** or **DEGREES**. See Remark 6. (Real,Default=**DEGREES**)

Remarks:

1. The **DCONSCF** entry is selected in Solution Control with the **DCONSTRAINT=SETID** option of the **SAERO** command.
2. The **ACCLAB** may refer to any of the **TRIM** Bulk Data entry trim parameters that are **structural accelerations**. Valid trim parameters are **NX**, **NY**, **NZ**, **PACCEL**, **QACCEL**, and **RACCEL**.
3. The **PRMLAB** may refer to **AESURF** or **CONLINK** control surfaces or to any of the **TRIM** entry parameters **except the structural accelerations**. Valid selections are: **PRATE**, **QRATE**, **RRATE**, **ALPHA**, **BETA**, **THKCAM** and any control surface label. Invalid trim parameters are: **NX**, **NY**, **NZ**, **PACCEL**, **QACCEL** and **RACCEL**
4. Any combination of forces or moments and trim parameters/control surfaces may be used on this entry **provided** they have the same symmetry as the associated **TRIM** entry.
   Furthermore, to apply the constraint to the flexible derivative, the degree of freedom corresponding to the force or moment must be supported in the boundary condition. For example, to constrain the pitching moment, **QACCEL**, due to angle of attack, **ALPHA**, the y-rotation of the support point must be on the **SUPPORT** entry for the boundary condition in which the **TRIM** is analyzed.
5. The stability derivatives are nondimensional quantities derived from the flexible forces and moments due to "unit" parameters. The constraint is applied to the nondimensional derivative at the user-de-

fined reference point. To assist the defining **PRMREQ**, the following normalizations are used in ASTROS:

SYMMETRIC DERVIATIVES	FORCES	CONTROL SURFACES	stability coeff = F/(QDP*S)
		RATES	stability coeff = F*2*VO/(QDP*S*C) "unit" rate = unit dimensional rate * C/2*VO
	MOMENTS	CONTROL SURFACES	stability coeff = M/(QDP*S*C)
		RATES	stability coeff = M*4*VO/(QDP*S*C**2) "unit" rate = unit dimensional rate * C/2*VO
ANTISYMMETRIC DERVIATIVES	FORCES	CONTROL SURFACES	stability coeff = F/(QDP*S)
		RATES	stability coeff = F*2*VO/(QDP*S*B) "unit" rate = unit dimensional rate * B/2*VO
	MOMENTS	CONTROL SURFACES	stability coeff = M/(QDP*S*B)
		RATES	stability coeff = M*4*VO/(QDP*S*B**2) "unit" rate = unit dimensional rate * B/2*VO

F and **M** are the dimensional flexible forces and moments for the full vehicle; **S**, **C**, and **B** are the non-dimensional factors from the **AEROS** Bulk Data entry (the inputs are assumed to be for the full vehicle); and **QDP** and **VO** are defined on the **TRIM** Bulk Data entry.

6. **RADIANS** or **DEGREES** refer to the units of the unit control surface deflection or unit rate. **RADIANS** imply the value due to a unit RAD or RAD/S while **DEGREES** imply the value due to a unit DEG or DEG/S. **THKCAM** has no valid angular unit, hence the **UNITS** field is ignored.

7. A **LOWER** bound constraint excludes all values to the left of **PRMREQ** on a real number line, while an **UPPER** bound excludes all values to the right, irrespective of the sign of **PRMREQ**.

Input Data Entry: **DCONSDE** BAR element side constraints

Description: Defines Side constraints on BAR element cross-sectional parameters.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONSDE	DVSYM	TMIN	TMAX	ETYPE	EID1	EID2	EID3	EID4	CONT
CONT	EID5	EID6	-etc-						

DCONSDE	D1	0.1	0.3	BAR	200	205	206		

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONSDE	DVSYM	TMIN	TMAX	ETYPE	EID1	THRU	EID2		

Field	Contents
DVSYM	Character symbol specifying the **PBAR1** cross-sectional parameter. (Remark 1) **D1** **D2** **D3** **D4** **D5** **D6** **D7** **D8** **D9** **D10**
TMIN	Minimum value of the **PBAR1** cross-sectional parameter specified by **DVSYM**. (Real, Default = 0.0001)
TMAX	Maximum value of the **PBAR1** cross-sectional parameter specified by **DVSYM**. (Real, Default = 0.0001)
ETYPE	Character input identifying the element type. Must be: **BAR**
EIDi	Element identification numbers (Integer > 0 or blank)

Remarks:

1. See the **PBAR1** Bulk Data entry for a description of the cross-sectional parameters.

Input Data Entry: **DCONSDL** BAR element side constraints

Description: Defines Side constraints on BAR element cross-sectional parameters by referencing list of elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONSDL	DVSYM	TMIN	TMAX	ELID1	ELID2	ELID3	ELID4	ELID5	CONT
CONT	ELID6	ELID7	-etc-						

DCONSDL	D3	0.001	0.05	99					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONSDL	DVSYM	TMIN	TMAX	ELID1	THRU	ELID2			

Field	Contents
DVSYM	Character symbol specifying the **PBAR1** cross-sectional parameter. (Remark 1) **D1** **D2** **D3** **D4** **D5** **D6** **D7** **D8** **D9** **D10**
TMIN	Minimum value of the **PBAR1** cross-sectional parameter specified by **DVSYM**. (Real, Default = 0.0001)
TMAX	Maximum value of the **PBAR1** cross-sectional parameter specified by **DVSYM**. (Real, Default = 0.0001)
ELIDi	Element list identification numbers (Integer > 0 or blank) (Remark 2)

Remarks:

1. See the **PBAR1** Bulk Data entry for a description of the cross-sectional parameters.
2. Element lists are defined using **ELEMLIST** Bulk Data entries. Only designed BAR elements which reference **PBAR1** property entries are affected.

Input Data Entry: **DCONTH2** Thickness constraints on layers of composite elements

Description: Defines a set of layers for a list of elements linked using **SHAPE** entries for which the ply thickness constraints are to be retained on all design iterations.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTH2	ETYPE	LAYRNUM	LAYRLST	EID	EID	EID	EID	EID	CONT
CONT	EID	EID	-etc-						

DCONTHK	QUAD4		100	101	200	205			

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONTH2	ETYPE	LAYRNUM	LAYRLST	EID	THRU	EID			

Field	Contents
ETYPE	Character input identifying the element type. One of the following: **QUAD4** **TRIA3** **QDMEM1** **TRMEM**
LAYRNUM	Layer number of the layer(s) to be retained. The given layer will be retained for each element in the list of elements (Integer > 0 or blank, See Remark 1)
LAYRLST	Set identification number of a **PLYLIST** bulk data entry naming a set of plies to be retained as active for each element. (Integer > 0 or blank, See Remark 1)
EID	Element identification number (Integer > 0 or blank)

Remarks:

1. One and only one of either **LAYRNUM** or **LAYRLST** must be given. Noncomposite elements must be called out on **DCONTHK** entries.
2. The purpose of this bulk data list is to ensure that adequate physical move limits are retained in optimization with shape function design variable linking without requiring retention of all move limits. For problems with large numbers of local variables using shape functions, the move limits often cause too many minimum thickness constraints (see Remark 3) to be retained in the optimization task. Using this bulk data entry ***or*** its noncomposite counterpart **DCONTHK** to name "critical" minimum gauge constraints (see Remark 4) will cause only the named elements' thickness constraints to be computed and retained. All layers of composite elements named on **DCONTHK** will be retained. Note that all thickness constraints for an element will always be computed irrespective of the **DCONTHK** entries, but may be deleted in the constraint deletion.
3. The global design variable in shape function linking is non-physical and no reasonable restriction for a global variable move limit (side constraint) can be defined. Therefore, constraints on the local design variables controlled by shape functions are generated by ASTROS to ensure that the design is reasonable (ie, nonnegative thicknesses).

4. The **DCONTH2** entry should select a minimum number of elements linked to shape functions that will enable the optimizer to select physically reasonable designs without retaining all the minimum thickness constraints (potentially a very large number). Typically, this means N+1 elements spread over the range of the shape function (e.g. span or chord) where N is the order of the shape (**N=O**, **UNIFORM: N=1**, **LINEAR**, etc.).

Input Data Entry: **DCONTH3**

Description: Defines a set of BAR element cross-sectional properties for a list of elements which are linked using **SHAPE** entries, and for which side constraints are to be retained for all design iterations.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTH3	ETYPE	DVSYM	EID1	EID2	EID3	EID4	EID5	EID6	CONT
CONT	EID7	EID8	-etc-						

DCONTH3	BAR	D1	100	101					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONTH3	ETYPE	DVSYM	EID1	THRU	EID2				

Field	Contents
ETYPE	Character input identifying the element type. Must be **BAR**.
DVSYM	Symbol selecting one of the **PBAR1** cross-sectional parameters. (Character) **D1** **D2** **D3** **D4** **D5** **D6** **D7** **D8** **D9** **D10**
EIDi	Element identification numbers (Integer > 0 or blank)

Remarks:

1. The purpose of this bulk data list is to ensure that adequate physical move limits are retained in optimization with shape function design variable linking without requiring retention of all move limits. For problems with large numbers of local variables using shape functions, the move limits often cause too many minimum thickness constraints (see Remark 3) to be retained in the optimization task. Using this bulk data entry to name "critical" minimum gauge constraints (see Remark 4) will cause only the named elements' thickness constraints to be computed and retained. Note that all thickness constraints for an element will always be computed irrespective of the **DCONTH3** entries, but may be deleted in the constraint deletion.

2. The global design variable in shape function linking is non-physical and no reasonable restriction for a global variable move limit (side constraint) can be defined. Therefore, constraints on the local design variables controlled by shape functions are generated by ASTROS to ensure that the design is reasonable.

3. The global design variable in shape function linking is non-physical and no reasonable restriction for a global variable move limit (side constraint) can be defined. Therefore, constraints on the local design variables controlled by shape functions are generated by ASTROS to ensure that the design is reasonable (ie, nonnegative thicknesses).

4. The **DCONTH2** entry should select a minimum number of elements linked to shape functions that will enable the optimizer to select physically reasonable designs without retaining all the minimum thickness constraints (potentially a very large number). Typically, this means N+1 elements spread over the range of the shape function (e.g. span or chord) where N is the order of the shape (**N=0**, **UNIFORM: N=1**, **LINEAR**, etc.).

Input Data Entry: **DCONTHK** Thickness constraints on elements

Description: Defines a list of elements linked using **SHAPE** entries for which thickness constraints are to be retained on all design iterations.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTHK	ETYPE	EID	EID	EID	EID	EID	EID	EID	CONT
CONT	EID	EID	-etc-						

DCONTHK	QDMEM1	100	101	200	205				

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONTHK	ETYPE	EID	THRU	EID					

Field	Contents
ETYPE	Character input identifying the element type. One of the following: **BAR** **QUAD4** **CONM2** **ROD** **ELAS** **SHEAR** **MASS** **TRIA3** **QDMEM1** **TRMEM**
EID	Element identification number (Integer > 0 or blank)

Remarks:

1. The purpose of this bulk data list is to ensure that adequate physical move limits are retained in optimization with shape function design variable linking without requiring retention of all move limits. For problems with large numbers of local variables using shape functions, the move limits often cause too many minimum thickness constraints (see Remark 2) to be retained in the optimization task. Using this bulk data entry OR its composite counterpart **DCONTH2** to name "critical" minimum gauge constraints (see Remark 3) will cause only the named elements' thickness constraints to be computed and retained. All layers of composite elements named on **DCONTHK** will be retained.
   NOTE that all elements' thickness constraints will always be computed irrespective of the **DCONTHK** entries, but may be deleted in the constraint deletion.

2. The global design variable in shape function linking is non-physical and no reasonable restriction for a global variable move limit (side constraint) can be defined. Therefore, constraints on the local design variables controlled by shape functions are generated by ASTROS to ensure that the design is reasonable (ie, nonnegative thicknesses).

3. The **DCONTHK** entry should select a minimum number of elements linked to shape functions that will enable the optimizer to select physically reasonable designs without retaining all the minimum thickness constraints (potentially a very large number). Typically, this means N+1 elements spread over the range of the shape function (e.g. span or chord) where N is the order of the shape (N=O, UNIFORM: N=1, LINEAR, etc.). Use **DCONTH2** for composite elements in which linking across layers may allow certain layers to be omitted from the retention set.

Input Data Entry **DCONTRM** Aeroelastic Trim Parameter Constraint

Description: Defines a trim parameter constraint of the form:

$$\delta_{trim} \leq \delta_{trim_{Req}} \quad \text{or} \quad \delta_{trim} \geq \delta_{trim_{Req}}$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTRM	SETID	PRMLAB	CTYPE	PRMREQ					
DCONTRM	100	AILERON	UPPER	25.0					

Field	Contents
SETID	Set identification number referenced by the **DCONSTRAINT** Solution Control command. (Integer > 0)
PRMLAB	Alphanumeric string identifying a constrained control surface or aeroelastic trim parameter (e.g. **ALPHA** or **PRATE**). (See Remark 2.)
CTYPE	Constraint type; either **UPPER**, for upper bound, or **LOWER** for lower bound. (Character, Default = **UPPER**)
PRMREQ	Bound for the trim parameter. For units, see Remark 3. (Real)

Remarks:

1. The **DCONTRM** entry is selected in Solution Control with the **DCONSTRAINT=SETID** option of the **SAERO** command.
2. The **PRMLAB** may refer to **AESURF** or **CONLINK** control surfaces or to any of the **TRIM** entry parameters, **NX**, **NY**, **NZ**, **PACCEL**, **QACCEL**, **RACCEL**, **PRATE**, **QRATE**, **RRATE**, **ALPHA**, or **BETA**. The only requirement is that the constrained control surface must be declared on the **TRIM** entry. The user will be **warned** if trim parameters not on the **TRIM** entry are constrained (since these parameters are fixed, they are design invariant).
3. The units for control surface deflections are degrees. For rates, the units should be radians/sec. For linear accelerations **NX**, **NY**, **NZ**, the units should be consistent, (length/sec/sec) or, if a **CONVERT,MASS** entry was used, should be dimensionless. Angular accelerations should be in radians/sec/sec.
4. A **LOWER** bound constraint excludes all values to the left of **PRMREQ** on a real number line, while an **UPPER** bound excludes all values to the right, irrespective of the sign of **PRMREQ**.

Input Data Entry **DCONTW** Tsai-Wu Stress Constraint Definition

Description: Defines Tsai-Wu stress constraints by specifying the identification numbers of constrained elements

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTW	SID	XT	XC	YT	YC	SS	F12	ETYPE	CONT
CONT	LAYRNUM	EID1	EID2	EID3	-etc-				

DCONTW	100	1.+6	-1.+6	1.+4	-1.+4	1.5+3		QUAD4	ABC
+BC	1	102	106	110					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONTW	SID	XT	XC	YT	YC	SS	F12	ETYPE	CONT
CONT	LAYRNUM	EID1	THRU	EID2					

Field	Contents
SID	Stress constraint set identification (Integer > 0)
XT	Tensile stress limit in the longitudinal direction (Real > 0.0)
XC	Compressive stress limit in the longitudinal direction (Real, Default = **XT**)
YT	Tensile stress limit in the transverse direction (Real > 0.0)
YC	Compressive stress limit in the transverse direction (Real, Default = **YT**)
SS	Shear stress limit for in-plane stress (Real > 0.0)
F12	Tsai-Wu interaction term (Real)
ETYPE	Element type (Character) selected from: **QDMEM1** **TRMEM** **QUAD4** **TRIA3**
LAYRNUM	The layer number of a composite element (Integer > 0 or blank)
EIDi	Element identification numbers (Integer > 0)

Remarks:

1. Stress constraints are selected in Solution Control with the discipline option:

   STRESS=sid

2. If the alternate form is used, **EID2** must be greater than or equal to **EID1**. Elements in the range which do not exist are ignored.

3. The strain limits for compression, **XC** and **YC**, are always treated as negative values regardless of the sign of the input values.

4. **LAYRNUM** is only used if the element is composed of a composite material defined with **PCOMP** Bulk Data entries.

Input Data Entry **DCONTWM** Tsai-Wu Stress Constraint Definition

Description: Defines Tsai-Wu stress constraints by specifying material identification numbers

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTWM	SID	XT	XC	YT	YC	SS	F12	MID1	CONT
CONT	MID2	MID3	MID4	-etc-					

DCONTWM	100	1.+6	-1.+6	1.+4	-1.+4	1.5+3		101	ABC
+BC	102	200	310						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONTWM	SID	XT	XC	YT	YC	SS	F12	MID1	CONT
CONT	THRU	MID2							

Field	Contents
SID	Stress constraint set identification (Integer > 0)
XT	Tensile stress limit in the longitudinal direction (Real > 0.0)
XC	Compressive stress limit in the longitudinal direction (Real, Default = **XT**)
YT	Tensile stress limit in the transverse direction (Real > 0.0)
YC	Compressive stress limit in the transverse direction (Real, Default = **YT**)
SS	Shear stress limit for in-plane stress (Real > 0.0)
F12	Tsai-Wu interaction term (Real)
MIDi	Material identification numbers (Integer > 0)

Remarks:

1. Stress constraints are selected in Solution Control with the discipline option:

   STRESS=sid

2. If the alternate form is used, **MID2** must be greater than or equal to **MID1**. Materials in the range which do not exist are ignored.

3. The stress limits for compression, **XC** and **YC**, are always treated as negative values regardless of the sign of the input values.

Input Data Entry **DCONTWP** Tsai-Wu Stress Constraint Definition

Description: Defines Tsai-Wu stress constraints by specifying element property identification numbers

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONTWP	SID	XT	XC	YT	YC	SS	F12	PTYPE	CONT
CONT	LAYRNUM	PID1	PID2	PID3	-etc-				

DCONTWP	100	1.+6	-1.+6	1.+4	-1.+4	1.5+3		PCOMP	ABC
+BC	100	200	300						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONTWP	SID	XT	XC	YT	YC	SS	F12	PTYPE	CONT
CONT	LAYRNUM	PID1	THRU	PID2					

Field	Contents
SID	Stress constraint set identification (Integer > 0)
XT	Tensile stress limit in the longitudinal direction (Real > 0.0)
XC	Compressive stress limit in the longitudinal direction (Real, Default = **XT**)
YT	Tensile stress limit in the transverse direction (Real > 0.0)
YC	Compressive stress limit in the transverse direction (Real, Default = **YT**)
SS	Shear stress limit for in-plane stress (Real > 0.0)
F12	Tsai-Wu interaction term (Real)
PTYPE	Property type (Character) selected from: **PQDMEM1 PTRMEM PSHELL PCOMP PCOMP1 PCOMP2**
LAYRNUM	The layer number of a composite element (Integer > 0 or blank)
PIDi	Property identification numbers (Integer > 0)

Remarks:

1. Stress constraints are selected in Solution Control with the discipline option:

   STRESS=sid

2. If the alternate form is used, **PID2** must be greater than or equal to **PID1**. Properties in the range which do not exist are ignored.

3. The stress limits for compression, **XC** and **YC**, are always treated as negative values regardless of the sign of the input values.

4. **LAYRNUM** is only used if the element is composed of a composite material defined with **PCOMP** Bulk Data entries.

Input Data Entry **DCONVM** Von-Mises Stress Constraint Definition

Description: Defines a Von-Mises stress constraint by specifying the identification numbers of constrained elements

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONVM	SID	ST	SC	SS	ETYPE	LAYRNUM	EID1	EID2	CONT
CONT	EID3	EID4	-etc-						

DCONVM	100	1.+6	-1.+6	1.+4	BAR		101	102	ABC
+BC	107	108	142						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONVM	SID	ST	SC	SS	ETYPE	LAYRNUM	EID1	THRU	CONT
CONT	EID2								

Field	Contents
SID	Stress constraint set identification (Integer > 0)
ST	Tensile stress limit (Real > 0.0 or blank)
SC	Compressive stress limit (Real, Default = **ST**).
SS	Shear stress limit (Real > 0.0 or blank)
ETYPE	Element type (Character) selected from: **BAR ROD** **QDMEM1 TRMEM QUAD4 TRIA3**
LAYRNUM	The layer number of a composite element (Integer > 0 or blank)
EIDi	Element identification numbers (Integer > 0)

Remarks:

1. Stress constraints are selected in Solution Control with the discipline option:

   STRESS=sid

2. If the alternate form is used, **EID2** must be greater than or equal to **EID1**. Elements in the range which do not exist are ignored.

3. The stress limit for compression, **SC**, is always treated as a negative value regardless of the sign of the input value.

4. **LAYRNUM** is only used if the element is composed of a composite material defined with **PCOMP** Bulk Data entries.

Input Data Entry **DCONVMM** Von-Mises Stress Constraint Definition

Description: Defines a Von-Mises stress constraint by specifying material identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONVMM	SID	ST	SC	SS	MID1	MID2	MID3	MID4	CONT
CONT	MID5	MID6	-etc-						

DCONVMM	100	1.+6	-1.+6	1.+4	101	201	301	401	ABC
+BC	501	601	701						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONVMM	SID	ST	SC	SS	MID1	THRU	MID2		

Field	Contents
SID	Stress constraint set identification (Integer > 0)
ST	Tensile stress limit (Real > 0.0 or blank)
SC	Compressive stress limit (Real, Default = **ST**)
SS	Shear stress limit (Real > 0.0 or blank)
MIDi	Material identification numbers (Integer > 0)

Remarks:

1. Stress constraints are selected in Solution Control with the discipline option:

   STRESS=sid

2. If the alternate form is used, **MID2** must be greater than or equal to **MID1**. Materials in the range which do not exist are ignored.

3. The stress limit for compression, **SC**, is always treated as a negative value regardless of the sign of the input value.

Input Data Entry **DCONVMP** Von-Mises Stress Constraint Definition

Description: Defines a Von-Mises stress constraint by specifying property identification numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DCONVMP	SID	ST	SC	SS	PTYPE	LAYRNUM	PID1	PID2	CONT
CONT	PID3	PID4	-etc-						

DCONVMP	100	1.+6	-1.+6	1.+4	PBAR		102	103	ABC
+BC	107	108	142						

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DCONVMP	SID	ST	SC	SS	PTYPE	LAYRNUM	PID1	THRU	CONT
CONT	PID2								

Field	Contents
SID	Stress constraint set identification (Integer > 0).
ST	Tensile stress limit (Real > 0.0 or blank)
SC	Compressive stress limit (Real, Default = **ST**)
SS	Shear stress limit (Real > 0.0 or blank)
PTYPE	Property type (Character) selected from: **PBAR** **PROD** **PSHEAR** **PQDMEM1** **PTRMEM** **PSHELL** **PCOMP** **PCOMP1** **PCOMP2**
LAYRNUM	The layer number of a composite element (Integer > 0 or blank)
PIDi	Property identification numbers (Integer > 0)

Remarks:

1. Stress constraints are selected in Solution Control with the discipline option:

   STRESS=sid

2. If the alternate form is used, **PID2** must be greater than or equal to **PID1**. Properties in the range which do not exist are ignored.

3. The stress limit for compression, **SC**, is always treated as a negative value regardless of the sign of the input value.

4. **LAYRNUM** is only used if the element is composed of a composite material defined with **PCOMP** Bulk Data entries.

Input Data Entry: **DENSLIST**

Description: Defines a list of density ratio values.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DENSLIST	SID	DENS1	DENS2	DENS3	DENS4	DENS5	DENS6	DENS7	CONT
CONT	DENS8	DENS9	-etc-						

DENSLIST	201	1.0	0.5	0.7					

Field	Contents
SID	Density set identification number (Integer > 0)
DENSi	Density ratio value (Real > 0.0)

Remarks:

1. **DENSLIST** Bulk Data entries are selected in the Function Packet.
2. The density ratios will be used to select particular intrinsic function values for those intrinsics that are associated with a density ratio; e.g. flutter roots.

Input Data Entry: **DESELM**

Description: Designates design variable properties when the design variable is uniquely associated with a single finite element

Format and Example:

1	2	3	4	5	6	7	8	9	10
DESELM	DVID	EID	ETYPE	VMIN	VMAX	VINIT	LAYERNUM	LABEL	CONT
CONT	DVSYMBL								

DESELM	1	10	CBAR	0.01	10.0	1.0			+ABC
+BC	D1								

Field	Contents
DVID	Design variable identification (Integer > 0)
EID	Element identification (Integer > 0)
ETYPE	Element type (Character) selected from: **CELASi** **CMASSi** **CONM2** **CBAR** **CROD** **CONROD** **CSHEAR** **CQDMEM1** **CTRMEM** **CQUAD4** **CTRIA3**
VMIN	Minimum allowable value of the design variable (Real ≥ 0.0) (Default = .001)
VMAX	Maximum allowable value of the design variable (Real ≥ 0.0) (Default = 1000.)
VINIT	Initial value of the design variable (Real, **VMIN** ≤ **VINIT** ≤ **VMAX**) (Default = 1.0)
LAYERNUM	The layer number of a composite element to be designed (Integer > 0, or blank)
LABEL	Optional user-supplied label to define the design variable (Character)
DVSYMBL	Design variable symbol associated with this local design variable (Remark 3)

Remarks:

1. The initial element thickness or area used in the structural analysis is derived from the **VINIT** value and the property value on the associated property entry:

   t_{init} = VINIT * *property_value*

   Similarly, the minimum and maximum values are the **VMIN** and **VMAX** values of the element property are derived from:

   t_{min} = VMIN * *property_value*

   t_{max} = VMAX * *property_value*

2. **DVID** must be unique among all **DESELM**, **DESVARP** and **DESVARS** entries.

3. If the designed element has only one designable property, the continuation containing DVSYMBL may be omitted. Otherwise, a selection must be made from the following table:

ELEMENTS	ALLOWABLE DVSYMBL VALUES
ELASi	K
MASSi, CONM2	M
BAR (PBAR), ROD, CONROD	A
BAR (PBAR1)	D1, D2, D3, D4, D5, D6, D7, D8, D9, D10
SHEAR,QDMEM1,TRMEM,QUAD4,TRIA3	T

Input Data Entry: **DESVARP**

Description: Designates physically linked global design variable properties

Format and Example:

1	2	3	4	5	6	7	8	9	10
DESVARP	DVID	LINKID	VMIN	VMAX	VINIT	LAYERNUM	LAYRLST	LABEL	
DESVARP	1		0.01	2.0	1.0	13		NBDTOP	

Field	Contents
DVID	Design variable identification (Integer > 0)
LINKID	link identification number referring to **ELIST**, **ELISTM** and/or **PLIST**, **PLISTM** entries (Integer > 0, or blank) (Default = **DVID**)
VMIN	Minimum allowable value of the design variable (Real ≥ 0.0) (Default = 0.001)
VMAX	Maximum allowable value of the design variable (Real ≥ 0.0) (Default = 1000.0)
VINIT	Initial value of the design variable (Real, **VMIN** ≤ **VINIT** ≤ **VMAX**) (Default = 1.0)
LAYRNUM	Layer number if referencing a single layer of composite element(s) (Integer > 0 or blank)
LAYRLST	Set identification number of **PLYLIST** entries specifying a set of composite layers to be linked (Integer > 0 or blank)
LABEL	Optional user supplied label to define the design variable (Character)

Remarks:

1. The elements linked to the **DESVARP** are specified using one or more **ELIST**, **ELISTM**, **PLIST**, and **PLISTM** entries.
2. The initial element thickness or area used in the structural analysis is derived from the **VINIT** value and the property value on the associated property entry:

   t_{init} = VINIT * *property_value*

   Similarly, the minimum and maximum values are the **VMIN** and **VMAX** values of the element property are derived from:

   t_{min} = VMIN * *property_value*

   t_{max} = VMAX * *property_value*
3. **LAYRNUM** and **LAYRLST** are mutually exclusive.
4. Noncomposite elements may be linked to composite layers by including them in the **ELIST**, **ELISTM** and/or **PLIST**, **PLISTM** sets.

Input Data Entry: **DESVARS**

Description: Designates shape function linked global design variable properties.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DESVARS	DVID	SHAPEID	VMIN	VMAX	VINIT	LAYERNUM	LAYRLST	LABEL	
DESVARS	1		0.01	2.0	1.0	13		INBDTOP	

Field	Contents
DVID	Design variable identification (Integer > 0)
SHAPEID	Identification number of **SHAPE**, **SHAPEM**, or **SHPGEN** Bulk Data entries defining the shape function (Integer > 0, or blank) (Default = **DVID**)
VMIN	Minimum allowable value of the design variable (Real) (Default = -10^{20})
VMAX	Maximum allowable value of the design variable (Real) (Default = 10^{20})
VINIT	Initial value of the design variable (Real, **VMIN** ≤ **VINIT** ≤ **VMAX**) (No default, a value must be supplied)
LAYRNUM	Layer number if referencing a single layer of composite element(s) (Integer > 0 or blank)
LAYRLST	Set identification of **PLYLIST** entries specifying a set of composite layers to be linked (Integer > 0 or blank)
LABEL	Optional user supplied label to define the design variable (Character)

Remarks:

1. The elements linked to the **DESVARS** are specified using **SHAPE** and/or **SHAPEM** Bulk Data entries.
2. The initial local variables are computed from:

   $\{t_{init}\} = \boldsymbol{P}\ \{\mathbf{VINIT}\}$

   Where $\boldsymbol{P}$ is the design variable linking matrix and the minimum and maximum values for the local variables are taken from the **TMIN** and **TMAX** values on the property and connectivity entries, respectively.
3. **LAYRNUM** and **LAYRLST** are mutually exclusive.
4. Noncomposite elements may be linked to composite layers by including them in the referenced **SHAPE** or **SHAPEM** set.

Input Data Entry: **DLAGS**

Description: This entry is used in conjunction with **RLOAD1**, **RLOAD2**, **TLOAD1** and **TLOAD2** data entries and defines time lags and phase lags as well as the set identification of the static load.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DLAGS	SID	LID	TAU	PHASE	LID	TAU	PHASE		
DLAGS	5	21	0.04	20.0	10	0.0	45.0		

Field	Contents
SID	Identification number of **DLAGS** set (Integer > 0)
LID	Identification number of time (or frequency) independent applied load (Integer > 0)
TAU	Time delay for the designated load set (Real)
PHASE	Phase lag (in degrees) for the designated load set (Real)

Remarks:

1. One or two dynamic load sets may be defined on a single entry.
2. Refer to **RLOAD1**, **RLOAD2**, **TLOAD1** or **TLOAD2** entries for formulas which define the manner in which **TAU** and **PHASE** are used.
3. The phase parameter is used only in conjunction with **RLOAD1** and **RLOAD2** data entries.
4. The **LID** set can refer to statically applied loads as well as to additional dynamic loads input on **DLONLY** entries.
5. **TAU** and **PHASE** can be defaulted to zero, but **LID** must not be zero.

Input Data Entry: **DLOAD**

Description: Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined using **RLOAD1** or **RLOAD2** entries (for frequency response) or **TLOAD1** or **TLOAD2** entries (for transient response)

Format and Example:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	CONT
CONT	S4	L4		–etc–					

DLOAD	17	1.0	2.0	6	-2.0	7	2.0	8	+A
+A	-2.0	9							

Field	Contents
SID	Load set identification number (Integer > 0)
S	Scale factor (Real ≠ 0.0)
Si	Scale Factors (Real ≠ 0.0)
Li	Load set identification numbers defined via bulk data entries enumerated above (Integer > 0)

Remarks:

1. The load vector being defined by this entry is given by

$$[P] = S \sum_{j} S_i P_i$$

2. The **Li** must be unique.
3. **SID** must be unique from all **Li**.
4. **TLOAD1** and **TLOAD2** loads may be combined only through the use of the **DLOAD** entry.
5. **RLOAD**1 and **RLOAD**2 loads may be combined only through the use of the **DLOAD** entry.
6. **SID** must be unique for all **TLOAD1**, **TLOAD2**, **RLOAD**1 and **RLOAD**2 entries.
7. Linear load sets must be selected by a solution control command (**DLOAD = SID**).

Input Data Entry: **DLONLY**

Description: This entry is used in conjunction with the **RLOAD1**, **RLOAD2**, **TLOAD1** and **TLOAD2** entries and defines the point where the dynamic load is to be applied with the scale factor A.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DLONLY	SID	P	C	A	P	C	A		
DLONLY	3	6	2	8.2	15	1	10.1		

Field	Contents
SID	Identification number of **DLONLY** set (Integer > 0)
P	Grid, extra point or scalar point identification number (Integer > 0)
C	Component number (1 through 6 for grid point; blank or 0 for extra points or scalar points)
A	Load factor A for the designated coordinate (Real)

Remarks:

1. One or two load factors may be defined on a single entry.
2. Refer to **RLOAD1**, **RLOAD2**, **TLOAD1** or **TLOAD2** entries for the formulas which define the load factor **A**.
3. Component numbers refer to global coordinates.
4. The **SID** field is referred to as the **DLAGS** entry.
5. The scale factor, **A**, applied to any grid/component will be the sum of all **Ai** for that degree of freedom on all **DLONLY** entries with the same **SID**.

Input Data Entry: **DMI** Direct Matrix Input

Description: Used to input matrix data base entities directly. Generates a real or complex matrix of the form:

$$\boldsymbol{A} = \begin{bmatrix} A_{11} & A_{12} & \dots & A_{1n} \\ A_{21} & A_{22} & \dots & A_{2n} \\ \dots & \dots & \dots & \dots \\ A_{n1} & A_{m2} & \dots & A_{mn} \end{bmatrix}$$

where the elements A_{ij} may be real or complex

Format and Example:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	PREC	FORM	M	N				CONT
CONT	C1	R1	A(R1, C1)	C2	R2	A(R1, C2)	A(R1+1, C2)	C3	CONT
CONT	R1	A(R1, C3)	C4	R2	A(R2, C4)				

DMI	TEST	RDP	REC	3	4				ABC
+BC	1	2	2.0	2	1	3.0	4.0	4	DEF
+EF	1	5.0	4	3	6.5				

Field	Contents
NAME	Any valid data base entity name (Character)
PREC	The precision of the matrix entity to be loaded (Character) selected from: **RSP** **CSP** **RDP** **CDP**
FORM	The form of the matrix entity to be loaded. Any one of the following **REC** **SYM** **DIAG** **IDENT** **SQUARE**
M	The number of rows in the matrix (Integer > 0)
N	The number of columns in the matrix (Integer > 0)
C_i	The column number of the column being loaded (Integer)
R_i	The row number of the first row in the string being loaded (Integer)
$A(R_i, C_i)$	Matrix terms (Real)

Remarks:

1. If the named entity exists, it will be flushed and reloaded. If the entity does not exist, it will be created.
2. Column and row identifiers (**Ci, Ri**) must always appear together although they can appear in any two contiguous fields.
3. Columns must be loaded in increasing column number order. If more than one string is to be loaded for a particular column, the **Ci** field must contain the same value as in the previous string. Strings must be loaded in increasing row order without overlap. Complex matrices require two real values for each matrix term. These can be split across physical entry boundaries.

Input Data Entry: **DMIG** Direct Matrix Input at Grid Points

Description: Defines structure-related direct input matrices with terms located by external grid/component values.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	PREC	FORM						CONT
CONT	GCOL	CCOL	GROW	CROW	X_{ij}	Y_{ij}			CONT
CONT	GCOL	CCOL	GROW	CROW	X_{ij}	Y_{ij}			CONT

DMIG	TEST	RDP	REC						ABC
+BC	1001	4	2001	2	1.25+5				DEF
+EF	1001	4	3001	3	2.67+4	-etc-			

Field	Contents
NAME	Any valid data base entity name (Character)
PREC	The precision of the matrix entity to be loaded. Any one of the following character strings: **RSP**, **RDP**, **CSP**, or **CDP**
FORM	The form of the matrix entity to be loaded. Any one of the following: **REC, SYM, DIAG, IDENT, SQUARE, TRIANG**
GCOL	Grid, scalar or extra point identification for column index (Integer)
CCOL	Component number for **GCOL**, 0 ≤ **CCOL** ≤ 6 if **GCOL** is a grid point, zero or blank for scalar or extra points. (Integer)
GROW	Grid, scalar or extra point identification for row index. (Integer)
CROW	Component number for **GROW**, 0 ≤ **CROW** ≤ 6 if **GROW** is a grid point, zero or blank for scalar or extra points. (Integer)
X_{ij}, Y_{ij}	Matrix term. X_{ij} is real part for real or complex matrices. Yij is the imaginary part for complex matrices and is ignored for real matrices.(Real)

Remarks:

1. If the named entity exists, it will be flushed and reloaded. If the entity does not exist, it will be created.
2. The number of rows and columns will be either p-set size or g-set size depending on whether the named entity is requested by **K2PP**, **M2PP**, **B2PP** or **K2GG**, **M2GG**, etc.
3. Each non-null term in the matrix requires a continuation entry. The column index and row index values can appear any number of times on a logical entry but a fatal error will occur if the same term is entered more than once.
4. The matrix terms can be entered in any order.
5. The **TRIANG** input **FORM** implies that only the upper ***or*** lower triangular portion of the symmetric matrix is input. ASTROS will automatically expand the input across the diagonal.

Input Data Entry: **DVTOPTE** Type definition for designed element thickness variation

Description: Defines the thickness variation type for a designed element by specifying the element identification numbers.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
DVTOPTE	TYPE	ETYPE	EID1	EID2	EID3	EID4	EID5	EID6	CONT
CONT	EID7	EID8	-etc-						

DVTOPTE	TOP	QUAD4	101	102	104				

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DVTOPTE	TYPE	ETYPE	EID1	THRU	EID2				

Field	Contents
TYPE	Designed element thickness variation type, one of the character values, **CENTER**, **TOP** or **BOTTOM**. (Character, default = **CENTER**) **CENTER** Element thickness varies about a fixed element reference plane **TOP** Element thickness varies about a fixed element top plane **BOTTOM** Element thickness varies about a fixed element bottom plane
ETYPE	Element type (Character) selected from: **QUAD4** **TRIA3**
EIDi	Element identification number (Integer > 0)

Remarks:

1. The thickness option for a selected element will be ignored if it is not a designed plate bending element.

Input Data Entry: **DVTOPTL** Type definition for designed element thickness variation.

Description: Defines the thickness variation type for a designed element by specifying the element list set ID number.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
DVTOPTL	TYPE	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	CONT
CONT	ELID8	ELID9	-etc-						

DVTOPTL	TOP	10	99	999					

Field	Contents
TYPE	Designed element thickness variation type, one of the character values, **CENTER**, **TOP** or **BOTTOM**. (Character, default = **CENTER**) **CENTER** Element thickness varies about a fixed element reference plane **TOP** Element thickness varies about a fixed element top plane **BOTTOM** Element thickness varies about a fixed element bottom plane
ELIDi	Element list set identification number (Integer > 0)

Remarks:

1. The thickness option for a selected element will be ignored if it is not a design bending element.
2. The elements in the specified list will be ignored if they are not **QUAD4** or **TRIA3**.

Input Data Entry: **DVTOPTP** Type definition for designed element thickness variation.

Description: Defines the thickness variation type for a designed element by specifying the element property identification numbers.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
DVTOPTP	TYPE	PTYPE	PID1	PID2	PID3	PID4	PID5	PID6	CONT
CONT	PID7	PID8	-etc-						

DVTOPTP	TOP	PSHELL	100	200					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
DVTOPTP	TYPE	PTYPE	PID1	THRU	PID2				

Field	Contents
TYPE	Designed element thickness variation type, one of the character values, **CENTER**, **TOP** or **BOTTOM**. (Character, default = **CENTER**) **CENTER** Element thickness varies about a fixed element reference plane **TOP** Element thickness varies about a fixed element top plane **BOTTOM** Element thickness varies about a fixed element bottom plane
PTYPE	Property type (Character) selected from: **PSHELL** **PCOMP** **PCOMP1** **PCOMP2**
PIDi	Property identification number (Integer > 0)

Remarks:

1. The thickness option for elements connected to the specified properties will be ignored if it they are not designed bending plate elements.

Input Data Entry: **DYNRED** Dynamic Reduction Data

Description: Defines dynamic reduction control data.

Format and Example:

1	2	3	4	5	6	7	8	9	10
DYNRED	SID	FMAX	NVEC						
DYNRED	1	50.0							

Field	Contents
SID	Set identification number (Integer > 0)
FMAX	Highest frequency of interest (Hertz) (Real > 0 or blank)
NVEC	Number of generalized coordinates desired (Integer > 0 or blank)

Remarks:

1. Dynamic reduction data must be requested in the Solution Control packet with:

   DYNRED=SID

2. The user should select either an **FMAX**, or both the **FMAX** and **NVEC** fields. **FMAX** should not be greater than necessary for the specific dynamic analysis. **NVEC**, if specified, should be significantly less than the size of the f-set to realize any computational cost savings. **NVEC** will limit dynamic reduction to using **NVEC** flexible vectors.

3. Dynamic reduction transforms the motions of the f-set to the motions of the user defined A-set plus motions of generalized coordinates created in the process. The generalized coordinates represent overall structure displacements which are approximate normal mode shapes. The generalized coordinates are identified by SCALAR points that are automatically generated. The SCALAR point identification numbers begin with 1 greater than the highest user GRID, SCALAR, or EXTRA point identification number.

Input Data Entry: **EIGC** Complex Eigenvalue Extraction Data.

Description: Specifies complex eigensolution control data.

Format and Example:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E			CONT
CONT	PA1	QA1	PB1	QB1	W1	NE1	ND1		CONT
CONT	PA2	QA2	PB2	QB2	W2	NE2	ND2		

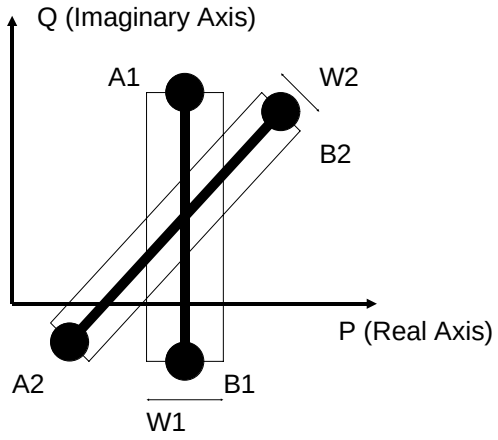
EIGC	14	INV	POINT	27		1.-8			ABC
+BC	2.0	5.6	2.0	-3.4	2.0	4	4		DEF
+EF	-5.5	-5.5	5.6	5.6	1.5	6	3		

Field	Contents
SID	Set identification number (Unique integer > 0)
METHOD	Method of complex eigenvalue extraction, one of the strings **INV** or **HESS** **INV** - Inverse power method **HESS** - Upper Hessenberg method
NORM	Method for normalizing eigenvectors, one of the strings **MAX** or **POINT** **MAX** - Normalize to a unit value for the real part and a zero value for the imaginary part, the component having the largest magnitude. **POINT** - Normalize to unit value of the component **G**,**C** (defaults to **MAX** if point is not defined)
G	Grid or scalar point identification number (Required if and only if **NORM = POINT** (Integer > 0)
C	Component number (Required if and only if **NORM = POINT** and **G** is a geometric grid point) (0 < Integer ≤ 6)
E	Convergence test (Real, Default = 10^{-6})
PAi, QAi PBi, QBi	Two complex points defining a line in the complex plane (Real)
W	Width of region in complex plane (Real > 0)
NEi	Estimated number of roots in each region (Integer > 0)
NDi	Desired number of roots in each region (Default is **3*NEi**) (Integer > 0)

Remarks:

1. The **SID** may be called out directly in the **CEIG** module call or may be entered via the Solution Control **CMETHOD** in the **BOUNDARY** command. One of these methods must be used.

2. Each continuation entry defines a rectangular search region. Any number of regions may be used and they may overlap. Roots in overlapping regions will not be extracted more than once.



3. The units of **P**, **Q**, and **W** are radians per unit time.

4. At least one continuation entry is required.

5. For the Upper Hessenberg method, **ND1** controls the number of vectors computed. Only one continuation entry is considered and the (**P**, **Q**) pairs, along with the parameters **W1** and **NE1** are ignored. All eigenvalues are computed for this method.

6. If (**P**, **Q**) pairs and parameters **W1** and **NE1** are provided, and insufficient memory exists for the Upper Hessenberg method , ASTROS will switch to the Inverse power method.

7. A pair (**P**, **Q**) defines a complex eigenvalue. From this pair the following may be computed:

   f_N = undamped frequency = $\frac{1}{2\pi}\sqrt{P^2+Q^2}$

   ξ = damping coefficient = $\frac{P}{\sqrt{P^2+Q^2}}$

   f_D = damped frequency = $f_N\sqrt{1-\xi^2}$

   for lightly damped systems, **Q** is a measure of the radian frequency and **P** is a measure of the damping.

8. **P**arameter **Wi** should be kept greater than 5 percent of the segment length **Ai** to **Bi** for relatively efficient processing.

Input Data Entry EIGR(GIVENS and Modified GIVENS)

Description: Specifies real eigensolution control data for the Givens methods which are used to extract all eigenvalues.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	FL	FU		NVEC		E	-cont-
-cont-	NORM	GID	DOF						

Requesting Eigenvectors in a Frequency Range:

EIGR	13	GIV	.0	20.0					+A
+A	POINT	32	4						

Requesting a Specified Number of Eigenvectors:

EIGR	13	MGIV				10			+A
+A	POINT	32	4						

Field	Contents
SID	Set identification number. (Required,Integer>0) [1]
METHOD	Method of eigenvalue extraction. [2,3] **GIV** Given's method **MGIV** Modified Given's method
FL,FU	Frequency range for eigenvector computations. (cycles/sec) (Real>0.0, **FL<FU**) [4]
NVEC	Number of eigenvectors to compute. (Integer>0, default 1)
E	Mass orthogonality test parameter. A non-zero value requests a check of the mass orthogonality of the eigenvectors. (Real>0.0, default 10^{-10})
NORM	Method for eigenvectors normalization. [5,6] Method for normalizing eigenvectors, one of the character values, **MASS**, **MAX**, or **POINT** **MASS** - Normalize to unit value of the generalized mass **MAX** - Normalize to unit value of the largest component in the analysis set (Default) **POINT** - Normalize to unit value of the component defined by **G**,**C** (defaults to "**MAX**" if point is not defined)
GID	Grid or scalar point identification number. (Required only if **NORM=POINT**) (Integer>0)
DOF	Component number (One of the integers 1-6) (Required only if **NORM =** "**POINT**" and **GID** is a geometric grid point)

Remarks:

1. The real eigenvalue extraction method set must be selected in Solution Control (METHOD=SID) to be used.
2. Both the **GIV** and **MGIV** methods are full-spectrum tridiagonalization procedures which compute all eigenvalues and a range of eigenvectors selected by the user. The **GIV** method requires that the a-set

mass matrix be positive definite. The **MGIV** method uses an additional transformation to remove this requirement.

3. If **METHOD** is **GIV**, the mass matrix for the analysis set must be positive definite. This means that all degrees of freedom, including rotations, must have mass properties.

4. The number of eigenvalues which are computed depend on the values of **FL**, **FU**, and **NVEC**. The following table summarizes the options.

FL	FU	NVEC	Mode Shapes Computed
Blank	Blank	Blank	The lowest mode only.
Blank	Blank	***n_val***	The first *n_val* modes.
Blank	***hi_val***	Blank	All modes between $-\infty$ and ***hi_val***.
Blank	***hi_val***	***n_val***	First ***n_val*** modes in the range $-\infty$ and ***hi_val***.
low_val	Blank	Blank	First mode above ***low_val***.
low_val	Blank	***n_val***	First ***n_val*** modes above ***low_val***.
low_val	***hi_val***	Blank	All modes between ***low_val*** and ***hi_val***.
low_val	***hi_val***	***n_val***	First ***n_val*** modes between ***low_val*** and ***hi_val***.
If you are extracting rigid body modes you should leave the FL Field blank.			

5. If you select **NORM=MASS**, the eigenvectors are normalized to a unit value of the generalized mass. If you select **NORM=MAX**, the eigenvectors are normalized with respect to the largest component value in the *g-set*. When using the **MAX** normalization with Dynamic Reduction, the *g-set* degrees of freedom, excluding the dynamic reduction generalized coordinates, are used in the normalization process. Finally, if you select **NORM=POINT**, the eigenvectors are normalized with respect to the value of the component defined by **GID** and **DOF**. This component must be in the analysis set.

Bulk Data Entry EIGR (INVERSE POWER)

Description: Specifies real eigensolution control data for the Inverse Power method which is used to extract a few eigenvalues in a specified frequency range.

Format and Example:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	FL	FU	NEST	NVEC		E	-cont-
-cont-	NORM	GID	DOF						

EIGR	13	INV	1.9	15.6	10	12		1.-6	+A
+A	POINT	32	4						

Field	Contents
SID	Set identification number. [1] Integer>0 Required
METHOD	Method of eigenvalue extraction. Character **SINV** Required
FL,FU	Frequency range of interest (cycles/sec). [2] Real **FL<FU** Required
NEST	Estimated number of roots in the frequency range **FL** to **FU**. (Integer>0, Required)
NVEC	The number of eigenvectors to be computed. [2] (Integer>0, default= 3*NEST)
E	The mass orthogonality test and eigenvalue convergence parameter. A non-zero value requests a check of the mass orthogonality of the eigenvectors.(Real>0.0, default 10^{-10})
NORM	Method for eigenvectors normalization. [5,6] Method for normalizing eigenvectors, one of the character values, **MASS**, **MAX**, or **POINT** **MASS** - Normalize to unit value of the generalized mass **MAX** - Normalize to unit value of the largest component in the analysis set (Default) **POINT** - Normalize to unit value of the component defined by **G**,**C** (defaults to "**MAX**" if point is not defined)
GID	Grid or scalar point identification number. (Required only if **NORM=POINT**) (Integer>0)
DOF	Component number (One of the integers 1-6) (Required only if **NORM =** "**POINT**" and **G** is a geometric grid point)

Remarks:

1. The real eigenvalue extraction method set must be selected in Solution Control (METHOD=SID) to be used.
2. The number of eigenvalues and eigenvectors extracted depends on the **FL,FU** and **NVEC** values. A summary is given in the table found with entry **EIGR** (**Lanczos**).
3. If you select **NORM=MASS**, the eigenvectors are normalized to a unit value of the generalized mass. If you select **NORM=MAX**, the eigenvectors are normalized with respect to the largest component value in the *g-set*. When using the **MAX** normalization with Dynamic Reduction, the *g-set* degrees of freedom, excluding the dynamic reduction generalized coordinates, are used in the normalization process. Finally, if you select **NORM=POINT**, the eigenvectors are normalized with respect to the value of the component defined by **GID** and **DOF**. This component must be in the analysis set.

Bulk Data Entry EIGR(LANCZOS)

Description: Specifies real eigensolution control data for the Lanczos method of eigenvalue extraction.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	FL	FU		NVEC		E	CONT
CONT	NORM	GID	DOF						

EIGR	1	LANCZOS	.0	20.0					

Field	Contents
SID	Set identification number. [1] Integer>0 Required
METHOD	Method of eigenvalue extraction. [2] Character **LANCZOS** Required
FL,FU	Frequency range for eigenvector computations. (cycles/sec) Real **FL<FU** [3]
NVEC	Number of eigenvectors to compute. Integer [3]
E	Mass orthogonality test parameter. A non-zero value requests a check of the mass orthogonality of the eigenvectors. Real>0.0 10^{-10}
NORM	Method for eigenvectors normalization. [5,6] Method for normalizing eigenvectors, one of the character values, **MASS**, **MAX**, or **POINT**: **MASS** Normalize to unit value of the generalized mass **MAX** Normalize to unit value of the largest component in the analysis set (Default) **POINT** Normalize to unit value of the component defined by **G**,**C** (defaults to **MAX** if point is not defined)
GID	Grid or scalar point identification number. (Required only if **NORM=POINT**) (Integer>0)
DOF	Component number (One of the integers 1-6) (Required only if **NORM =** "**POINT**" and **G** is a geometric grid point)

Remarks:

1. The real eigenvalue extraction method set must be selected in Solution Control (METHOD=SID) to be used.
2. The Lanczos eigenvalue extraction technique is optimized for processing large, sparse matrices. It is not recommended to perform either Guyan reduction or Dynamic Reduction with the Lanczos technique.

3. The number of eigenvalues and eigenvectors extracted depends on the **FL,FU** and **NVEC** values. A summary is given in the following table:

FL	FU	NVEC	Eigenvalues and Mode Shapes Computed
Blank	Blank	Blank	The lowest mode only.
Blank	Blank	***n_val***	The first *n_val* modes.
Blank	***hi_val***	Blank	All modes between $-\infty$ and ***hi_val***.
Blank	***hi_val***	***n_val***	First ***n_val*** modes in the range $-\infty$ and ***hi_val***.
low_val	Blank	Blank	First mode above ***low_val***.
low_val	Blank	***n_val***	First ***n_val*** modes above ***low_val***.
low_val	***hi_val***	Blank	All modes between ***low_val*** and ***hi_val***.
low_val	***hi_val***	***n_val***	First ***n_val*** modes between ***low_val*** and ***hi_val***.
If you are extracting rigid body modes you should leave the FL Field blank.			

4. If you select **NORM=MASS**, the eigenvectors are normalized to a unit value of the generalized mass. If you select **NORM=MAX**, the eigenvectors are normalized with respect to the largest component value in the *g-set*. When using the **MAX** normalization with Dynamic Reduction, the *g-set* degrees of freedom, excluding the dynamic reduction generalized coordinates, are used in the normalization process. Finally, if you select **NORM=POINT**, the eigenvectors are normalized with respect to the value of the component defined by **GID** and **DOF**. This component must be in the analysis set.

Input Data Entry: **ELEMLIST**

Description: Defines a list of elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
ELEMLIST	SID	ETYPE	EID1	EID2	EID3	EID4	EID5	EID6	CONT
CONT	EID7	EID8	-etc-						CONT

ELEMLIST	100	QDMEM1	100	101	200	205			

Alternate Form:

1	2	3	4	5	6	7	8	9	10
ELEMLIST	SID	ETYPE	EID1	THRU	EID2				

Field	Contents
SID	Set identification number referenced by Solution Controlcommands or Bulk Data entries. (Integer > 0)
ETYPE	Character input identifying the element type. One of the following: **BAR** **QDMEM1** **CONM2** **QUAD4** **ELAS** **ROD** **IHEX1** **SHEAR** **IHEX2** **TRIA3** **IHEX3** **TRMEM** **MASS** **CONROD**
EIDi	Element identification numbers (Integer > 0 or blank)

Remarks:

1. If the alternate form is used, **EID2** must be greater than or equal to **EID1**.
2. Nonexistent elements may be referenced, and if so, no error messages are issued.
3. Any number of continuations is allowed.

Input Data Entry: **ELIST**

Description: Defines elements associated with a design variable.

Format and Example:

1	2	3	4	5	6	7	8	9	10
ELIST	LINKID	ETYPE	EID1	EID2	EID3	EID4	EID5	EID6	CONT
CONT	EID7	EID8	EID9	-etc-					CONT

ELIST	6	CROD	12	14	22				

Alternate form:

1	2	3	4	5	6	7	8	9	10
ELIST	LINKID	ETYPE	EID1	THRU	EID2				

Field	Contents
LINKID	Element list identifier (Integer > 0)
ETYPE	Character input identifying the element type. One of the following: **CELAS1 CELAS2 CMASS1 CMASS2 CONM2** **CBAR CROD CONROD** **CSHEAR CQDMEM1 CTRMEM CQUAD4 CTRIA3**
EIDi	Element identification numbers (Integer > 0, or blank)

Remarks:

1. If the alternate form is used, **EID2** must be greater than or equal to **EID1**.
2. All elements listed as **ELIST** entries for a particular **LINKID**, will be designed by (linked to) that design variable referencing the **ELIST LINKID**.
3. If **PBAR1** cross-sectional parameters are to be linked, the **ELISTM** Bulk Data entry must be used.

Input Data Entry: **ELISTM**

Description: Defines elements, and their local design variables, associated with a design variable.

Format and Example:

1	2	3	4	5	6	7	8	9	10
ELISTM	LINKID	ETYPE	EID1	DVSYM1	EID2	DVSYM2	EID3	DVSYM3	CONT
CONT	EID4	DVSYM4	-etc-						CONT

ELISTM	6	BAR	12	A	22	A			

Field	Contents
LINKID	Element list identifier (Integer > 0)
ETYPE	Character input identifying the element type. One of the following: **CELAS1 CELAS2 CMASS1 CMASS2 CONM2** **CBAR CROD CONROD** **CSHEAR CQDMEM1 CTRMEM CQUAD4 CTRIA3**
EIDi	Element identification numbers (Integer > 0, or blank)
DVSYMi	Symbol defining the local design variable. (Remarks 2 and 3)

Remarks:

1. The **LINKID** is referenced by **DESVARP** data to connect the global design variable to the local variables.
2. The following symbols may be used for the different types of elements:

ELEMENTS	ALLOWABLE DVSYM VALUES
ELASi	**K**
MASSi, CONM2	**M**
BAR (PBAR), ROD, CONROD	**A**
BAR (PBAR1)	**D1, D2, D3, D4, D5, D6, D7, D8, D9, D10**
SHEAR,QDMEM1,TRMEM,QUAD4,TRIA3	**T**

3. If all elements to be linked have only one possible **DVSYM** (e.g. **K**), the **ELIST** Bulk Data entry may be used.

Input Data Entry: **EPOINT** Extra Point List

Description: Defines extra points of the structural model for use in dynamics problems.

Format and Example:

1	2	3	4	5	6	7	8	9	10
EPOINT	SETID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	CONT
CONT	ID8	ID9	-etc-						
EPOINT	1000	3	18	1	4	16	2		

Alternate Form:

1	2	3	4	5	6	7	8	9	10
EPOINT	SETIC	ID1	THRU	ID2					

Field	Contents
SETID	Extra point sets identification numbers. (Integer > 0)
IDi	Extra point identification number (Integer > 0)

Remarks:

1. The extra point set identification is selected on the **BOUNDARY** entry. All extra points defined with this **SETID** will be used in dynamic analyses in the boundary condition.
2. All extra point identification numbers must be unique with respect to all other structural and scalar points.
3. This entry is used to define coordinates used in transfer function definitions (see **TF** entry) and Direct Matrix input.
4. If the alternate form is used, **ID2** must be greater than or equal to **ID1**.

Input Data Entry: **FFT**

Description: Defines parameters for controlling the Fast Fourier Transformation (**FFT**) during time domain response analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FFT	SID	TIME	NT	RDELTF	RF	FRIM	OTYPE	FLIM	
FFT	3	20.	1024	1.0					

Field	Contents
SID	**FFT** set identification number (Integer > 0)
TIME	Length of time period to be analyzed (Real > 0.0)
NT	Number of time points to be used for the **FFT** (Integer ≥ 2)
RDELTF	Ratio of incremental frequency (del F) to 1 / T. See remarks 4 and 6. (Default = 1.0, Real > 0.0)
RF	Ratio of total frequency duration (F) to **NT** / 2*T. See remarks 5 and 6. (Default = 1.0, Real > 0.0)
FRIM	Frequency response interpolation method. Character string **LINEAR** or **CUBIC**. Default is **LINEAR**.
OTYPE	Type of response to be output. Character string **TIME**, **FREQ** or **BOTH**. Default is **TIME**.
FLIM	Frequency load interpolation method. Character string **LINEAR** or **CUBIC**. Default is **LINEAR**.

Remarks:

1. **SID** must be selected by a **FFT** option on a **TRANSIENT** command in solution control.
2. **TIME** is the period for periodic dynamic loads defined in the time domain. For non-periodic loads, T is the total time duration of the excitation plus any quiet portion desired for response decay. T may be larger than the time duration defined by **TLOAD1** or **TLOAD2** data, in which case the forcing function will be automatically set to zero for the additional time.
3. **NT** should be a power of 2; i.e., **NT** = 2**m, m = 1,2,...; or **NT** = 2, 4, 8, ... If **NT** is not a power of 2, it will be automatically set to the next highest power of 2 value.
4. The incremental frequency, Δ F, required by the **FFT** algorithm, is 1 / T. The value of Δ F may be adjusted by the user with the **RDELTF** factor. However, the most accurate results are normally obtained with the default case of **RDELTF** = 1.0.
5. The frequency duration required by the **FFT** algorithm is F = **NT** / 2*T. This is the frequency duration used when the default value of **RF** = 1.0 is used. If **RF** < 1.0, the response between **RF** and 1.0 is set to zero when using the inverse Fourier transform to compute time domain responses.
6. The frequency list used in the frequency response calculations is generated using a constant incremental frequency of del F = **RDELTF** * Δ F , and the total frequency duration is F = **RF** * F.

Input Data Entry: **FLFACT** Aerodynamic Physical Data

Description: Used to specify density ratios, velocity lists, and reduced frequencies for **FLUTTER** analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FLFACT	SID	F1	F2	F3	F4	F5	F6	F7	CONT
CONT	F8	F9	-etc-						

FLFACT	97	.3	.7	3.5					

Field	Contents
SID	Set identification number (Integer > 0).
Fi	Aerodynamic factor (Real).

Remarks:

1. Only the factors selected by a **FLUTTER** data entry will be used.
2. Embedded blank fields are forbidden.
3. Parameters must be listed in the order in which they are to be used within the looping of **FLUTTER** analysis.
4. All **FLFACT** entries having the same **SETID** will be treated as a single set.

Input Data Entry: **FLUTTER** Aerodynamic **FLUTTER** Data

Description: Defines data needed to perform **FLUTTER** analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FLUTTER	SID	METHOD	DENS	MACH	VEL	MLIST	KLIST	EFFID	CONT
CONT	SYMXZ	SYMXY	EPS	CURFIT	NROOT	VTYPE	GFLUT	GFILTER	

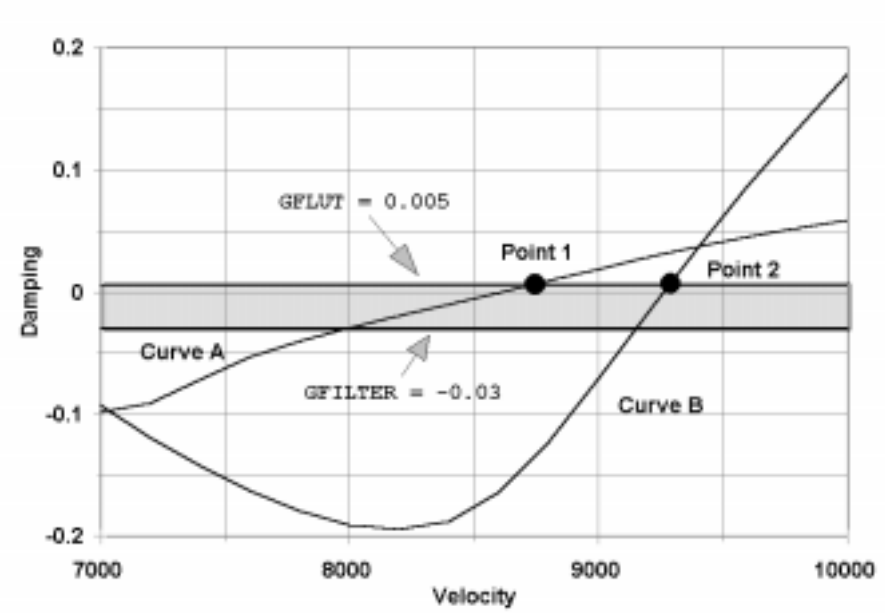
FLUTTER	19	PKIT	119	0.85	319				ABC
+BC	-1		0.01	CUBIC		EQUIV			

Field	Contents
SID	Set identification number (See Remark 1) (Integer > 0)
METHOD	**FLUTTER** analysis method, **PK** or **PKIT** (See Remark 2) (Character, Default=**PK**)
DENS	Identification number of an **FLFACT** set specifying density ratios to be used in **FLUTTER** analysis (See Remark 3) (Integer > 0).
MACH	Mach number to be used in the **FLUTTER** analysis (Real ≥ 0.0)
VEL	Identification number of an **FLFACT** set specifying velocities to be used in the **FLUTTER** analysis. (Integer > 0).
MLIST	Identification number of a **SET1** set specifying a list of normal modes to be ***omitted*** from the **FLUTTER** analysis (See Remark 4) (Integer > 0, or blank).
KLIST	Identification number of an **FLFACT** set specifying a list of ***hard point*** reduced frequencies for the given Mach number for use in the **FLUTTER** analysis (See Remark 5) (Integer ≥ 0, or blank)
EFFID	Identification number of a **CONEFFF** set specifying control surface effectiveness values (See Remark 6) (Integer ≥ 0, or blank)
SYMXZ, SYMXY	Symmetry flags associated with the aerodynamics (See Remark 7) (Integer) +1 Symmetric 0 or blank Asymmetric -1 Antisymmetric
EPS	Convergence parameter for **FLUTTER** eigenvalue (Real, Default = 10^{-5})
CURFIT	Type of curve fit to be used in the **PK FLUTTER** analysis. One of **LINEAR, QUAD, CUBIC,** or **ORIG** (See Remarks 8, 9, and 10) (Character, Default = **LINEAR**)
NROOT	Requests that only the first NROOT eigenvalues be found (Integer or blank)
VTYPE	Input velocities are in units of true, TRUE, or equivalent, EQUIV, speed. (See Remark 11) (Character, Default=TRUE)
GFLUT	The damping a mode must exceed to be considered a flutter crossing (See Remark 12) (Real ≥ 0, Default=0.0)
GFILTER	The damping a mode must attain to be considered stable before a flutter crossing (See Remark 12) (Real, Default=0.0)

Remarks:

1. The **FLUTTER** data entry must be selected in the Solution Control packet. Only those Mach numbers and symmetries selected in Solution will be processed in the **UNSTEADY** aerodynamic preface.
2. When **PK** is selected Muller's method is used, and when **PKIT** is selected the iterative method is used.
3. The density is given by $\rho \times \rho_{ref}$, where ρ_{ref} is the reference value given on the **AERO** Bulk Data entry, and ρ is the density ratio from the **FLFACT** entry.
4. If the **MLIST** is blank or zero, all computed eigenvectors will be retained in the **FLUTTER** analysis.
5. If the **KLIST** is blank or zero, all "hard point" k values (those on the **MKAEROi** entries) associated with the Mach number/symmetries on the **FLUTTER** entry will be used in the interpolation of the aerodynamics. Specifying a subset may be used to improve the **ORIG** interpolation. Those **MKAEROi** hard point k values nearest in value to those listed on the **FLFACT** will be used. No duplicate hard point k's will be used and no errors will be printed.
6. If the **EFFID** is blank or zero, no effectiveness corrections will be made.
7. The symmetry flags are used to select the appropriate unsteady aerodynamic matrices generated from the list on the **MKAEROi** entries.
8. The **LINEAR, QUAD,** and **CUBIC** fits are separate first, second and third order, respectively, fits of the real and complex terms of the generalized aerodynamic matrix between each hard point k. Only the closest 2, 3 or 4, respectively, k's are utilized for each fit and **LINEAR** fitting is used off the ends of the hard point **KLIST**. The program automatically reduces the order of the fit if too few points are available for the higher order fit (e.g., **CUBIC** becomes **QUAD** if only 3 k's are used in the **KLIST**) (Refer to the Version 9.0 Release Notes for more information).
9. The **ORIG**inal fit (documented in the Theoretical Manual) is a cubic fit over all the hard point k's. Its use is not recommended since it tends to experience numerical problems for any but small k ranges and small numbers of k's.
10. For all fitting options, the generalized aerodynamic matrices are normalized by the hard point k value before fitting, as documented on the Theoretical Manual.
11. Equivalent velocity is defined as the true velocity multiplied by the density ratio (See Remark 3).

12. When **PKIT** is selected, the fields **GFLUT** and **GFILTER** effect the flutter crossings reported during a flutter analysis. **GFLUT** defines the damping value at which flutter occurs. **GFILTER** is used to filter out crossings of lightly damped modes. A flutter crossing will only be identified if the damping in the mode drops below **GFILTER** before exceeding **GFLUT**. This allows lightly damped modes to be filtered even if **GFLUT** otherwise defines a flutter crossing, i.e. has damping of zero. The figure below shows two example curves. **GFLUT** is **0.005** and **GFILTER** is **-0.03**. Point 1 on Curve A would not be considered a flutter crossing even though the curve exceeds **GFLUT**. This occurs because the damping was not less than **GFILTER** before **GFLUT** was exceeded. Point 2 on Curve B would be reported as a flutter crossing at the velocity where the curve crosses **GFLUT**. **GFLUT** and **GFILTER** have no effect during optimization.



Input Data Entry: **FORCE** Static Load

Description: Defines a static load at a grid point by specifying a vector.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		
FORCE	2	5	6	2.9	0.0	1.0	0.0		

Field	Contents
SID	Load set identification number (Integer > 0)
G	Grid point identification number (Integer > 0)
CID	Coordinate system identification number (Integer ≥ 0, or blank) (Default = 0)
F	Scale factor (Real)
Ni	Components of a vector measured in the coordinate system defined by **CID** (Real; must have at least one nonzero component)

Remarks:

1. The static load applied to grid point **G** is given by

   $$\{f\} = F \{N\}$$

   where **{N}** is the vector defined in Fields 6, 7 and 8.

2. A **CID** of zero references the basic coordinate system.

Input Data Entry: **FORCE1** Static Load, Alternate Form 1

Description: Used to define a static load by specification of a value and two grid points which determine the direction.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				
FORCE1	6	13	-2.93	16	13				

Field	Contents
SID	Load set identification number (Integer > 0)
G	Grid point identification number (Integer > 0)
F	Value of load (Real)
Gi	Grid point identification numbers (Integer > 0; **G1** ≠ **G2**)

Remarks:

1. The direction of the force is determined by the vector from **G1** to **G2.**

Input Data Entry: **FREQ**

Description: Defines a set of frequencies to be used in the solution of frequency response problems.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	CONT
CONT	F8	F9	-etc-						CONT

FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	ABC
+BC	29.2	22.4	19.3						

Field	Contents
SID	Frequency set identification number (Integer > 0)
Fi	Frequency value (Real $\geq$ 0.0)

Remarks:

1. The units for the frequencies are cycles per unit time.
2. Frequency sets must be selected by the Solution Control (**FSTEP=SID**) to be used.
3. All **FREQ**, **FREQ1** and **FREQ2** entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. **fN** and **fN-1** are considered duplicated if

$$| f_N - f_{N-1} | < 10^{-5} * (f_{MAX} - f_{MIN})$$

Input Data Entry: **FREQ1**

Description: Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and number of increments desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					
FREQ1	6	2.9	0.5	13					

Field	Contents
SID	Frequency set identification number (Integer > 0)
F1	First frequency in set (Real ≥ 0.0)
DF	Frequency increment (Real > 0.0)
NDF	Number of frequency increments (Integer > 0)

Remarks:

1. The units for the frequency **F1** and the frequency increment **DF** are cycles per unit time.
2. The frequencies defined by this entry are given by:

$$\mathbf{fi = F1 + (i-1)\ DF,\ i = 1,\ NDF + 1}$$

3. Frequency sets must be selected by the Solution Control (**FSTEP=SID**) to be used.
4. All **FREQ**, **FREQ1** and **FREQ2** entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. **fN** and **fN-1** are considered duplicated if

$$| f_N - f_{N-1} | < 10^{-5} * (f_{MAX} - f_{MIN})$$

Input Data Entry: **FREQ2**

Description: Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and number of logarithmic increments desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					
FREQ2	6	1.0	8.0	6					

Field	Contents
SID	Frequency set identification number (Integer > 0)
F1	First frequency (Real > 0.0)
F2	Last frequency (Real > 0.0; **F2** > **F1**)
NF	Number of logarithmic intervals (Integer > 0)

Remarks:

1. The units for the frequencies **F1** and **F2** are cycles per unit time.
2. The frequencies defined by this entry are given by:

$$f_i = F_1 e^{(i-1)d}$$

where,

$$d = \left(\frac{1}{NF}\right) \ln\left(\frac{f_2}{f_1}\right)$$

For the example shown, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.

3. Frequency sets must be selected by the Solution Control (**FSTEP=SID**) to be used.
4. All **FREQ**, **FREQ1** and **FREQ2** entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. **fN** and **fN-1** are considered duplicated if:

$$| f_N - f_{N-1} | < 10^{-5} * (f_{MAX} - f_{MIN})$$

Input Data Entry: **FREQLIST**

Description: Defines a list of frequencies for which outputs are defined.

Format and Example:

1	2	3	4	5	6	7	8	9	10
FREQLIST	SID	FREQ1	FREQ2	FREQ3	FREQ4	FREQ5	FREQ6	FREQ7	CONT
CONT	FREQ8	FREQ9	-etc-						

FREQLIST	100	10.0	20.0	50.0	100.0				

Field	Contents
SID	Set identification number referenced by Solution Control (Integer > 0)
FREQi	Frequency (in Hertz) at which outputs are desired (Real)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. The nearest frequency to **FREQ,** either above or below, which was used in the Frequency Response analysis will be used to satisfy the output requests.
3. Any number of continuations is allowed.

Input Data Entry **GDVLIST** Global Design Variable List

Description: Defines a list of global design variables for which outputs are desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GDVLIST	SID	GDVID1	GDVID23	GDVID3	GDVID4	GDVID5	GDVID6	GDVID7	CONT
CONT	GDVID8	GDVID9	-etc-						

GDVLIST	100	1	2	3	5	7	9		

Alternate Form

1	2	3	4	5	6	7	8	9	10
GDVLIST	SID	GDVID1	THRU	GDVID2					

Field	Contents
SID	Set identification number referenced by Solution Control (Integer > 0)
GDVID	Global design variable identification number (Integer > 0 or blank)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. If the alternate form is used, **GDVID2** must be greater than or equal to **GDVID1**.
3. Nonexistent global design variables may be referenced and will result in no error message.
4. Any number of continuations is allowed, except when using the alternate form, which allows no continuations.

Input Data Entry **GENEL** General Element

Description: Defines a general element of the structural model by a stiffness or flexibility matrix.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		GIDI1	DOFI1	GIDI2	DOFI2	GIDI3	DOFI3	CONT
CONT	GIDI4	DOFI4	CONTINUES IN GROUPS OF 2						CONT
CONT	"**UD**"		GIDD1	DOFD1	GIDD2	DOFD2	GIDD3	DOFD3	CONT
CONT	GIDD4	DOFD4	CONTINUES IN GROUPS OF 2						CONT
CONT	"**K**"/"**Z**"	K11	K21	K31	...	K22	K32	K42	CONT
CONT	...	K33	K43	K53	CONTINUES WITH LIST OF TERMS				CONT
CONT	"**S**"	S11	S12	S13	S14	...	S21	S22	CONT
CONT	S23	...	S31	S32	CONTINUES WITH LIST OF TERMS				CONT

GENEL	3000		3	1	3	2	3	6	+G31
+G31	UD		4	1	4	2	4	6	+G32
+G32	K	6.0	0.0	0.0	6.0	3.0	2.0		+G33
+G33	S	1.0	0.0	0.0	0.0	1.0	-1.0	0.0	+G34
+G34	0.0	1.0							

Field	Contents
EID	Element identification number. See Remark 1 (Integer>0) (Required)
GIDIi	Grid or scalar point identification numbers of points in the **UI** list. (Integer>0) (Required)
DOFIi	Single degree of freedom corresponding to the points **GIDIi**. (DOF Code) (Required)
"**UD**"	Indicates the start of the **UD** degrees of freedom. (Character) (Required)
GIDDi	Grid or scalar point identification numbers of points in the **UD** list. (Integer>0) (Required)
DOFDi	Single degree of freedom corresponding to the points **GIDDi**. (DOF Code) (Required)
"**K**","**Z**"	Indicates the start of the element stiffness, **K**, or flexibility, **Z**, matrix. (Character) (Required)
Kij	Elements of the **K** or **Z** matrix. See Remark 2 (Real) (Default = 0.0)
"**S**"	Indicates the start of data defining the rigid body, **S**, matrix. (Character) (Required)
Sij	Elements of the **S** matrix. See Remark 3 (Real) (Default = 0.0)

Remarks:

1. Element identification numbers must be unique.
2. The **K** or **Z** matrices are entered as lower triangular matrices by columns. High precision input format may be used.
3. The **S** matrix is entered by rows.
4. There are four distinct sections of data to input; the **UI** list, the **UD** list, the **K** or **Z** matrix, and the **S** matrix.
5. The stiffness approach:

$$\begin{Bmatrix} \boldsymbol{f}_i \\ \boldsymbol{f}_d \end{Bmatrix} = \begin{bmatrix} \boldsymbol{K} & -\boldsymbol{K}\boldsymbol{S} \\ -\boldsymbol{S}^T\boldsymbol{K} & \boldsymbol{S}^T\boldsymbol{K}\boldsymbol{S} \end{bmatrix} \begin{Bmatrix} \boldsymbol{u}_i \\ \boldsymbol{u}_d \end{Bmatrix}, \text{ or}$$

6. The flexibility approach:

$$\begin{Bmatrix} \boldsymbol{u}_i \\ \boldsymbol{f}_d \end{Bmatrix} = \begin{bmatrix} \boldsymbol{Z} & \boldsymbol{S} \\ -\boldsymbol{S}^T & \boldsymbol{O} \end{bmatrix} \begin{Bmatrix} \boldsymbol{f}_i \\ \boldsymbol{u}_d \end{Bmatrix}$$

where

$$\boldsymbol{u}_i = [u_{i1}, u_{i2}, \ldots, u_{im}]^T$$

$$\boldsymbol{u}_d = [u_{d1}, u_{d2}, \ldots, u_{dn}]^T$$

$$\boldsymbol{K}\boldsymbol{Z} = \boldsymbol{I} \text{ or } \boldsymbol{Z} = \begin{bmatrix} \boldsymbol{K}\boldsymbol{Z}_{11} & \boldsymbol{K}\boldsymbol{Z}_{12} & \ldots & \boldsymbol{K}\boldsymbol{Z}_{1m} \\ \cdot & \boldsymbol{K}\boldsymbol{Z}_{22} & \ldots & \cdot \\ \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot \\ \boldsymbol{K}\boldsymbol{Z}_{m1} & \ldots & \ldots & \boldsymbol{K}\boldsymbol{Z}_{mm} \end{bmatrix} \text{ and } \boldsymbol{K}\boldsymbol{Z}^T = \boldsymbol{K}\boldsymbol{Z}$$

$$\boldsymbol{S} = \begin{bmatrix} \boldsymbol{S}_{11} & \ldots & \ldots & \ldots & \boldsymbol{S}_{1n} \\ \cdot & & & & \cdot \\ \cdot & & & & \cdot \\ \boldsymbol{S}_{m1} & \ldots & \ldots & \ldots & \boldsymbol{S}_{mn} \end{bmatrix}$$

The required input is the $\boldsymbol{U}_i$ list and the lower triangular portion of $\boldsymbol{K}$ or $\boldsymbol{Z}$. Additional input may include the $\boldsymbol{U}_d$ list and $\boldsymbol{S}$. If $\boldsymbol{S}$ is input, $\boldsymbol{U}_d$ must also be input. If $\boldsymbol{U}_d$ is input but $\boldsymbol{S}$ is omitted, $\boldsymbol{S}$ is internally calculated. In this case, $\boldsymbol{U}_d$ must contain six and only six degrees of freedom.

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body whose rigid body motions are defined by $\boldsymbol{U}_i = \boldsymbol{S}\ \boldsymbol{U}_d$.

Input Data Entry **GPWG** Weight Generator Data

Description: Contains definition of the location about which to perform grid point weight generation

Format and Example:

1	2	3	4	5	6	7	8	9	10
GPWG	GID/X0	Y0	Z0						

GPWG	10								

Field	Contents
GID	Grid point identification of the **GPWG** reference point (Integer)
X0	X component of basic coordinates of the reference point (Real)
Y0	Y component of basic coordinates of the reference point (Real)
Z0	Z component of basic coordinates of the reference point (Real)

Remarks:

1. Either a grid point identification number or the basic x, y, z components of the reference point may be given.
2. If no **GPWG** data entry exists, the grid point weight generation will be computed about the origin of the basic coordinate system.
3. If more than one **GPWG** entry exists, the first one appearing in the sorted bulk data echo will be used.

Input Data Entry: **GRAV** Gravity Vector

Description: Used to define gravity vectors for use in determining gravity loading for the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	G	N1	N2	N3			
GRAV	1	3	32.2	0.0	0.0	-1.0			

Field	Contents
SID	Set identification number (Integer > 0)
CID	Coordinate system identification number (Integer ≥ 0)
G	Gravity vector scale factor (Real ≠ 0.0)
Ni	Gravity vector components (Real; at least one nonzero component)

Remarks:

1. The gravity vector is defined by **{g} = G{Ni}.** The direction of **{g}** is the direction of free fall.
2. A **CID** of zero references the basic coordinate system.
3. Gravity loads may be combined with "simple loads" (e.g., **FORCE**, **MOMENT**) by specification on a **LOAD** entry or by **GRAV = SID**. Gravity loads with the same **SID** as simple load entries will not be used unless referenced by one of these methods.
4. Load sets must be selected in Solution Control to be used.
5. The units of **G** should be length/sec^2 in consistent length units.

Input Data Entry: **GRDSET** Grid Point Default

Description: Defines default options for Fields 3, 7, and 8 of all GRID entries.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS		

GRDSET		16				32	3456		

Field	Contents
CP	Identification number of coordinate system in which the location of the grid point is defined (Integer ≥ 0)
CD	Identification number of coordinate system in which the displacements are measured at grid point (Integer ≥ 0)
PS	Permanent single-point constraints associated with grid point (any of the digits 1 through 6 with no embedded blanks) (Integer ≥ 0)

Remarks:

1. The contents of Fields 3, 7, or 8 of this entry are assumed for the corresponding fields of any GRID entry whose Field 3, 7, and 8 are blank. If any of these fields on the GRID entry are blank, the default option defined by this entry occurs for that field. If no permanent single-point constraints are desired or one of the coordinate systems is basic, the default may be overridden on the GRID entry by making one of the Fields 3, 7, or 8 zero (rather than blank). Only one **GRDSET** entry may appear in the user's Bulk Data packet.
2. The primary purpose if this entry is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned-joint problems).
3. At least one of the entries **CP**, **CD**, or **PS** must be nonzero.

Input Data Entry: **GRID** Grid Point

Description: Defines the location of a geometric grid point of the structural model, the directions of its displacement, and its permanent single-point constraints.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS		
GRID	2	3	1.0	2.0	3.0		315		

Field	Contents
ID	Grid point identification number (Integer > 0)
CP	Identification number of coordinate system in which the location of the grid point is defined (Integer > 0 or blank)
Xi	Location of the grid point in coordinate system **CP** (Real)
CD	Identification number of coordinate system in which displacements, degrees of freedom, constraints, and solution vectors are defined at the grid point (Integer > 0 or blank)
PS	Permanent single-point constraints associated with grid point (any of the digits 1-6 with no embedded blanks) (Integer > 0 or blank)

Remarks:

1. All grid point identification numbers must be unique with respect to all other structural and scalar points.
2. The meaning of **X1, X2,** and **X3** depend on the type of coordinate system, **CP**, as follows:

TYPE	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ (deg)	Z
Spherical	R	θ (deg)	φ (deg)

   Also see **CORDij** entry descriptions.
3. The collection of all **CD** coordinate systems defined on all GRID entries is called the Global Coordinate System. All degrees-of-freedom, constraints, and solution vectors are expressed in the Global Coordinate System.

Input Data Entry: **GRIDLIST**

Description: Defines a list of points at which outputs are desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GRIDLIST	SID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	CONT
CONT	GID8	GID9	-etc-						

GRIDLIST	100	1001	1010	1020					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
GRIDLIST	SID	GID1	THRU	GID2					

Field	Contents
SID	Set identification number referenced by Solution Control (Integer > 0)
GIDi	Grid, scalar or extra point id at which outputs are desired (Integer > 0)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. If the alternate form is used, **GID2** must be greater than or equal to **GID1**.
3. Nonexistent points may be referenced and will result in no error message.
4. Any number of continuations is allowed.

Input Data Entry: **GUST** Aerodynamic Gust Load Description

Description: Defines a stationary vertical gust for use in aeroelastic analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
GUST	SID	GLOAD	WG	XO	V	QDP	MACH		CONT
CONT	SYMXZ	SYMXY							

GUST	133	61	1.0	0.	1.+4	13.5	0.9		ABC
+BC	1	0							

Field	Contents
SID	Gust set identification number (Integer > 0)
GLOAD	The **SID** of a **TLOAD** or **RLOAD** data entry which defines the time or frequency dependence (Integer > 0)
WG	Scale factor (gust velocity/forward velocity) for gust velocity (Real ≠ 0.)
XO	Location of reference plane in aerodynamic coordinates (Real).
V	Velocity of vehicle (Real > 0.0)
QDP	Dynamic pressure (Real > 0.0)
MACH	Mach number (Real ≥ 0.0)
SYMXZ, SYMXY	Symmetry flags associated with aerodynamics (Integer) **+1** Symmetric **0** or Blank Asymmetric **-1** Antisymmetric

Remarks:

1. The **GUST** entry is selected as a discipline option for **FREQUENCY** or **TRANSIENT** in Solution Control.
2. The gust angle is in the +z direction of the aerodynamic coordinate system. The value is,

$$WG = T\ [\ t - \frac{x - x_o}{v}\ ]$$

   where T is the tabular function.

3. The symmetry flags will be used to select the appropriate unsteady aerodynamic matrices from the list of m-k pairs for each symmetry option given on the **MKAEROi** entries.

Input Data Entry: **IC** Transient Initial Condition

Description: Defines values for the initial conditions of coordinates used in transient analysis. Both displacement and velocity values may be specified at independent coordinates of the structural model.

Format and Example:

1	2	3	4	5	6	7	8	9	10
IC	SID	G	C	UO	VO				
IC	1	3	2	5.0	-6.0				

Field	Contents
SID	Set identification number (Integer > 0)
G	Grid or scalar or extra point identification number (Integer > 0)
C	Component number (blank or zero for scalar or extra points, any one of the digits 1 through 6 for a grid point)
UO	Initial displacement value (Real)
VO	Initial velocity value (Real)

Remarks:

1. Transient initial condition sets must be selected in the Solution Control (**IC=SID**) to be used.
2. If no **IC** set is selected, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on **IC** entries will be assumed zero.
4. Initial conditions may be used only in direct formulation. In a modal formulation the initial conditions are all zero.

Input Data Entry **ITERLIST** Iteration List

Description: Defines a list of iteration steps for which outputs are desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
ITERLIST	SID	ITER	ITER	ITER	ITER	ITER	ITER	ITER	CONT
CONT	ITER	ITER	-etc-						

ITERLIST	100	1	2	3	5	7	9		

Alternate Form:

1	2	3	4	5	6	7	8	9	10
ITERLIST	SID	ITER	THRU	ITER					

Field	Contents
SID	Set identification number referenced by Solution Control. (Integer > 0)
ITER	Iteration step number. (Integer > 0 or blank)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. Nonexistent iteration steps may be referenced and will result in no error message.
3. Any number of continuations is allowed, except when using the alternate form, which allows no continuations.

Input Data Entry: **JSET** Select Coordinates for the j-set

Description: Defines coordinates (degrees of freedom) that the user desires to use in the computation of inertia relief mode shape in Dynamic Reduction.

Format and Example:

1	2	3	4	5	6	7	8	9	10
JSET	SETID	ID	C	ID	C	ID	C		CONT
CONT	ID	C	ID	C	-etc-				

JSET	16	2	23	3516					

Field	Contents
SETID	The set identification number of the **INERTIA** set. (Integer > 0)
ID	Grid or scalar point identification number (Integer > 0).
C	Component number, zero or blank for scalar points, any unique combination of the digits 1 through 6 for grid points. (Integer)

Remarks:

1. Coordinates specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets.
2. When **JSET** and/or **JSET1** entries are present, all degrees of freedom not otherwise constrained will be placed on the o-set.
3. Use of **JSET** in dynamic reduction:

   a. **JSET** defines the structural/nonstructural j-set degrees of freedom (inertia relief shapes). An alternate input format is provided by the **JSET1** entry.

   b. The **SID** is selected by the Solution Control Command **BOUNDARY INERTIA** = n.

   c. Use "0" as the grid point identification number to select the origin of the basic coordinate system as one of the j-set degrees of freedom.
4. Any number of continuations are allowed.

Input Data Entry: **JSET1** Select Coordinates for the j-set, Alternate Form

Description: Defines coordinates (degrees of freedom) that the user desires to use in the computation of inertia relief mode shape(s) in Dynamic Reduction.

Format and Example:

1	2	3	4	5	6	7	8	9	10
JSET1	SETID	C	GID1	GID2	GID3	GID4	GID5	GID6	CONT
CONT	GID7	GID8	-etc-						

JSET1	345	2	1	3	10	9	6		ABC
+bc	7	8							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
JSET1	SETID	C	GID1	THRU	GID2				

Field	Contents
SETID	The **INERTIA** set identification number
C	Component number (any unique combination of the digits 1 through 6 (with no embedded blanks) when point identification numbers are grid points; must be blank or zero if point identification numbers are scalar points.
GIDi	Grid or scalar point identification numbers (Integer > 0).

Remarks:

1. Coordinates specified on this entry form members of a set that is exclusive from other sets defined by bulk data entries.
2. When **JSET** and/or **JSET1** entries are present, all degrees of freedom not otherwise constrained will be placed in the o-set.
3. If the alternate form is used, all points in the sequence **ID1** through **ID2** are required to exist and **ID2** must be greater than or equal to **ID1**.
4. Use of **JSET1** in dynamic reduction:

   a. **JSET1** defines the structural and nonstructural j-set degrees of freedom (inertia relief shapes). An alternate input format is provided by the **JSET** entry.

   b. The **SID** is selected by Solution Control Command **BOUNDARY INERTIA** = n.

   c. Use "0" as the grid point identification number to select the origin of the basic coordinate system as one of the j-set degrees freedom.

Input Data Entry **LDVLIST** Local Design Variable List

Description: Defines a list of local design variables for which outputs are desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
LDVLIST	SID	ETYPE	LAYER	DVSYMBL	EID1	EID2	EID3	EID4	CONT
CONT	EID5	EID6	-etc-						

LDVLIST	100	QUAD4	2	100	100	200	300	700	

Alternate Form

1	2	3	4	5	6	7	8	9	10
LDVLIST	SID	ETYPE	LAYER	EID1	THRU	EID2			

Field	Contents
SID	Set identification number referenced by Solution Control. (Integer > 0)
ETYPE	Character input identifying the element type. One of the following:

BAR	**MASS**	**ROD**	**TRMEM**
CONM2	QDMEM1	**SHEAR**	
ELAS	QUAD4	**TRIA3**	

LAYER	Layer number if element is composite laminate. (Integer > 0 or blank)
DVSYMBL	Character symbol specifying the **PBAR1** cross-sectional parameter if **ETYPE** is **PBAR**.

D1	**D2**	**D3**	**D4**	**D5**
D6	**D7**	**D8**	**D9**	**D10**

EIDi	Element identification number. (Integer > 0 or blank)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. If the alternate form is used **EID2** must be greater than or equal to **EID1**.
3. Nonexistent elements may be referenced and will result in no error message.
4. If a layer number is omitted for a composite laminate element then all layers in that element will be selected.
5. Any number of continuations is allowed.
6. See the **PBAR1** Bulk Data entry for a description of the cross-sectional parameters.

Input Data Entry: **LOAD** Static Load Combination (Superposition)

Description: Defines a static load as a linear combination of load sets defined using **FORCE, MOMENT, FORCE1, MOMENT1, PLOAD**, and **GRAV** entries.

Format and Example:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	CONT
CONT	S4	L4							

LOAD	101	-0.5	1.0	3	6.2	4	4.5	10	ABC
+BC	2.3	115							

Field	Contents
SID	Load set identification number (Integer >0)
S	Scale factor (Real ≠ 0.0)
Si	Scale factors (Real ≠ 0.0)
Li	Load set identification numbers defined via data entry types enumerated above (Integer > 0)

Remarks:

1. The load vector defined is given by

   $$P = S \sum S_i L_i$$

2. The **Li** must be unique. The remainder of the physical entry containing the last entry must be blank.
3. Load sets must be selected in the Solution Control if they are to be applied to the structural model.
4. A **LOAD** entry may not reference a set identification number defined by another **LOAD** entry.

Input Data Entry: **MACHLIST**

Description: Defines a list of Mach numbers.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MACHLIST	SID	MACH1	MACH2	MACH3	MACH4	MACH5	MACH6	MACH7	CONT
CONT	MACH8	MACH9	-etc-						

MACHLIST	201	1.0	0.5	0.7					

Field	Contents
SID	Mach set identification number (Integer > 0)
MACHi	Mach number (Real > 0.0)

Remarks:

1. **MACHLIST** Bulk Data entries are selected in the Function Packet.

Input Data Entry: **MAT1** Material Property Definition, Form 1

Description: Defines the material properties for linear, temperature-independent, isotropic materials

Format and Example:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	CONT
CONT	ST	SC	SS	MCSID					

MAT1	17	3.+7		0.33	4.28	6.5-6	5.37-6	0.23	ABC
+B	20.+4	15.+4	12.+4						

Field	Contents
MID	Material identification number (Integer >0)
E	Young's modulus (Real > 0.0, or blank)
G	Shear modulus (Real or blank)
NU	Poisson's ratio (–1.0 < Real ≤ 0.5 or blank)
RHO	Mass density (Real ≥ 0.0)
A	Thermal expansion coefficient (Real)
TREF	Thermal expansion reference temperature (Real)
GE	Structural element damping coefficient (Real)
ST, SC, SS	Stress limits for tension, compression, and shear (Real). (Used to compute margins of safety in certain elements).
MCSID	Material Coordinate System identification number (Integer > 0 or blank).

Remarks:

1. The material identification number must be unique for all **MAT1**, **MAT2**, **MAT8**, and **MAT9** bulk data entries.
2. The mass density, **RHO**, will be used to automatically compute mass for all structural elements.
3. Weight density may be used in Field 6 if the value 1/g is entered on the **CONVERT** entry where g is the acceleration of gravity.
4. Either **E** or **G** must be specified (i.e., nonblank).
5. If any one of **E**, **G**, or **NU** is blank, it will be computed to satisfy the identity **E = 2 *(1+NU)*G**; otherwise, values supplied by the user will be used.
6. If **E** and **NU** or **G** and **NU** are both blank, they will both be given the values 0.0.
7. Implausible data on one or more **MAT1** entries will result in a warning message. Implausible data is defined as any of **E <0.0 or G < 0.0 or NU > 0.5 or NU < 0.0 or |1 - E/(2(1+NU)G)| >0.01** except for cases covered by Remark 6.
8. It is strongly recommended that only two of the three values **E**, **G**, and **NU** be input. The three values may be input independently on the **MAT2** entry.

Input Data Entry: **MAT2** Material Property Definition, Form 2

Description: Defines the material properties for linear, temperature-independent, anisotropic materials for two-dimensional elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	CONT
CONT	A1	A2	A12	TO	GE	ST	SC	SS	CONT
CONT	MCSID								

MAT2	13	6.2+3			6.2+3		5.1+3	0.056	ABC
+BC	6.5-6	6.5-6		-500.0	0.002	20.+5			

Field	Contents
MID	Material identification number (Integer > 0)
Gij	The material property matrix (Real)
RHO	Mass density (Real ≥ 0.0)
Ai	Thermal expansion coefficient vector (Real)
TO	Thermal expansion reference temperature (Real)
GE	Structural element damping coefficient (Real)
ST, SC, SS	Stress limits for tension, compression, and shear (Real). (Used to compute margins of safety in certain elements).
MCSID	Material Coordinate System identification number (Integer > 0 or blank).

Remarks:

1. Material identification numbers must be unique for all **MAT1**, **MAT2**, **MAT8**, and **MAT9** bulk data entries.
2. The mass density, **RHO**, will be used to automatically compute mass for all structural elements.
3. Weight density may be entered in Field 9 if the value 1/g, where g is the acceleration of gravity, is entered on the **CONVERT** entry.
4. The convention for the $\mathbf{G}_{ij}$ in Fields 3 through 8 are represented by the matrix relationship

$$\begin{Bmatrix}\sigma_1\\ \sigma_2\\ \tau_{12}\end{Bmatrix} = \begin{bmatrix}G_{11} & G_{12} & G_{13}\\ G_{12} & G_{22} & G_{23}\\ G_{13} & G_{23} & G_{33}\end{bmatrix}\left(\begin{Bmatrix}\varepsilon_1\\ \varepsilon_2\\ \gamma_{12}\end{Bmatrix} - (T - T_o)\begin{Bmatrix}A_1\\ A_2\\ A_{12}\end{Bmatrix}\right)$$

5. 2x2 matrices (for example, transverse shear) use elements **G11, G12**, and **G22**. For this case, **G33** must be blank.
6. If the **MAT2** entry is referenced by the **PCOMP** entry, the transverse shear flexibility for the referenced laminae is zero.
7. Unlike the **MAT1** entry, data from the **MAT2** entry are used directly, without adjustment of equivalent **E**, **G**, or **NU** values.

Input Data Entry: **MAT8** Material Property Definition, Form 8

Description: Defines the material property for an orthotropic material.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1, Z	G2, Z	RHO	CONT
CONT	A1	A2	TREF	Xt	Xc	Yt	Yc	S	CONT
CONT	GE	F12							

MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	+ABC
+BC	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	+DEF
+EF	1.-4								

Field	Contents
MID	Material identification number (Integer > 0)
E1	Modulus of elasticity in longitudinal direction (also defined as fiber direction or 1-direction) (Real ≠ 0.0)
E2	Modulus of elasticity in lateral direction (also defined as matrix direction or 2-direction) (Real ≠ 0.0)
NU12	Poisson's ratio [$\frac{(\varepsilon_2)}{(\varepsilon_1)}$ for uniaxial loading in 1-direction]. Note that **NU21** = $\frac{(\varepsilon_1)}{(\varepsilon_2)}$ for uniaxial loading in 2-direction is related to **NU12, E1, E2** by the relation **NU12*E2 = NU21*E1**. (Real)
G12	In-plane shear modulus (Real > 0.0)
G1,Z	Transverse shear modulus for shear in 1-Z plane (Real > 0.0 or blank) (default implies infinity)
G2,Z	Transverse shear modulus for shear in 2-Z plane (Real > 0.0 or blank) (default implies infinity)
RHO	Mass density (Real ≥ 0.0)
A1	Thermal expansion coefficient in the 1-direction (Real)
A2	Thermal expansion coefficient in the 2-direction (Real)
TREF	Thermal expansion reference temperature (Real)
Xt, Xc	Allowable stresses in tension and compression, respectively, in the longitudinal direction. Required if failure index is desired. (Real ≥ 0.0) (Default value for **Xc** is **Xt)**
Yt, Yc	Allowable stresses in tension and compression, respectively, in the transverse direction. Required if failure index is desired. (Real ≥ 0.0) (Default value for **Yc** is **Yt**)
S	Allowable stress for in-plane shear (Real ≥ 0.0)
GE	Structural damping coefficient (Real)

F12 Interaction term in the tensor polynomial theory of Tsai-Wu (Real). Required if failure index or stress constraint by Tsai-Wu theory is desired and if value of F_{12} is different from 0.0.

Remarks:

1. If **$G_{1,Z}$** and **$G_{2,Z}$** values specified as zero, or are not supplied, transverse shear flexibility calculations will not be performed.
2. An approximate value for **$G_{1,Z}$** and **$G_{2,Z}$** is the in-plane shear modulus **G_{12}**.. If test data are not available to accurately determine **$G_{1,Z}$** and **$G_{2,Z}$** for the material and transverse shear calculations are deemed essential, the value of **G_{12}** may be supplied for **$G_{1,Z}$** and **$G_{2,Z}$**.
3. **X_{t}**, **X_{c}**, **Y_{t}**, **Y_{c}** and **SS** are used for composite element failure calculations when requested in the **FT** field of the **PCOMPi** entry.
4. The mass density, **RHO**, is used to automatically compute mass for all structural elements.
5. Weight density may be entered in Field 9 if the value 1/g, where g is the acceleration of gravity, is entered on the **CONVERT** entry.

Input Data Entry: **MAT9** Material Property Definition, Form 9

Description: Defines the material properties for linear, temperature-independent, anistropic materials for solid isoparametric elements

Format and Example:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	CONT
CONT	G23	G24	G25	G26	G33	G34	G35	G36	CONT
CONT	G44	G45	G46	G55	G56	G66	RHO	A1	CONT
CONT	A2	A3	A4	A5	A6	TREF	GE		

MAT9	17	6.2+3						6.2+3	ABC
+BC									DEF
+EF	5.1+3			5.1+3		5.1+3	3.2	6.6-6	

Field	Contents
MID	Material identification number (Integer > 0)
Gij	Elements of the 6x6 symmetric material property matrix (Real ≥ 0.0)
RHO	Mass density (Real ≥ 0.0)
Ai	Thermal expansion coefficient vector (Real)
TREF	Thermal expansion reference temperature (Real)
GE	Structural element damping coefficient (Real)

Remarks:

1. The material identification numbers must be unique for all **MAT1**, **MAT2**, **MAT8**, and **MAT9** entries.
2. The mass density **RHO** will be used to automatically compute mass in a structural dynamics problem.
3. Weight density may be entered in Field 9 if the value 1/g, where g is the acceleration of gravity, is entered on the **CONVERT** entry.
4. Continuation number 4 need not be used.
5. The subscripts 1 through 6 refer to x, y, z, xy, yz, zx, for example:

$$\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{zx}\end{Bmatrix} = \begin{bmatrix} G_{11} & & & & & \\ G_{12} & G_{22} & & \text{SYM} & & \\ G_{13} & G_{23} & G_{33} & & & \\ G_{14} & G_{24} & G_{34} & G_{44} & & \\ G_{15} & G_{25} & G_{35} & G_{45} & G_{55} & \\ G_{16} & G_{26} & G_{36} & G_{46} & G_{56} & G_{66} \end{bmatrix} \left(\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_z\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{zx}\end{Bmatrix} - \begin{Bmatrix}A_1\\ A_2\\ A_3\\ A_4\\ A_5\\ A_6\end{Bmatrix} (T - T_o) \right)$$

6. The damping coefficient, **GE** is:

$$GE = 2\frac{C}{C_0}$$

Input Data Entry: **MFORM** Mass Matrix Form

Description: Defines the form of the mass matrix as consistent (coupled) or lumped.

Format and Example:

1	2	3	4	6	5	7	8	9	10
MFORM	VALUE								

MFORM	LUMPED								

Field	Contents
VALUE	A character string denoting the form of the mass matrix. The available forms are: 1) **LUMPED** 2) **COUPLED**

Remarks:

1. If more than one **MFORM** is included in the Bulk Data, any **COUPLED** value will result in coupled mass being used.
2. If no **MFORM** is indicated, the **LUMPED** formulation will be used.

Input Data Entry: **MKAERO1** Mach Number - Frequency Table

Description: Provides a table of Mach numbers (**m**) and reduced frequencies (**k**) for unsteady aerodynamic matrix calculation.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MKAERO1	SYMXZ	SYMXY	m_1	m_2	m_3	m_4	m_5	m_6	CONT
CONT	k_1	k_2	k_3	k_4	k_5	k_6	k_7	k_8	

MKAERO1	1	0	0.1	0.7					+ABC
+ABC	0.3	0.6	1.0						

Field	Contents
SYMXZ, SYMXY	Symmetry flags (Integer). See Remarks 4 and 6.
m_i	List of from 1 to 6 Mach numbers (Real ≥ 0.0 or blank)
k_j	List of from 1 to 8 reduced frequencies (Real ≥ 0.0 or blank)

Remarks:

1. All combinations of (**m,k**) will be used.
2. The continuation entry is required.
3. Several **MKAEROi** entries may be in the input packet. If these data entries are in the packet, they will be used.
4. The symmetry flags have the following definition:
   +1 for symmetric (Cannot be used with **SYMXY** option)
   0 for asymmetric
   -1 for antisymmetric

   The **m-k** pairs generated by this entry will generate aerodynamic matrices having the symmetries selected.
5. **m-k** pairs may be repeated with different symmetry options.
6. These are the following restrictions associated with the symmetry flags:
   a) Ground effect is limited to antisymmetric only, **SYMXY = 0** or **-1**.
   b) Ground effect is not available at all for supersonic flow.
7. Reduced frequency is computed using:

$$k = \frac{b\omega}{2v}$$

   where b is the reference chord defined by an **AERO** entry, ω is the frequency in radians per sec, and v is the true velocity.

Input Data Entry: **MKAERO2** Mach Number - Frequency Table

Description: Provides a list of Mach numbers (**m**) and reduced frequencies (**k**) for aerodynamic matrix calculation.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MKAERO2	SYMXZ	SYMXY	m_1	k_1	m_2	k_2	m_3	k_3	CONT
CONT	m_4	k_4	m_5	k_5	-etc-				

MKAERO2	0	0	0.10	0.60	0.70	0.30	0.70	1.0	ABC
+BC	0.8	0.9	0.8	1.0					

Field	Contents
SYMYZ, SYMXY	Symmetry flags (Integer). See Remarks 4 and 6.
mi,ki	List of pairs of Mach numbers (Real ≥ 0.) and reduced frequencies (real ≥ 0.)

Remarks:

1. This entry will cause the aerodynamic matrices to be computed for the given sets of parameter pairs.
2. Several **MKAEROi** entries may be in the input packet. If these data entries are in the packet, they will be used.
3. Any number of continuations are allowed.
4. The symmetry flags have the following definition:
   +1 for symmetric (Cannot be used with **SYMXY** option)
   0 for asymmetric
   -1 for antisymmetric

   The **m**-**k** pairs listed on the entry will generate aerodynamic matrices having the symmetries selected.
5. **m**-**k** pairs may be repeated with different symmetry options.
6. The following restrictions are imposed on the symmetry flags:
   a) Ground effect (if present) must be antisymmetric **SYMXY = 0** or **-1**.
   b) Ground effect is not available at all for supersonic flow.
7. Reduced frequency is computed using:

$$k = \frac{b\omega}{2v}$$

   where b is the reference chord defined by an **AERO** entry, ω is the frequency in radians per sec, and v is the true velocity.

Input Data Entry: **MODELIST**

Description: Defines a list of modes at which outputs are desired.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MODELIST	SID	MODE1	MODE2	MODE3	MODE4	MODE5	MODE6	MODE7	CONT
CONT	MODE8	MODE9	-etc-						CONT
MODELIST	100	1	2	4					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
MODELIST	SID	MODE1	THRU	MODE2					

Field	Contents
SID	Set identification number referenced by Solution Control (Integer > 0)
MODEi	Mode number of mode at which outputs are desired. (Integer > 0)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. If the alternate form is used **MODE2** must be greater than or equal to **MODE1**.
3. Modes are numbered from 1 to n, starting at the lowest frequency for which a eigenvector was computed.
4. Nonexistent modes may be referenced and will result in no error message.

Input Data Entry: **MOMENT** Static Moment

Description: Defines a static moment at a grid point by specifying a vector.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		
MOMENT	2	5	6	2.9	0.0	1.0	0.0		

Field	Contents
SID	Load set identification number (Integer > 0)
G	Grid point identification number (Integer > 0)
CID	Coordinate system identification number (Integer ≥ 0)
M	Scale factor (Real)
Ni	Components of vector measured in coordinate system defined by **CID** (Real; at least one nonzero component)

Remarks:

1. The static moment applied to grid point **G** is given by
   {m} = M {N}
2. A **CID** of zero references the basic coordinate system.

Input Data Entry: **MOMENT1** Static Moment, Alternate Form 1

Description: Defines a static moment by specification of a value and two grid points which determine the direction.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	M	G1	G2				
MOMENT	6	13	-2.93	16	13				

Field	Contents
SID	Load set identification number (Integer > 0)
G	Grid point identification number (Integer > 0)
M	Value of moment (Real)
Gi	Grid point identification numbers (Integer > 0; **G1** ≠ **G2**)

Remarks:

1. The direction of the moment vector is determined by the vector from **G1** and **G2**.

Input Data Entry: **MPC** Multipoint Constraint

Description: Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0.0$$

Format and Example:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G0	C0	A0	G	C	A		CONT
CONT		G	C	A	G	C	A		

MPC	3	28	3	6.2	2	3	4.29		+B
+B		1	4	-2.91					

Field	Contents
SID	Set identification (Integer > 0)
G0,G	Identification number of grid or scalar point (Integer > 0)
C0,C	Component number - any one of the digits 1 through 6 in the case of geometric grid points; blank or zero in the case of scalar points (Integer)
A0,A	Coefficient (Real; **A0** must be nonzero)

Remarks:

1. The first coordinate (**G0**, **C0**) in the sequence is assumed to be the dependent coordinate. A dependent degree of freedom assigned by one **MPC** entry cannot be assigned dependent by another **MPC** entry or by a rigid element.
2. Forces of multipoint constraint are not recovered.
3. Multipoint constraint sets must be selected in Solution Control (**MPC** = **SID**) to be used.
4. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets.

Input Data Entry: **MPCADD** Multipoint Constraint Set Combination

Description: Defines a multipoint constraint set as a union of multipoint constraint sets defined via **MPC** entries.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	CONT
CONT	S8	S9	-etc-						

MPCADD	101	2	3	1	6	4			

Field	Contents
SID	Set identification number (Integer > 0)
S_j	Set identification numbers of multipoint constraint sets defined via **MPC** entries (Integer > 0)

Remarks:

1. The **Sj** must be unique.
2. Multipoint constraint sets must be selected in Solution Control (**MPC** = **SID**) to be used.
3. **Sj** may not be the identification number of a multipoint constraint set defined by another **MPCADD** entry.
4. **MPCADD** entries take precedence over **MPC** entries. If both have the same set identification number, only the **MPCADD** entry will be used.

Input Data Entry: **MPPARM**

Description: Identify values of user defined optimizer parameters that overrides the default values.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MPPARM	PARAM	VALUE	PARAM	VALUE	PARAM	VALUE	PARAM	VALUE	CONT
CONT	PARAM	VALUE	-etc-						

MPPARM	ISCAL	0	STOL	0.005					

Field	Contents
PARAM	Name of parameter to be overridden (Character)
VALUE	Integer or real value to be used for the parameter.

Remarks:

1. Any number of **PARAM**-**VALUE** combinations can be specified on an **MPPARM** entry.
2. See the EDO software manual (ADS V 1.10) for a definition of parameters, but the most useful are shown below:

REAL PARAMETER	DEFINITION	DEFAULT
CT	Constraint tolerance in the Method of Feasible Directions or the Modified Method of Feasible Directions. A constraint is active if its numerical value is more positive than **CT**.	-0.003
CTL	Same as **CT**, but for linear constraints.	-0.003
CTLMIN	Same as **CTMIN**, but for linear constraints.	0.0005
CTMIN	Minimum constraint tolerance for nonlinear constraints. If a constraint is more positive than **CTMIN**, it is considered to be violated.	0.0005
DABOBJ	Maximum absolute change in the objective between two consecutive iterations to indicate convergence in optimization.	$max(0.001\|F_o\|, 0.0001)$
DABOBM	Absolute convergence criterion for the optimization sub-problem when using sequential minimization techniques.	**(Note 3)**
DABSTR	Same as **DABOBJ**, but used at the strategy level.	**(Note 3)**
DELOBJ	Maximum relative change in the objective between two consecutive iterations to indicate convergence in optimization.	0.001

REAL PARAMETER	DEFINITION	DEFAULT
DELOBM	Relative convergence criterion for the optimization sub-problem when using sequential minimization techniques.	**(Note 3)**
DELSTR	Same as **DELOBJ**, but used at the strategy level.	**(Note 3)**
DOBJ1	Relative change in the objective function attempted on the first optimization iteration. Used to estimate initial move in the one-dimensional search. Updated as the optimization progresses.	0.1
DOBJ2	Absolute change in the objective function attempted on the first optimization iteration. Used to estimate initial move in the one-dimensional search. Updated as the optimization progresses.	$0.2 \max(X_i)$
DX1	Maximum relative change in a design variable attempted on the first optimization iteration. Used to estimate initial move in the one-dimensional search. Updated as the optimization progresses.	0.01
DX2	Maximum absolute change in a design variable attempted on the first optimization iteration. Used to estimate initial move in the one-dimensional search. Updated as the optimization progresses.	0.02
EXTRAP	Maximum multiplier on the one-dimensional search parameter, **ALPHA** in the one-dimensional search using polynomial interpolation/extrapolation.	**(Note 3)**
SCFO	The user-simplified value of the scale factor for the objective function if the default or calculated value is to be overridden.	**(Note 3)**
SCLMIN	Maximum numerical value of any scale factor allowed.	**(Note 3)**
STOL	Tolerance on the components of the calculated search direction to indicate that the Kuhn-Tucker conditions are satisfied.	**(Note 3)**
THETAZ	nominal value of the push-off factor in the Method of Feasible Directions.	**(Note 3)**

REAL PARAMETER	DEFINITION	DEFAULT
XMULT	Multiplier on the move parameter, **ALPHA**, in the one-dimensional search to find bounds on the solution.	**(Note 3)**
ZRO	Numerical estimate of zero on the computer. Usually the default value is adequate. If a computer with a short word length is used, **ZRO** = 1.0E-4 may be preferred.	**(Note 3)**

INTEGER PARAMETER	DEFINITION	DEFAULT
ISCAL	Scaling parameter. By default, scaling is done every **NDV** iterations, otherwise scaling is performed every **ISCA** iterations.	**-1**
ITMAX	Maximum number of iterations allowed at the optimizer level.	**40**
ITROMP	The number of consecutive iterations for which the absolute or relative convergence criteria must be met to indicate convergence at the optimizer level.	**2**
ITRMST	The number of consecutive iterations for which the absolute or relative convergence criteria must be met to indicate convergence at the optimizer level.	**(Note 3)**
JTMAX	Maximum of iterations allowed at the strategy level.	**(Note 3)**

3. Some of these parameters, indicated in the tables, are used only with the original version of the ADS optimizer. They are not used in MicroDOT.

Input Data Entry: **OMIT** Omitted Coordinates

Description: Defines degrees of freedom that the user desires to omit from the problem through matrix partitioning. Used to reduce the number of independent degrees of freedom.

Format and Example:

1	2	3	4	5	6	7	8	9	10
OMIT	SETID	ID	C	ID	C	ID	C		
OMIT	10	16	2	23	3516	54	23		

Field	Contents
SETID	The reduce set identification number (Integer > 0).
ID	Grid or scalar point identification number (Integer > 0).
C	Component number, zero or blank for scalar points, any unique combination of the digits 1 through 6 for grid points.

Remarks:

1. Coordinates specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets.
2. In many cases it may be more convenient to use **OMIT1**, **ASET** or **ASET1** entries.

Input Data Entry: **OMIT1** Omitted Coordinates, Alternate Form

Description: Defines degrees of freedom that the user desires to omit from the problem through matrix partitioning. Used to reduce the number of independent degrees of freedom.

Format and Example:

1	2	3	4	5	6	7	8	9	10
OMIT1	SETID	C	GID1	GID2	GID3	GID4	GID5	GID6	CONT
CONT	GID7	GID8	-etc-						

OMIT1	3	2	1	3	10	9	6	5	ABC
+BC	7	8							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
OMIT1	SETID	C	GID1	THRU	GID2				

Field	Contents
SETID	The reduce set identification number (Integer > 0).
C	Component number (Any unique combination of the digits 1 through 6 (with no embedded blanks) when point identification numbers are grid points; must be null or zero if point identification numbers are scalar points).
GIDi	Grid or scalar point identification number (Integer > 0).

Remarks:

1. Coordinates specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets.
2. If the alternate form is used, points in the sequence **ID1** through **ID2** are required to exist and **ID2** must be greater than or equal to **ID1**.

Input Data Entry: **PAER01** Aerodynamic Panel Property

Description: Gives associated bodies for the panels in the unsteady aerodynamic model.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PAERO1	PID	B1	B2	B3	B4	B5	B6		
PAERO1	1	3							

Field	Contents
PID	Property identification number (referenced by **CAER01**) (Integer > 0)
Bi	Identification number of **CAERO2** entries for associated bodies (Integer ≥ 0, or blank)

Remarks:

1. The associated bodies must be in the same aerodynamic group.
2. The **Bi** numbers above must appear on a **CAERO2** entry to define these bodies completely.

Input Data Entry: **PAERO2** Aerodynamic Body Properties

Description: Defines the cross-section properties of unsteady aerodynamic bodies.

Format and Examples:

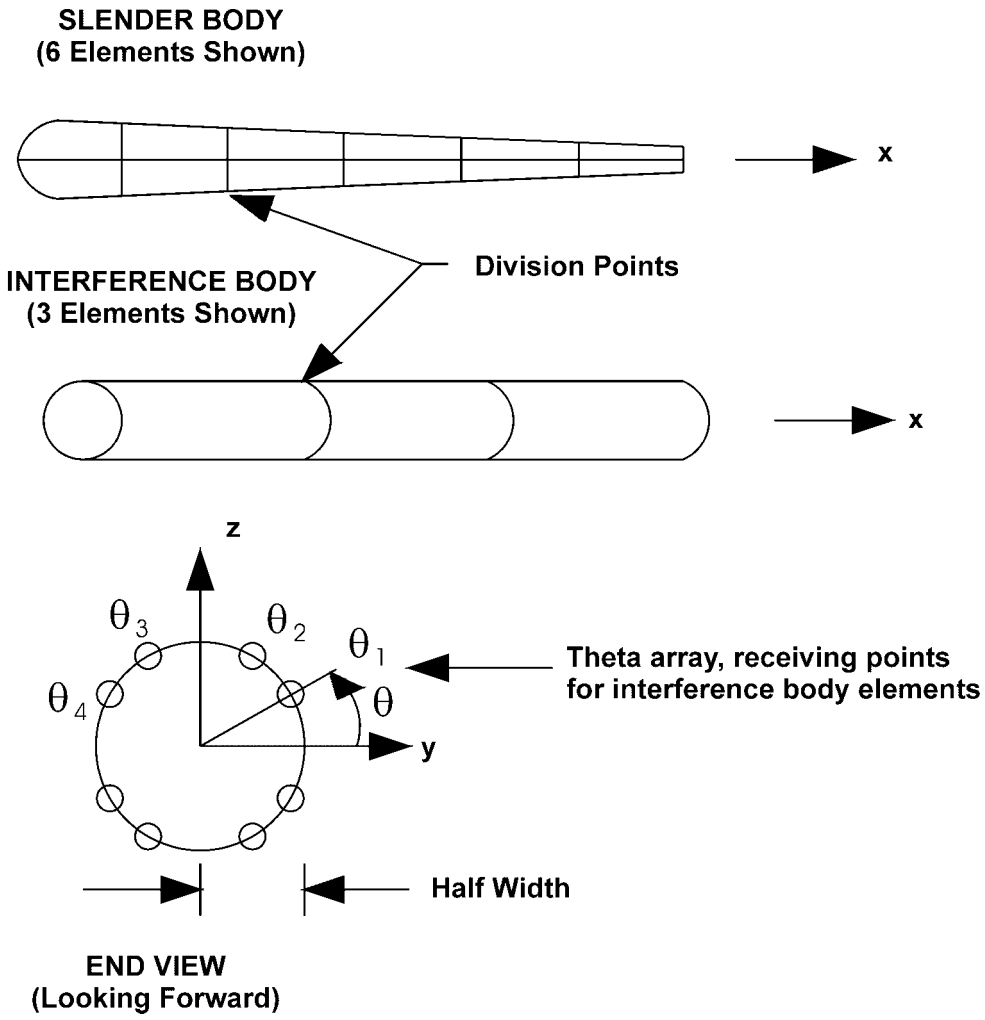
1	2	3	4	5	6	7	8	9	10
PAERO2	PID	ORIENT	WIDTH	AR	LRSB	LRIB	LTH1	LTH2	CONT
CONT	THI1	THN1	THI2	THN2	THI3	THN3			

PAERO2	2	Z	6.0	1.0	22	91	100		abc
+bc	1	3							

Field	Contents
PID	Property identification number (Integer > 0)
ORIENT	Orientation flag "**Z**", "**Y**", or "**ZY**". Type of motion allowed for bodies (Character). Refers to the aerodynamic coordinate system "y" and "z" directions (see **AERO** data entry)
WIDTH	Reference half-width of body (Real > 0.0)
AR	Aspect ratio (height/width) (Real > 0.0)
LRSB	Identification number of an **AEFACT** data entry containing a list of slender body half-widths. If blank, the value of **WIDTH** will be used (Integer > 0 or blank)
LRIB	Identification number of an **AEFACT** data entry containing a list of interference body half-widths. If blank, the value of **WIDTH** will be used (Integer > 0 or blank)
LTH1, LTH2	Identification number of **AEFACT** data entries for defining theta arrays for interference calculations (Integer ≥ 0)
THIi, THNi	The first and last interference element of a body to use the θ_i array (Integer ≥ 0)

Remarks:

1. The **EID** of all **CAERO2** elements in any **IGID** group must be ordered, so that their corresponding **ORIENT** values appear in the order **Z**, **ZY**, **Y**.
2. The half-widths (given on **AEFACT** data entries referenced in fields 6 and 7) are specified at division points. The number of entries on an **AEFACT** data entry used to specify half-widths must be one greater than the number of elements.
3. The half-width at the first point (i.e., the nose) on a slender body is usually 0.; thus, it is recommended (but not required) that the **LRSB** data is supplied with a zero first entry.
4. **THIi** and **THNi** are interference element locations on a body. The element numbering begins at one for each body.
5. A body is represented by a slender body surrounded by an interference body. The slender body creates the down wash due to the motion of the body, while the interference body represents the effects upon panels and other bodies. This is illustrated in the following Figure.


SLENDER BODY
(6 Elements Shown)
x
Division Points
INTERFERENCE BODY
(3 Elements Shown)
x
z
θ3
θ2
θ1
θ4
θ
y
Theta array, receiving points
for interference body elements
Half Width
END VIEW
(Looking Forward)

Input Data Entry: **PAERO6**

Description: Defines body analysis parameters for steady aerodynamics.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PAERO6	BCID	CMPNT	CP	IGRP	NRAD	LRAD	LAXIAL		
PAERO6	10	FUSEL	0	3	4				

Field	Contents
BCID	Body component identification number (Integer > 0)
CMPNT	Component type (**FUSEL** for the fuselage and **POD** for a **POD**)
CP	Coordinate system of the geometry input (Integer ≥ 0, or blank)
IGRP	Group flag (Integer > 0)
NRAD	Number of equal radial cuts used to define the body panels (Integer ≥ 0 or blank)
LRAD	Identification number of an **AEFACT** data entry which defines the angular locations in degrees of the body panels (Integer ≥ 0 or blank)
LAXIAL	Identification number of an **AEFACT** data entry which defines the axial locations of in degrees of the body panels (Integer ≥ 0 or blank)

Remarks:

1. **NRAD** and **LRAD** are mutually exclusive.
2. If **LRAD** iand **NRAD** are zero or blank, the radial cuts specified by the **BODY** or **AXSTA** entries are used.
3. **LAXIAL** is used only for **FUSEL** components. Inputs on the **AEFACT** entry are the dimensional fuselage stations.
4. If **LAXIAL** is blank, the axial locations are the same as those given by **AXSTA** data entries for the given body component.

Input Data Entry: **PBAR** Simple Beam Property

Description: Defines the properties of a simple beam (bar) which is used to create bar elements via the **CBAR** entry.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM	TMIN	CONT
CONT	C1	C2	D1	D2	E1	E2	F1	F2	CONT
CONT	K1	K2	I12	R12	R22	ALPHA			

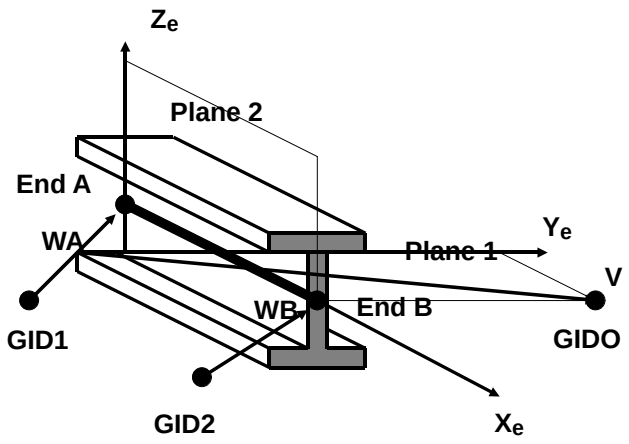
PBAR	39	6	2.9		5.97				123
+23			2.0	4.0					

Field	Contents
PID	Property identification number (Integer > 0)
MID	Material identification number (Integer > 0)
A	Area of bar cross-section (Real ≥ 0.0)
Ii	Area moments of inertia (Real) (**I1** ≥ 0.0, **I2** ≥ 0.0, $I_1 I_2 > I_{12}^2$)
J	Torsional constant (Real ≥ 0.0)
NSM	Nonstructural mass per unit length (Real ≥ 0.0)
TMIN	The minimum cross-sectional area in design (Real, Default = 0.0001)
K1,K2	Area factor for shear (Real)
Ci,Di,Ei,Fi	Stress recovery coefficients (Real)
R12,R22,ALPHA	Inertia linking terms for design (see Remark 6)

Remarks:

1. The BAR element geometry and coordinate system is shown in the Figure on the following page.
2. **PBAR** entries may only reference **MAT1** material entries.
3. The transverse shear stiffnesses in planes 1 and 2 are (**K1**)AG and (**K2**)AG, respectively. The default values for **K1** and **K2** are infinite. In other words, the transverse shear flexibilities are set equal to zero. **K1** and **K2** are ignored if **I12 ≠ 0**.
4. The stress recovery coefficients **C1** and **C2**, etc., are the y and z coordinates in the **BAR** element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the **BAR**.
5. The **TMIN** value is used only for shape function design variable linking.

6. For design, the following applies to the **R12** and **R22** values. The moments of inertia are linked to the



cross-sectional area by the following expressions:

I1 = R12 * AALPHA**

I2 = R22 * AALPHA**

(A) If **R12** = 0.0 then the missing value is computed from **R12=I1/(A**ALPHA)** . The same is true for **R22** and **I2**.
(B) The **ALPHA** value defaults to 1.0 and must be ≥ 1.0.
(C) If both **I1** and **R12** or **I2** and **R22** are given, the linking expression will override the input I_i values.

7. If the **CBAR** is to be designed, the following restrictions apply.
(A) J = NSM = K1 = K2 = I12 = 0.0
If any of these values are not zero, a warning message will be issued and the value set to zero.

Input Data Entry **PBAR1** Geometric BAR element property

Description: Defines the properties of a BAR element by specifying its cross-sectional characteristics.

Format and Example:

1	2	3	4	5	6	7	8	9	10
PBAR1	PID	MID	SHAPE	D1	D2	D3	D4	D5	-cont-
-cont-	NSM			D6	D7	D8	D9	D10	

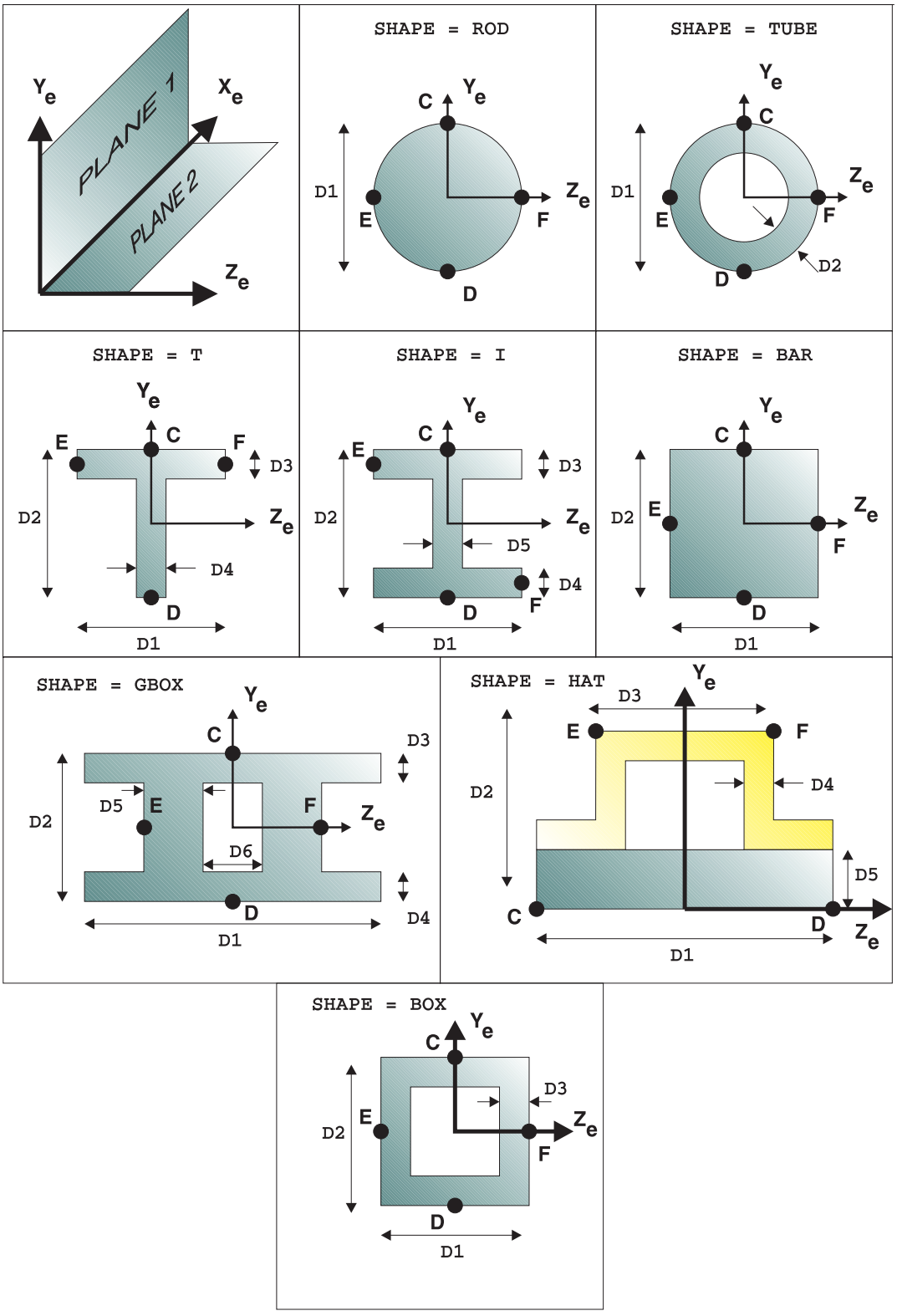
PBAR1	101	56	TUBE	2.0	0.1				+A
+A	1.25								

Field	Contents
PID	Property identification number (Integer>0).
MID	Material identification number (Integer>0). (See Remark 1)
SHAPE	Cross-sectional shape (Character I, T, BOX, BAR, TUBE, ROD, HAT or GBOX). (See Remark 2)
Di	Cross-sectional dimensions (Real>0.0). (See Remark 2)
NSM	Nonstructural mass per unit length (Real).

Remarks:

1. **PBAR1** entries may only reference **MAT1** material data.
2. The cross-sectional properties and shear flexibility factors of the BAR are computed using the **SHAPE** and **Di** geometric data as defined by the figures on the following page. The stress recovery points are also shown. Note that the orientation of the element coordinate system is important for the element definition.

Definition of Cross-Sectional Geometry and Stress Recovery Points



Input Data Entry: **PCOMP** Layered Composite Element Property

Description: Defines the properties of an n-ply composite material laminate.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SBOND	F.T.	TMIN		LOPT	CONT
CONT	MID1	T1	THi	SOUT1	MID2	T2	TH2	SOUT2	CONT
CONT	MID3	T3	TH3	SOUT3	-etc-				

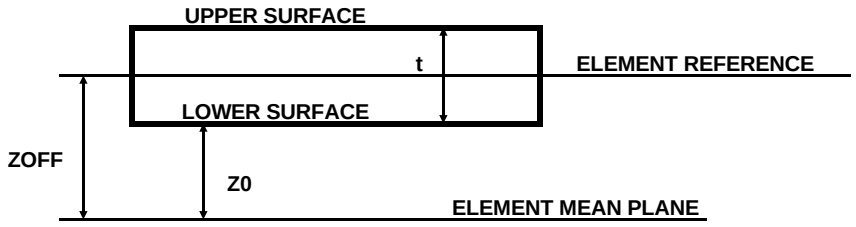
PCOMP	100	-0.5	1.5	5.+3	HOFF			MEM	ABC
+BC	150	0.05	90.	YES			-45.		DEF
+EF			45.0						

Field	Contents
PID	Property identification number (Integer > 0).
Z0	Offset of the laminate lower surface from the element mean plane. A positive value means the $+z_e$ direction. (Real or blank, see Remark 2)
NSM	Nonstructural mass per unit area (Real ≥ 0.0).
SBOND	Allowable shear stress of the bonding material. (Real ≥ 0.0)
F.T.	Failure theory, one of the strings **HILL**, **HOFF**, **TSAI**, **STRESS**, or **STRAIN**. See Remark 4.
TMIN	Minimum ply thickness for design (Real > 0.0 or blank). (Default = 10^{-4})
LOPT	Lamination generation option, **MEM** or blank. (See Remark 5).
MIDi	Material identification number of the *i-th* layer. (Integer > 0 or blank)
T_i	Thickness of the i(th) layer (Real > 0.0 or blank).
TH_i	Angle between the longitudinal direction of the fibers of the *i-th* layer and the material X-axis. (Real or blank)
SOUTi	Stress output request for *i-th* layer,one of the strings **YES** or **NO**. (Default = **NO**)

Remarks:

1. For non-designed elements, the plies are numbered from 1 to n beginning with the bottom layer.

2. For composities there are two methods for specifying the offset of the element reference plane from the element mean plane: **Z0** on this entry and **ZOFF** on the **CQUAD4** or **CTRIA3** Bulk Data entries. The distinction is shown in the figure below:



   You may only specify a **Z0** on this entry if the **ZOFF** field of any **CQUAD4** or **CTRIA3** referencing it is blank. The default value for **Z0** is $-t/2$ where t is the overall thickness of the laminate.

3. **SBOND** is required if bonding material failure index calculations are desired.

4. The failure theory is used to determine the element failure on a ply-by-ply basis. The available theories are:

   HILL - Hill Theory
   HOFF - Hoffman Theory
   TSAI - Tsai-Wu Theory
   STRESS - For Maximum Stress Theory
   STRAIN - For Maximum Strain Theory

5. **MEM** indicates a layup of membrane only plies.

6. The material properties, **MIDi**, may reference only **MAT1**, **MAT2**, and **MAT8** Bulk Data entries.

7. If any of the **MIDi**, **Ti** or **THi** are blank, then the last non-blank values specified for each will be used to define the values for the ply.

8. **TMIN** will be ignored unless the element is linked to design variables by **SHAPE** entries.

Input Data Entry: **PCOMP1** Layered Composite Element Property

Description: Defines the properties of an n-ply laminated composite material where all plies are composed of the same material and are of equal thickness.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PCOMP1	PID	Z0	NSM	SBOND	F.T.	TMIN	MID	LOPT	CONT
CONT	TPLY	TH1	TH2	TH3	TH4	TH5	TH6	TH7	CONT
CONT	TH8	TH9	TH10	-etc-					

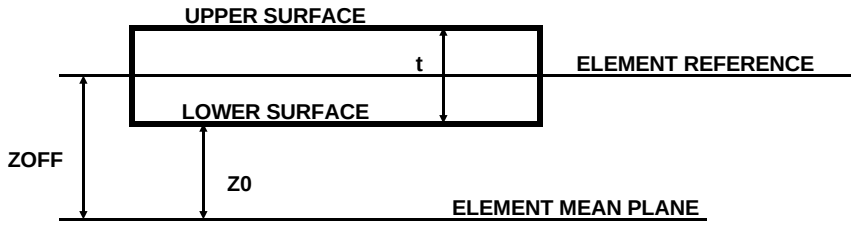
PCOMP1	100	-0.5	1.7	5.+3	STRAIN		200		ABC
+BC	0.25	-45.0	45.0	90.0	90.0	45.0			

Field	Contents
PID	Property identification number (1,000,000 > Integer > 0).
Z0	Offset of the laminate lower surface from the element mean plane. A positive value means the $+z_e$ direction. (Real or blank, see Remark 2)
NSM	Nonstructural mass per unit area (Real ≥ 0.0).
SBOND	Allowable shear stress of the bonding material. (Real ≥ 0.0)
F.T.	Failure theory, one of the strings **HILL**, **HOFF**, **TSAI**, **STRESS** or **STRAIN**. (See Remark 4).
TMIN	Minimum ply thickness for design (Real > 0.0 or blank) (Default = 0.0001)
MID	Material identification number for all layers. (Integer > 0.0 or blank)
LOPT	Lamination generation option, **MEM**.or blank. (See Remark 5).
TPLY	Thickness of each layer. (Real > 0.0).
TH_i	Angle between the longitudinal direction of the fibers of the i(th) layer and the material X-axis. (Real or blank)

Remarks:

1. For nondesigned elements, the plies are numbered from 1 to n beginning with the bottom layer.

2. For composities there are two methods for specifying the offset of the element reference plane from the element mean plane: **Z0** on this entry and **ZOFF** on the **CQUAD4** or **CTRIA3** Bulk Data entries. The distinction is shown in the figure below:



   You may only specify a **Z0** on this entry if the **ZOFF** field of any **CQUAD4** or **CTRIA3** referencing it is blank. The default value for **Z0** is -t/2 where t is the overall thickness of the laminate.

3. **SBOND** is required if bonding material failure index calculations are desired.

4. The failure theory is used to determine the element failure on a ply-by-ply basis. The available theories are:

   HILL - Hill Theory
   HOFF - Hoffman Theory
   TSAI - Tsai-Wu Theory
   STRESS - For Maximum Stress Theory
   STRAIN - For Maximum Strain Theory

5. **MEM** indicates a layup of membrane only plies.

6. The material properties, **MIDi**, may reference only **MAT1**, **MAT2**, and **MAT8** Bulk Data entries.

7. **TMIN** will be ignored unless the element is linked to design variables by **SHAPE** entries.

Input Data Entry: **PCOMP2** Layered Composite Element Property

Description: Defines the properties of an n-ply laminated composite material where all plies are composed of the same material but are of different thickness.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PCOMP2	PID	Z0	NSM	SBOND	F.T.	TMIN	MID	LOPT	CONT
CONT	T1	TH1	T2	TH2	T3	TH3	-etc-		

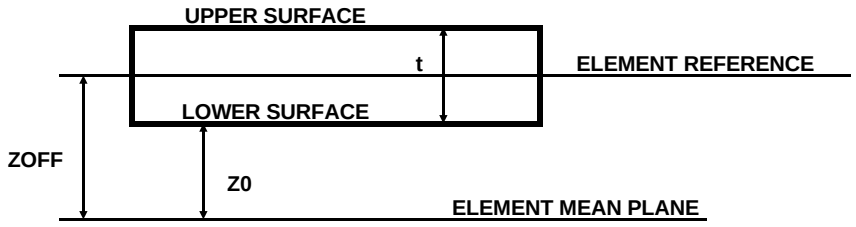
PCOMP2	100	-0.5	1.7	5.+3	TSAI		200		ABC
+BC	0.25	-45.0	0.5	90.0	0.25	45.0			

Field	Contents
PID	Property identification number (Integer > 0).
Z0	Offset of the laminate lower surface from the element mean plane. A positive value means the $+z_e$ direction. (Real or blank, see Remark 2)
NSM	Nonstructural mass per unit area (Real ≥ 0.0).
SBOND	Allowable shear stress of the bonding material. (Real ≥ 0.0)
F.T.	Failure theory, one of the strings **HILL**, **HOFF**, **TSAI**, **STRESS**, or **STRAIN**. See Remark 4.
TMIN	Minimum ply thickness for design (Real > 0.0 or blank) (Default = 10^{-4})
MID	Material identification number for all layers. (Integer > 0.0 or blank)
LOPT	Lamination generation option, **MEM** or blank. (See Remark 5).
T_i	Thickness of the i-th layer. (Real > 0.0 or blank).
TH_i	Angle between the longitudinal direction of the fibers of the i-th layer and the material X-axis. (Real or blank)

Remarks:

1. For nondesigned elements, the plies are numbered from 1 to n beginning with the bottom layer.

2. For composities there are two methods for specifying the offset of the element reference plane from the element mean plane: **Z0** on this entry and **ZOFF** on the **CQUAD4** or **CTRIA3** Bulk Data entries. The distinction is shown in the figure below:



   You may only specify a **Z0** on this entry if the **ZOFF** field of any **CQUAD4** or **CTRIA3** referencing it is blank. The default value for **Z0** is -t/2 where t is the overall thickness of the laminate.

3. **SBOND** is required if bonding material failure index calculations are desired.

4. The failure theory is used to determine the element failure on a ply-by-ply basis. The available theories are:

   HILL - Hill Theory
   HOFF - Hoffman Theory
   TSAI - Tsai-Wu Theory
   STRESS - For Maximum Stress Theory
   STRAIN - For Maximum Strain Theory

5. **MEM** indicates a layup of membrane only plies.

6. The material properties, **MIDi**, may reference only **MAT1**, **MAT2**, and **MAT8** Bulk Data entries.

7. If any of the **Ti** or **THi** are blank, then the last non-blank values specified for each will be used to define the values for the ply.

8. **TMIN** will be ignored unless the element is linked to design variables by **SHAPE** entries.

Input Data Entry: **PELAS** Scalar Elastic Property

Description: Used to define the stiffness, damping coefficient, and stress coefficient of a scalar elastic element (spring) defined by means of the **CELAS1** entry.

Format and Example:

1	2	3	4	5	6	7	8	9	10
PELAS	PID	K	GE	S	TMIN				
PELAS	7	4.29	0.06	7.92					

Field	Contents
PID	Property identification number (Integer > 0)
K	Elastic property value (Real)
GE	Damping coefficient (Real ≥ 0.0)
S	Stress coefficient (Real)
TMIN	Minimum value for design (Real > 0.0, or blank, Default = 0.0001)

Remarks:

1. The user is cautioned to be careful using negative spring values.
2. **TMIN** is ignored unless the element is designed using shape function design variable linking.

Input Data Entry: **PIHEX** Isoparametric Hexahedron Property

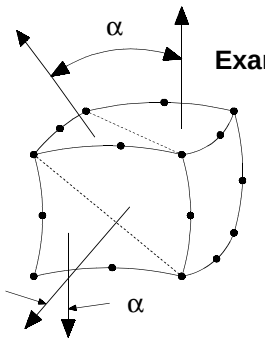
Description: Defines the properties of an isoparametric solid element, including a material reference and the number of integration points. Referenced by the **CIHEX**1, **CIHEX2**, and **CIHEX3** entries.

Format and Examples:

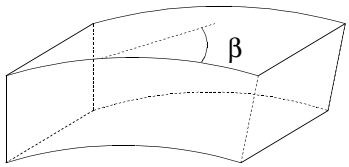
1	2	3	4	5	6	7	8	9	10
PIHEX	PID	MID	CID	NIP	AR	ALPHA	BETA		
PIHEX	15	3		3			5.0		

Field	Contents
PID	Property identification number (Integer > 0)
MID	Material identification number (Integer > 0)
CID	Identification number of the coordinate system in which the material referenced by **MID** is defined (Integer ≥ or blank)
NIP	Number of integration points along each edge of the element (Integer = 2, 3, 4, or blank)
AR	Maximum aspect ratio (ratio of longest to shortest edge) of the element (Real > 1.0 or blank)
ALPHA	Maximum angle in degrees between the normals of two subtriangles comprising a quadrilateral face (Real, 0.0 ≤ **ALPHA** ≤ 180.0 or blank) (Default = 45.0)
BETA	Maximum angle in degrees between the vector connecting a corner point to an adjacent midside point and the vector connecting that midside point and the other midside or corner point (Real, 0.0 < **BETA** < 180.0 or blank) (Default = 45.0)

Examples of Field Definitions:



Example of ALPHA



Example of BETA

Remarks:

1. All **PIHEX** entries must have unique identification numbers.
2. **CID** is not used for isotropic materials.
3. The default for **CID** is the basic coordinate system.
4. The default for **NIP** is 2 for **IHEX** and 3 for **IHEX2** and **IHEX3**.
5. **AR**, **ALPHA**, and **BETA** are used for checking the geometry of the element. The defaults are:

	AR	ALPHA (degrees)	BETA (degrees)
CIHEX1	5.0	45.0	–
CIHEX2	10.0	45.0	45.0
CIHEX3	15.0	45.0	45.0

Input Data Entry: **PLIST**

Description: Defines property entries associated with a design variable.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PLIST	LINKID	PTYPE	PID1	PID2	PID3	PID4	PID5	PID6	CONT
CONT	PID7	PID8	PID9	-etc-					

PLIST	6	PROD	12	14	22				

Alternate Form:

1	2	3	4	5	6	7	8	9	10
PLIST	DVID	PTYPE	PID1	THRU	PID2				

Field	Contents
LINKID	Property list identifier (Integer > 0).
PTYPE	Property type associated with this list (e.g., **PROD**).
PID1,PID2, PID3	Property entry identifications. (Integer > 0, or blank)

Remarks:

1. Allowable **PTYPES** are: **PROD, PSHEAR, PCOMP, PCOMP1, PCOMP2, PELAS, PSHELL , PMASS, PTRMEM, PQDMEM1,** and **PBAR**.
2. If the alternate form is used, **PID2** must be greater than or equal to **PID1**.
3. All elements using properties listed on **PLIST** entries for a particular **LINKID** will be designed by (linked to) that design variable that references the **PLIST LINKID**.

Input Data Entry: **PLISTM**

Description: Defines elements, and their local design variables, associated with a design variable by referencing an element property entry.

Format and Example:

1	2	3	4	5	6	7	8	9	10
PLISTM	LINKID	PTYPE	PID1	DVSYM1	PID2	DVSYM2	PID3	DVSYM3	
CONT	PID4	DVSYM4	-etc-						

PLISTM	6	PBAR1	12	D1	22	D1			

Field	Contents
LINKID	Element list identifier (Integer > 0)
PTYPE	Character input identifying the property type. One of the following: **PELAS PMASS** **PBAR PBAR1 PROD** **PSHEAR PQDMEM1 PTRMEM PSHELL** **PCOMP PCOMP1 PCOMP2**
PIDi	Property identification numbers (Integer > 0, or blank)
DVSYMi	Symbol defining the local design variable. (Remarks 2 and 3)

Remarks:

1. The **LINKID** is referenced by **DESVARP** data to connect the global design variable to the local variables.
2. The following symbols may be used for the different types of properties:

ELEMENTS	ALLOWABLE DVSYM VALUES
PELAS	**K**
PMASS	M
PBAR, PROD	**A**
PBAR1	**D1, D2, D3, D4, D5, D6, D7, D8, D9, D10**
SHEAR,QDMEM1,TRMEM,PSHELL PCOMP,PCOMP1,PCOMP2	**T**

3. If all elements to be linked have only one possible **DVSYM** (e.g. **K**), then the **PLIST** Bulk Data entry may be used.

Input Data Entry: **PLOAD** Static Pressure Load

Description: Defines a static pressure load on a triangular or quadrilateral surface.

Format and Examples:

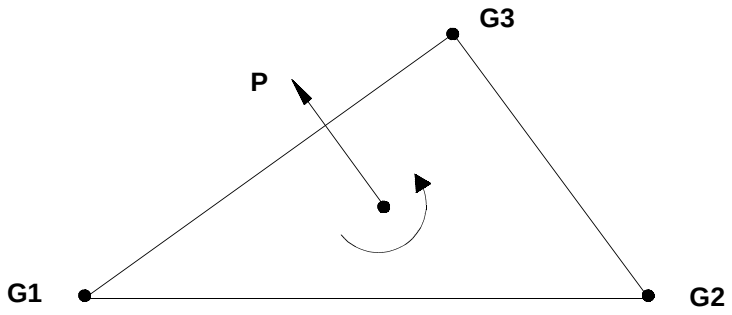
1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			
PLOAD	1	-4.0	16	32	11				

Field	Contents
SID	Load set identification number (Integer > 0)
P	Pressure (Real)
Gi	Grid point identification numbers (Integer > 0; **G4** may be zero)

Remarks:

1. The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If **G4** is zero or blank, the surface is assumed to be triangular.

2. In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points **G1**, **G2**, and **G3** as illustrated below.



   The total load on the surface, AP, is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.

3. In the case of a quadrilateral surface, the grid points **G1**, **G2**, **G3**, and **G4** should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface. The following specific procedures are adopted to accommodate irregular and/or warped surfaces:
   a. The surface is divided into two sets of overlapping triangular surfaces. Each triangular surface is bounded by two of the sides and one of the diagonals of the quadrilateral.
   b. One-half of the pressure is applied to each triangle which is then treated in the manner described in Remark 2.

4. Load sets must be selected in Solution Control to be used.

Input Data Entry **PLOAD2** Plate element static pressure load

Description: Defines a uniform static pressure load applied to plate elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
PLOAD2	LID	P	EID1	EID2	EID3	EID4	EID5	EID6	
PLOAD2	156	98.2	101	432	657				

Alternate Form:

1	2	3	4	5	6	7	8	9	10
PLOAD2	LID	P	EID1	"THRU"	EID2				

Field	Contents
LID	Load set identification number (Integer>0).
P	Pressure value (Real). [1,2]
EIDi	Element identification numbers (Integer>0). (Remark 3)

Remarks:

1. The pressure intensity is the load per unit surface area.
2. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element connection entry. If the surface of an element is curved, the direction of the pressure may vary over the surface. Refer to **PLOAD4** for a more general pressure load capability.
3. For compatibility with commercial NASTRAN products, ASTROS element type identifiers are not used. Therefore, the referenced element identification numbers must be unique among the plate element types.
4. Equivalent grid point loads are computed which depend on the specific element geometry and type. A uniform pressure may not result in equal grid point loads.

Input Data Entry **PLOAD4** Plate element static pressure load

Description: Defines a load on the surface of a TRIA3 or QUAD4 element.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PLOAD4	LID	EID	P1	P2	P3	P4			CONT
CONT	CID	V1	V2	V3					

PLOAD4	101	2043	15.	18.	23.6	12.4			+ABC
+BC	52	1.0	0.0	0.0					

Alternate Form:

1	2	3	4	5	6	7	8	9	10
PLOAD4	LID	EID1	P1	P2	P3	P4	"THRU"	EID2	CONT
CONT	CID	V1	V2	V3					

PLOAD4	1	101	10.	10.	20.	20.	THRU	201	

Field	Contents
LID	Load set identification number (Integer>0).
EID	Element identification number (Integer>0). (Remark 1)
Pi	Pressure at the grid points defining the element surface (Real). (Remarks 2,3,4)
CID	Coordinate system identification number (Integer>0). (Remarks 3,4)
Vi	Components of a vector in system **CID** that defines the direction of the grid point loads generated by the pressure (Real). (Remarks 3,4)

Remarks:

1. For compatibility with commercial NASTRAN products, ASTROS element type identifiers are not used. Therefore, the referenced element identification numbers must be unique among the plate element types.
2. If only **P1** is given, the pressure is assumed to be uniform over the element surface. The **P4** value is ignored for a triangular face. The pressure intensity is the load per unit surface area.
3. If a direction vector is not specified, the direction of the grid point loads is normal to the element mid-surface at each grid point in the local +z direction. If the surface of the element is curved, the direction of pressure may vary from point to point.
4. When the direction vector is defined and a value for **CID** is not entered, the grid point load vectors are applied in the Basic Coordinate System.
5. Equivalent grid point loads are computed which depend on the specific element geometry and type. A uniform pressure may not result in equal grid point loads.

Input Data Entry: **PLYLIST** A list of composite element layer numbers.

Description: Defines a set of layers of composite elements by a list.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PLYLIST	SID	P1	P2	P3	P4	P5	P6	P7	CONT
CONT	P8	-etc-							

PLYLIST	3	1	2	3	4	16	15	14	ABC
+BC	13								

Alternate Form:

1	2	3	4	5	6	7	8	9	10
PLYLIST	SID	P1	THRU	P2					

Field	Contents
SID	Set of identification numbers (Integer > 0)
Pi	List of ply numbers (Integer > 0)

Remarks:

1. These entries are referenced by the **DESVARP**, **DESVARS**, **DCONLMN**, **DCONPMN**, **DCONLAM** and **DCONTH2** data entries.
2. When using the **THRU** option, all intermediate plies will be assumed to exist.
3. When used by **DESVARS** and **DESVARP**, the entry refers to composite layer numbers to be linked together in the design model.
4. When used by **DCONLMN**, **DCONPMN** and **DCONLAM**, the entry refers to composite layers that, together, define a "ply" or a "laminate" whose summed thicknesses will be contribute to the constraint.

Input Data Entry: **PMASS** Scalar Mass Property

Description: Used to define the mass value of a scalar mass element which is defined by means of the **CMASS1** entries.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PMASS	PID	M	TMIN	PID	M	TMIN			
PMASS	7	4.29	0.2	6	13.2	0.1			

Field	Contents
PID	Property identification number (Integer > 0).
M	Value of scalar mass (Real).
TMIN	The minimum mass value in design. Default = 0.0001

Remarks:

1. This entry defines a mass value.
2. Up to 2 mass values may be defined by this entry.
3. **TMIN** is ignored unless the mass element is linked to design variables through **SHAPE** entries.

Input Data Entry: **PQDMEM1** Quadrilateral Membrane Property

Description: Used to define the properties of a quadrilateral membrane referenced by the **CQDMEM1** entry. No bending properties are included.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PQDMEM1	PID	MID	T	NSM	TMIN				
PQDMEM1	235	2	0.5	0.0					

Field	Contents
PID	Property identification number (Integer > 0).
MID	Material identification number (Integer > 0).
T	Thickness of membrane (Real ≥ 0.0)
NSM	Nonstructural mass per unit area (Real ≥ 0.0).
TMIN	Minimum thickness for design (Real > 0.0 or blank) (Default = 0.0001)

Remarks:

1. All **PQDMEM1** entries must have unique property identification numbers.
2. **TMIN** is ignored unless the element is linked to the global design variables by a **SHAPE** entry.

Input Data Entry: **PROD** Rod Property

Description: Defines the properties of a rod which is referenced by the **CROD** entry.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM	TMIN		
PROD	17	23	42.6	17.92	4.236	0.5			

Field	Contents
PID	Property identification number (Integer > 0)
MID	Material identification number (Integer > 0)
A	Area of rod (Real ≥ 0.0, or blank)
J	Torsional constant (Real ≥ 0.0, or blank)
C	Coefficient to determine torsional stress (Real ≥ 0.0, or blank)
NSM	Nonstructural mass per unit length (Real ≥ 0.0, or blank)
TMIN	Minimum rod area for design (Real > 0.0, or blank). Default = 0.0001

Remarks:

1. **PROD** entries must all have unique property identification numbers.
2. For structural problems, **PROD** entries may only reference **MAT1** material entries.
3. The formula used to compute torsional stress is:

   $$\tau = \frac{cM_\theta}{J}$$

   where M_θ is the torsional moment.
4. **TMIN** is ignored unless the rod element is linked to the design variables by **SHAPE** entries.

Input Data Entry: **PSHEAR** Shear Panel Property

Description: Defines the elastic properties of a shear panel. Referenced by the **CSHEAR** entry.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	TMIN				
PSHEAR	13	2	4.9	16.2					

Field	Contents
PID	Property identification number (Integer > 0)
MID	Material identification number (Integer > 0)
T	Thickness of shear panel (Real > 0.0)
NSM	Nonstructural mass per unit area (Real ≥ 0.0, or blank)
TMIN	Minimum panel thickness for design (Real ≥ 0.0, or blank). Default = 0.0001

Remarks:

1. All **PSHEAR** entries must have unique identification numbers.
2. **PSHEAR** entries may reference only **MAT1** material entries.
3. **TMIN** is ignored unless the element is linked to global design variables by **SHAPE** entries.

Input Data Entry: **PSHELL** Shell Element Property

Description: Defines the membrane, bending, transverse shear, and coupling properties of the shell elements. (**QUAD4** and **TRIA3**)

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T3	MID3	TS/T	NSM	CONT
CONT	Z1	Z2	MID4	MCSID	SCSID	ZOFF	TMIN		

PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	ABC
+BC	+.95	-.95		0	0	0.01			

Field	Contents
PID	Property identification number (Integer > 0)
MID1	Material identification number for membrane (Integer > 0 or blank)
T	Default value for membrane thickness (Real > 0.0, or blank)
MID2	Material identification number for bending (Integer > 0, or blank)
12I/T3	Bending stiffness parameter (Real > 0.0, or blank, Default = 1.0)
MID3	Material identification number for transverse shear (Integer > 0, or blank), must be blank unless **MID2** > 0)
TS/T	Transverse shear thickness divided by membrane thickness (Real > 0.0 or blank, Default = .833333).
NSM	Nonstructural mass per unit area (Real > 0.0, or blank)
Z1,Z2	Fiber distances for stress computation. The positive direction is determined by the right-hand rule and the order in which the grid points are listed on the connection entry. (Real or blank, defaults are -1/2 T for Z1 and 1/2 T for Z2.)
MID4	Material identification number for membrane-bending coupling (Integer > 0 or blank, must be blank unless **MID1** > 0 and **MID2** > 0, may not equal **MID1** or **MID2**)
MCSID	Identification number of material coordinate system (Real or blank, or Integer ≥ 0) (See Remark 9)
SCSID	Identification number of stress coordinate system (Real or blank, or Integer ≥ 0) (See Remark 9)
ZOFF	Offset of the element reference plane from the plane of grid points. A positive value means the $+z_e$ direction. (Real or blank, default = 0.0) (See Remark 10)
TMIN	Minimum thickness for design (Real > 0.0 or blank) (Default = 0.0001)

Remarks:

1. All **PSHELL** property entries must have unique identification numbers.
2. The structural mass is computed from the density using the membrane material properties.

3. The results of leaving an **MID** field blank are:
   - **MID1** No membrane or coupling stiffness.
   - **MID2** No bending, coupling, or transverse shear stiffness.
   - **MID3** No transverse shear flexibility.
   - **MID4** No bending-membrane coupling.
4. The continuation entry is not required.
5. The **MID4** field should be left **blank** if the material properties are symmetric with respect to the middle surface of the shell.
6. This entry is used only with the QUAD4 and TRIA3 elements.
7. For structural problems, **PSHELL** entries may reference **MAT1**, **MAT2**, or **MAT8** material property entries.
8. If the transverse shear material, **MID3**, references **MAT2** data, then **G33** must be zero. If **MID3** references **MAT8** data, then **G1**, **Z** and **G2**, **Z** must not be zero.
9. If **MCSID/SCSID** is left blank (0.0) or is real, it is considered to be the angle of rotation of the X axis of the material/stress coordinate system with respect to the X axis of the element coordinate system in the XY plane of the latter. If Integer, the orientation of the material/stress x-axis is along the projection of the x-axis of the specified coordinate system onto the x-y plane of the element system. The value of **MCSID** is the default value for the **TM** field on **CQUAD4** Bulk Data entries.
10. The offset **ZOFF** may also be provided on the **CQUAD4** or **CTRIA3** Bulk Data entry. The element reference plane is located at the mid-thickness of the element parallel to the element mean plane.
11. **TMIN** is ignored unless element is linked to global design variables by **SHAPE** entries.
12. The hierarchy of local coordinate systems is:

    MCSID supplies the default value for the **TM** field on the element connectivity entry

    TM overrides **MCSID if TM** is not blank

    SCSID defaults to the material coordinate system if **SCSID** is blank

Input Data Entry: **PTRMEM**

Description: Defines property data for **TRMEM** element.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
PTRMEM	PID	MID	T	NSM	TMIN				
PTRMEM	500	1000	0.15						

Field	Contents
PID	Property entry identification number (Integer > 0)
MID	Material property identification (Integer > 0)
T	Thickness of membrane element (Real > 0.0)
NSM	Nonstructural mass associated with the element (Real > 0.0, or blank)
TMIN	Minimum thickness for design (Real > 0.0 or blank) (Default = 0.0001)

Remarks:

1. The **PTRMEM** entry can reference either **MAT1**, **MAT2** or **MAT8** entries.
2. **TMIN** is ignored unless the element is linked to global design variables by **SHAPE** entries.

Input Data Entry **RBAR** Rigid Bar

Description: Defines a Rigid Bar element with 6 degrees of freedom at each end.

Format and Example:

1	2	3	4	5	6	7	8	9	10
RBAR	SETID	EID	GA	GB	CNA	CNB	CMA	CMB	
RBAR	1001	5	1	2	234	123			

Field	Contents
SETID	Multipoint constraint set identification number specified in Solution Control. (Integer > 0)
EID	Rigid Bar element identification number. (Integer > 0)
GA,GB	Grid point identification numbers of connection points. (Integer > 0)
CNA, CNB	Independent degrees of freedom in the global coordinate system for the elements at grid point **GA** and **GB**. Indicated by any of the digits 1 through 6 with no embedded blanks. (Integer ≥ 0, or blank) (Remark 2)
CMA, CMB	Component numbers of dependent degrees of freedom in the global coordinate system assigned by the element at grid point **GA** and **GB**. Indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0 or blank) (Remarks 3 and 4)

Remarks:

1. The **RBAR** entry is selected in the Solution Control with the **MPC=SETID** option of the **BOUNDARY** command. ***THIS IS AN ENHANCEMENT TO THE NASTRAN METHOD, WHICH DOES NOT ALLOW RIGID CONNECTIONS TO BE CHANGED FOR DIFFERENT BOUNDARY CONDITIONS.***
2. The total number of components in **CNA** and **CNB** must be six; for example, **CNA=1236**, **CNB=34**. The components must jointly be capable of representing any general rigid body motion of the element.
3. If both **CMA** and **CMB** are zero or blank, all of the degrees of freedom not in **CNA** and **CNB** will be made dependent, i.e. they will be placed in the m-set.
4. The m-set degrees of freedom specified on this entry may not be specified on other entries that define mutually exclusive sets.
5. Rigid element identification numbers must be unique within each element type for each **MPC** set identification number.

Input Data Entry **RBE1** Rigid Body Element, Form 1

Description: Defines a rigid body connected to an arbitrary number of grid points.

Format and Example:

1	2	3	4	5	6	7	8	9	10
RBE1	SETID	EID	GN1	CN1	GN2	CN2	GN3	CN3	CONT
CONT		GN4	CN4	GN5	CN5	GN6	CN6		CONT
CONT	"UM"	GM1	CM1	GM2	CM2	GM3	CM3		CONT
CONT		GM4	CM4	GM5	CM5	-etc-			

RBE1	1001	11	1	2	2	134	3	5	ABC
+BC		4	2						DEF
+EF	UM	1	13	2	1	12	5		

Field	Contents
SETID	Multipoint constraint set identification number specified in Solution Control. (Integer > 0)
EID	Rigid body element identification number. (Integer > 0)
GNi	Grid point identification numbers at which independent degrees of freedom are assigned. (Integer > 0)
CNi	Component numbers of independent degrees of freedom in the global coordinate system at grid points **GNi**, indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0) (Remark 2)
"UM"	Character string indicating the start of the list of dependent degrees of freedom.
GMj	Grid point identification numbers at which dependent degrees of freedom are assigned. (Integer > 0)
CMj	Component numbers of dependent degrees of freedom in the global coordinate system at grid points **GMj**, indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0) (Remark 2)

Remarks:

1. The **RBE1** entry is selected in the Solution Control with the **MPC=SETID** option of the **BOUNDARY** command. ***THIS IS AN ENHANCEMENT TO THE NASTRAN METHOD, WHICH DOES NOT ALLOW RIGID CONNECTIONS TO BE CHANGED FOR DIFFERENT BOUNDARY CONDITIONS.***
2. The total number of components in **CNi** must be six; for example, **CN1=123**, **CN2=3**, **CN3=2** and **CN4=3**. The components must jointly be capable of representing any general rigid body motion of the element. The m-set degrees of freedom specified on this entry may not be specified on other entries that define mutually exclusive sets.
3. A degree-of-freedom cannot be both independent and dependent for the same element. However, both independent and dependent components may exist at the same grid point.
4. Rigid element identification numbers must be unique within each element type for each **MPC** set identification number.

Input Data Entry **RBE2** Rigid Body Element, Form 2

Description: Defines a body whose independent degrees of freedom are specified at a single grid point and whose dependent degrees of freedom are specified at an arbitrary number of grid points.

Format and Example:

1	2	3	4	5	6	7	8	9	10
RBE2	SETID	EID	GN	CM	GM1	GM2	GM3	GM4	CONT
CONT	GM5	GM6	GM7	GM8	-etc-				

RBE2	1001	9	8	12	10	12	14	15	ABC
+BC	16	20							

Field	Contents
SETID	Multipoint constraint set identification number specified in Solution Control. (Integer > 0)
EID	Rigid body element identification number. (Integer > 0)
GN	Grid point identification number at which all 6 independent degrees of freedom are assigned. (Integer > 0)
CM	Component numbers of dependent degrees of freedom in the global coordinate system assigned by the element at grid points **GM1**, **GM2**, etc. Indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0 or blank)
GMi	Grid point identification number at which dependent degrees of freedom are assigned. (Integer > 0)

Remarks:

1. The **RBE2** entry is selected in the Solution Control with the **MPC=SETID** option of the **BOUNDARY** command. ***THIS IS AN ENHANCEMENT TO THE NASTRAN METHOD, WHICH DOES NOT ALLOW RIGID CONNECTIONS TO BE CHANGED FOR DIFFERENT BOUNDARY CONDITIONS.***
2. The components indicated by **CM** are made dependent at all grid points **GMi**.
3. The m-set degrees of freedom specified on this entry may not be specified on other entries that define mutually exclusive sets.
4. Rigid element identification numbers must be unique within each element type for each **MPC** set identification number.

Input Data Entry **RBE3** Rigid Body Element, Form 3

Description: Defines the motion of a reference grid point as the weighted average of motions at a set of other grid points.

Format and Example:

1	2	3	4	5	6	7	8	9	10
RBE3	SETID	EID	REFG	REFC	WT1	C1	G1,1	G1,2	CONT
CONT		G1,3	WT2	C2	G2,1	G2,2	-etc-	WT3	CONT
CONT		C3	G3,1	-etc-	-etc-	WT4	C4	G4,1	CONT
CONT		G4,2	-etc-						CONT
CONT	"UM"	GM1	CM1	GM2	CM2	GM3	CM3		CONT
CONT		GM4	CM4	-etc-					

RBE3	1001	14	100	1234	1.0	123	1	3	ABC
+BC		5	4.7	1	2	4	6	5.2	DEF
+EF		2	7	8	9	5.1	1	15	GHI
+HI		16							JKL
+KL	UM	100	14	5	3	7	2		

Field	Contents
SETID	Multipoint constraint set identification number specified in Solution Control. (Integer > 0)
EID	Rigid body element identification number. (Integer > 0)
REFG	Reference grid point identification number. (Integer > 0)
REFC	Component numbers of degrees of freedom in the global coordinate system that will be computed at **REFG**, Indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0)
WTi	Weighting factor for most common defined by **Gi,j**. (Real)
Ci	Component numbers of degrees of freedom in the global coordinate system which have weighting factor **WTi**, at grid points **Gi,j**. Indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0)
Gi,j	Grid point identification number whose components **Ci** have weighting factor **WTi**. (Integer > 0)
"UM"	Character string indicating the start of the list of dependent degrees of freedom. The default is that all of the components in **REFC** at **REFG**, and no others, will be placed in the m-set.
GMi	Grid point identification numbers with components in the m-set. (Integer > 0)
CMi	Component numbers in the global coordinate system at grid points **GMi** which are placed in the m-set. Indicated by any of the digits 1 through 6 with no embedded blanks. (Integer > 0 or blank) (Remark 2)

Remarks:

1. The **RBE3** entry is selected in the Solution Control with the **MPC=SETID** option of the **BOUNDARY** command. ***THIS IS AN ENHANCEMENT TO THE NASTRAN METHOD, WHICH DOES NOT ALLOW RIGID CONNECTIONS TO BE CHANGED FOR DIFFERENT BOUNDARY CONDITIONS.***
2. The form of **Gi,j** is different than NASTRAN. The first data field on the continuations has been reserved for the **"UM"** identifier. The **Gi,j** list must be contained within data fields 3 through 9. Blanks may appear anywhere in the list.
3. The default for **"UM"** should be used except in cases where the user wishes to include some or all of the **REFC** components in displacement sets other that the m-set. If the default is not used for **"UM"** then:
   the total number of components in **"UM"** must equal the number of components in **REFC**.
   the components in **"UM"** must be a subset of the components specified in the **(REFG,REFC)** and **(Gi,j,Ci)**.
   the m-set coefficient matrix in the constraint equation must be nonsingular.
4. The m-set degrees of freedom specified on this entry may not be specified on other entries that define mutually exclusive sets.
5. Rigid element identification numbers must be unique within each element type for each **MPC** set identification number.

Input Data Entry: **RLOAD1**

Description: Defines a frequency dependent dynamic load of the form.

$$P(f) = A\,[C(f) + iD(f)]\,e^{i(\theta - 2\pi f\tau)}$$

Format and Examples:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	DLAGID	TC	TD					
RLOAD1	10	3	1	2					

Field	Contents
SID	Set identification number (Integer > 0)
DLAGID	Identification number of a **DLAGS** set which defines A, θ and τ (Integer > 0)
TC	Set identification number of **TABLEDi** entry which gives C(f) (Integer ≥ 0; **TC + TD** > 0)
TD	Set identification number of **TABLEDi** entry which gives D(f) (Integer ≥ 0; **TC + TD** > 0)

Remarks:

1. **RLOAD1** loads may be combined with **RLOAD2** loads only by specification on a **DLOAD** entry.
2. **SID** must be unique for all **RLOAD1**, **RLOAD2**, **TLOAD1** and **TLOAD2** entries.

Input Data Entry: **RLOAD2**

Description: Defines a frequency dependent dynamic load of the form.

$$P(f) = AB(f)\, e^{i(\varphi(f) + \theta - 2\pi f\tau)}$$

Format and Examples:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	DLAGID	TB	TP					

RLOAD2	10	6	100	101					

Field	Contents
SID	Set identification number (Integer > 0)
DLAGID	Identification of a **DLAGS** entry which defines A, θ and τ (Integer > 0)
TB	Set identification number of **TABLEDi** entry which gives B(f) (Integer > 0)
TP	Set identification number of **TABLEDi** entry which gives φ(f) in degrees (Integer $\geq$ 0)

Remarks:

1. **RLOAD2** loads may be combined with **RLOAD1** loads only by specification on a **DLOAD** entry. That is, the **SID** on a **RLOAD2** entry may not be the same as that on a **RLOAD1** entry.
2. **SID** must be unique for all **RLOAD1**, **RLOAD2**, **TLOAD1** and **TLOAD2** entries.

Input Data Entry **RROD** Rigid Rod

Description: Defines a pin-ended rod that is rigid in extension.

Format and Example:

1	2	3	4	5	6	7	8	9	10
RROD	SETID	EID	GA	GB	CMA	CMB			
RROD	1001	14	1	2	2				

Field	Contents
SETID	Multipoint constraint set identification number specified in Solution Control. (Integer > 0)
EID	Rigid Rod element identification number. (Integer > 0)
GA, GB	Grid point identification numbers of connection points. (Integer > 0)
CMA, CMB	Component number of one, and only one, dependent degree-of-freedom in the global coordinate system assigned by the element at either grid point **GA** or **GB**. (Integer 1,2 or 3, either **CMA** or **CMB** may contain the digit and the other must be blank)

Remarks:

1. The **RROD** entry is selected in the Solution Control with the **MPC=SETID** option of the **BOUNDARY** command. ***THIS IS AN ENHANCEMENT TO THE NASTRAN METHOD, WHICH DOES NOT ALLOW RIGID CONNECTIONS TO BE CHANGED FOR DIFFERENT BOUNDARY CONDITIONS.***
2. The degree-of-freedom selected to be dependent must have a nonzero component along the axis of the rod; which also implies that the rod must have a finite length.
3. The m-set degrees of freedom specified on this entry may not be specified on other entries that define mutually exclusive sets.
4. Rigid element identification numbers must be unique within each element type for each **MPC** set identification number.

Input Data Entry: **SAVE**

Description: Defines a list of data base entities that are not to be purged.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SAVE	NAME1	NAME2	NAME3	NAME4	NAME5	NAME6	NAME7	NAME8	CONT
CONT	NAME9	NAME10	NAME11	-etc-					

SAVE	DVCT								

Field	Contents
NAMEi	The name of a data base entity whose contents are not to be purged.

Remarks:

1. Any number of continuations are allowed.
2. This data entry is used by the **UTPURG** utility to determine if a requested purge of an entity will take place.

Input Data Entry: **SEQGP** Grid and Scalar Point Resequencing

Description: Used to manually order the grid points and scalar points of the problem. The purpose of this card is to allow the user to reidentify the formation sequence of the grid and scalar points of the structural model in such a way as to optimize bandwidth.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SEQGP	ID	SEQID	ID	SEQID	ID	SEQID	ID	SEQID	CONT
CONT	ID	SEQID	-etc-						

SEQGP	5392	15.6	596	0.2	2	1.9.2.6	3		

Field	Contents
ID	Grid point identification number (Integer > 0)
SEQID	Sequenced identification number (a special number described below)

Remarks:

1. **ID** is any grid or scalar point identification number which is to be reidentified for sequencing purposes. The sequence number identifies a special number which may have any of the following forms where X is a decimal integer digit - **XXXX.X.X.X**, **XXXX.X.X**, **XXXX.X** or **XXXX** where any of the leading Xes may be omitted. This number must contain no embedded blanks. The leading character must not be a decimal point.
2. If the user wishes to insert a point between two already existing grid or scalar points, such as 15 and 16, for example, he would define it as, say 5392, and then use this card to insert extra point number 5392 between them by equivalencing it to, say, 15.6. All output referencing this point will refer to 5392.3. The **SEQID** numbers must be unique and may not be the same as a point **ID** which is not being changed. No extra point **ID** may be referenced more than once.
3. The **SEQID** numbers must be unique and may not be the same as a point **ID** which is not being changed. No extra point **ID** may be referenced more than once.
4. If a point **ID** is referenced more than once, the last reference will determine its sequence.

Input Data Entry: **SET1** Set definition for aerodynamic analysis.

Description: Defines a set of integers by a list.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SET1	SID	G1	G2	G3	G4	G5	G6	G7	CONT
CONT	G8	-etc-							

SET1	3	31	62	93	124	16	17	18	ABC
+BC	19								

Alternate Form:

1	2	3	4	5	6	7	8	9	10
SET1	SID	G1	THRU	G2					

Field	Contents
SID	Set of identification numbers (Integer > 0)
Gi	List of integers (Integer > 0)

Remarks:

1. These entries are referenced by the **SPLINE1** and **FLUTTER** data entries.
2. When using the **THRU** option, all intermediate quantities will be assumed to exist.
3. When used by **SPLINE1**, the entry refers to a list of structural grid points.
4. When used by **FLUTTER**, the entry refers to mode numbers to be omitted in the flutter analysis.

Input Data Entry: **SET2** Grid Point List

Description: Defines a set of structural grid points in terms of aerodynamic macro elements.

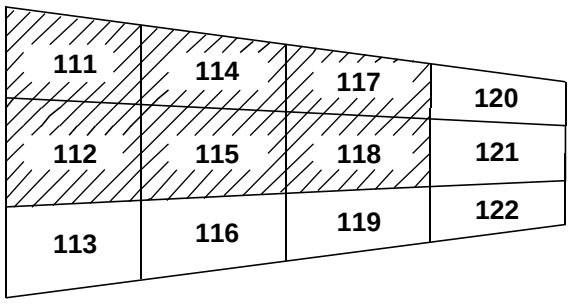
Format and Examples:

1	2	3	4	5	6	7	8	9	10
SET2	SID	SP1	SP2	CH1	CH2	ZMAX	ZMIN		
SET2	3	0.0	0.73	0.0	0.667	1.0	-3.51		

Field	Contents
SID	Set identification number (Integer > 0)
SP1,SP2	Lower and higher span division points defining prism containing set (1.01 > Real > –0.01)
CH1,CH2	Lower and higher chord division points defining prism containing set (1.01 > Real > –0.01)
ZMAX,ZMIN	Z-coordinates of top and bottom (using right-hand rule with the order of the corners as listed on a **CAEROi** entry) of the prism containing set (Real). Usually **ZMAX** > 0.0, **ZMIN** < 0.0

Remarks:

1. These entries are referenced by the **SPLINE1** data entries.
2. Every grid point, within the defined prism and within the height range, will be in the set. For example,



   The shaded area in the figure defines the cross-section of the prism for the sample data given above. Points exactly on the boundary may be missed, hence, to get all the grid points within the area of the macro element, use **SP1** = -0.01, **SP2** = 1.01, etc.

3. A zero value for **ZMAX** or **ZMIN** implies infinity is to be used.

Input Data Entry: **SHAPE**

Description: Defines element connectivity entries associated with a design variable.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SHAPE	SHAPEID	ETYPE	EID1	PREF1	EID2	PREF2	EID3	PREF3	CONT
CONT	EID4	PREF4	EID5	PREF5	-etc-				

SHAPE	10	CROD	12	12.0	22	1.0			

Field	Contents
SID	Shape function identification number (Integer > 0)
ETYPE	Character input identifying the element type. One of the following: **CELASi CMASSi CONM2** **CBAR CROD CONROD** **CSHEAR CQDMEM1 CTRMEM CQUAD4 CTRIA3**
EIDi	Element identification numbers (Integer > 0, or blank)
PREFi	Linking factor for the associated **EID** (Real)

Remarks:

1. The shape function identification is referenced by the **DESVARS** entry to connect the global variable to the shape.
2. The linking factors define a shape function to be used as the global design variable.
3. Designed properties (e.g., thicknesses) of elements listed on **SHAPE** entries will be set to unity to ensure proper shape function definition; that is, the **PREF** values define the shape to be applied to a uniform property distribution.
4. If **PBAR1** cross-sectional parameters are used as design variables, the **SHAPEM** Bulk Data entry must be used.

Input Data Entry: **SHAPEM**

Description: Defines element connectivity entries, and their local variables, associated with a design variable.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SHAPEM	SHAPEID	ETYPE	EID1	DVSYM1	PREF1	EID2	DVSYM2	PREF2	CONT
CONT	EID3	DVSYM3	PREF3	-etc-					

SHAPEM	10	CROD	12	A	1.0	22	A	0.5	

Field	Contents
SHAPEID	Shape function identification (Integer > 0)
ETYPE	Character input identifying the element type. One of the following: **CELASi CMASSi CONM2** **CBAR CROD CONROD** **CSHEAR CQDMEM1 CTRMEM CQUAD4 CTRIA3**
EIDi	Element identification numbers (Integer > 0, or blank)
DVSYMi	Symbol defining the local design variable. (Remarks 2 and 3)
PREFi	Linking factor for the associated local design variable (Real)

Remarks:

1. The shape function identification number is referenced by the **DESVARS** entry to connect the global variable to the shape.
2. The following symbols may be used for the different types of properties:

ELEMENTS	ALLOWABLE DVSYM VALUES
PELAS	**K**
PMASS	M
PBAR, PROD	**A**
PBAR1	**D1, D2, D3, D4, D5, D6, D7, D8, D9, D10**
SHEAR,QDMEM1,TRMEM,PSHELL PCOMP,PCOMP1,PCOMP2	**T**

3. If all elements to be linked have only one possible **DVSYM** (e.g. **K**), then the **SHAPE** Bulk Data entry may be used.

Input Data Entry: **SHPGEN**

Description: Defines a design variable which performs shape linking using the automatic shape generation capability.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SHPGEN	SHAPEID	ESETID	SHAPE	X0,CID	Y0	Z0	DVSYMBL		
SHPGEN	10	11	12	12.0	22.0	1.0			

Field	Contents
SHAPEID	Shape function identification (Integer > 0)
ESETID	Identification number of an **ELEMLIST** Bulk Data entry (Integer > 0)
SHAPE	Desired shape function (Character) (Remark 1)
CID	The identification number of a user-defined coordinate system in which the origin is the new origin for shape generation, and the shape function contribution is in the direction of these coordinate axes.
X0	The x-coordinate, in the basic system, of the new origin for shape generation (Real)
Y0	The y-coordinate, in the basic system, of the new origin for shape generation (Real)
Z0	The z-coordinate, in the basic system, of the new origin for shape generation (Real)
DVSYMBL	Character symbol specifying the **PBAR1** cross-sectional parameter if **ETYPE** is **PBAR**. **D1** **D2** **D3** **D4** **D5** **D6** **D7** **D8** **D9** **D10**

Remarks:

1. **SHAPEID** is referenced by a **DESVARS** Bulk Data entry which defines the shape used for the global variable.
2. To print or punch the resulting **SHAPE** or **SHAPEM** entries, you may use the **DEBUG** command **SHPGEN**.
3. The **SHAPE** is a character string that consists of one, two or three digits. The first digit specifies the order of the contribution to the shape of the basic x-coordinate of the element centroid. The second and third digits represent the same data for the y-coordinate and z-coordinate of the centroid, respectively. The value of each digit may vary from 0 to 9, which represents the order of the shape term, as:

   232 indicates to use the terms $\left(x - x_o \right)^2$; $\left(y - y_o \right)^3$; $\left(z - z_o \right)^2$

4. The shape function contributions are about the specified point in the basic coordinate system unless you specify a **CID**. Then the contributions are relative to this system.

Input Data Entry: **SPC** Single-Point Constraint

Description: Defines sets of single-point constraints and enforced displacements.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G	C	D	G	C	D		
SPC	2	32	436	-2.6	5		+2.9		

Field	Contents
SID	Identification number of single-point constraint set (Integer > 0)
G	Grid or scalar point identification number (Integer > 0)
C	Component number of global coordinate (6 ≥ Integer ≥ 0; up to 6 unique digits may be placed in the field with no embedded blanks.)
D	Value of enforced displacement for all coordinates designed by **G** and **C** (Real)

Remarks:

1. Degrees of freedom specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets.
2. Single-point forces of constraint are recovered during stress data recovery.
3. Single-point constraint sets must be selected in Solution Control (**SPC= SID**) to be used.
4. **SPC** degrees of freedom may be redundantly specified as permanent constraints on the **GRID** entry.

Input Data Entry: **SPCADD** Single-Point Constraint Set Combination

Description: Defines a single-point constraint set as a union of single-point constraint sets defined via **SPC** or **SPC1** entries.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	CONT
CONT	S8	S9	-etc-						

SPCADD	101	3	2	9	1				

Field	Contents
SID	Identification number for new single-point constraint set (Integer > 0)
Si	Identification numbers of single-point constraint sets defined via **SPC** or by **SPC1** entries (Integer > 0; **SID** ≠ **Si**)

Remarks:

1. Single-point constraint sets must be selected in Solution Control (**SPC = SID**) to be used.
2. No **Si** may be the identification number of a single-point constraint set defined by another **SPCADD** entry.
3. The **Si** values must be unique.
4. **SPCADD** entries take precedence over **SPC** or **SPC1** entries. If both have the same set **ID**, only the **SPCADD** entry will be used.

Input Data Entry: **SPC1** Single-Point Constraint, Alternate Form 1

Description: Defines sets of single-point constraints

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	CONT
CONT	G7	G8	G9	-etc-					

SPC1	3	2	1	3	10	9	6	5	ABC
+BC	2	8							

Alternate Form:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	GID1	THRU	GID2				

SPC1	313	456	10	THRU	1000				

Field	Contents
SID	Identification number of single-point constraint set (Integer > 0)
C	Component number of global coordinate (any unique combination of the digits 1 through 6 (with no embedded blanks) when point identification numbers are grid points; must be null if point identification numbers are scalar points)
Gi, GIDi	Grid or scalar point identification numbers (Integer > 0)

Remarks:

1. Note that enforced displacements are not available via this entry. As many continuation entries as desired may appear.
2. Coordinates specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets.
3. Single-point constraint sets must be selected in Solution Control (**SPC = SID**) to be used.
4. **SPC** degrees of freedom may be redundantly specified as permanent constraints on the **GRID** entry.
5. If the alternate form is used, points in the sequence **GID1** through **GID2** are required to exist.

Input Data Entry: **SPLINE1** Surface Spline

Description: Defines a surface spline for interpolating out-of-plane motion for aeroelastic problems.

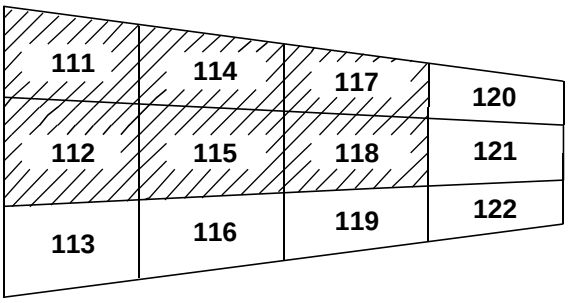
Format and Examples:

1	2	3	4	5	6	7	8	9	10
SPLINE1	EID	CP	MACROID	BOX1	BOX2	SETG	DZ		
SPLINE1	3		111	111	118	14	0.0		

Field	Contents
EID	Element identification number (Integer > 0)
CP	Coordinate system defining the spline plane (Integer ≥ 0, or blank)
MACROID	Identification number of a **CAEROi** entry which defines plane of spline (Integer > 0)
BOX1,BOX2	First and last box whose motions are interpolated using this spline (Integer > 0)
SETG	Refers to a **SETi** entry which lists the structural grid points to which the spline is attached (Integer > 0)
DZ	Linear attachment flexibility (Real ≥ 0.0)

Remarks:

1. The interpolated points (k-set) will be defined by aero-cells. The sketch shows the cells for which u_k is interpolated if **BOX1** = 111 and **BOX2** = 118.



2. The attachment flexibility (units of area) is used for smoothing the interpolation. If **DZ** = 0.0, the spline will pass through all deflected grid points. If **DZ** >> (area of spline), a least squares plane fit will occur. Intermediate values will provide smoothing.
3. If no **CP** is specified, the spline plane is assumed to be the **CAERO** macro element plane.
4. The **SPLINE EID** is used only for error messages and need not be related to the macroelement identification number.

Input Data Entry: **SPLINE2**

Description: Defines a beam spline for interpolating panels and bodies for steady and unsteady aeroelastic analyses.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SPLINE2	EID	MACROID	BOX1	BOX2	SETG	DZ	DTOR	CID	CONT
CONT	DTHX	DTHY							

SPLINE2	1000	5000	5000	5100	10	0.	1.0	4	+ABC
+BC	-1.								

Field	Contents
EID	Element identification number (Integer > 0)
MACROID	The identification of a **CAERO1**, **CAERO2**, **CAERO6** or **PAERO6** aerodynamic macroelement to be splined (Integer > 0)
BOX1,BOX2	The identification numbers of the first and last boxes on the macroelement to be interpolated using this spline (Integer > 0)
SETG	The identification of a **SETi** entry which lists the structural grid points to which the spline is attached (Integer > 0)
DZ	Linear attachment flexibility (Real ≥ 0.0)
DTOR	Torsional flexibility, $\frac{EI}{GJ}$ (Real ≥ 0.0; use 1.0 for bodies)
CID	Rectangular coordinate system which defines the y-axis of the spline (Integer > 0 if lifting surface or blank; not used for bodies)
DTHX,DTHY	Rotational attachment flexibility. **DTHX** is for rotation about the x-axis; not used for bodies. **DTHY** is for rotation about the y-axis; used for slope of bodies. (Real)

Remarks:

1. The interpolation points (k-set) will be defined by aero-cells.
2. For panels, the spline axis is the projection of the y-axis of coordinate system **CID**, projected onto the plane of the panel. For bodies, the spline axis is parallel to the x-axis of the aerodynamic coordinate system.
3. The flexibilities are used for smoothing. Zero attachment flexibilities will imply rigid attachment, i.e., no smoothing. Negative values of **DTHX** and /or **DTHY** will imply no attachment.
4. The continuation card is optional.
5. The **SPLINE2 EID** must be unique with respect to all other **SPLINE**i data entries, it is used only for error messages.

Input Data Entry: **SPOINT** Scalar Point List

Description: Defines scalar points of the structural model

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	
SPOINT	3	18	1	4	16	2			

Alternate Form:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	"THRU"	ID2						

Field	Contents
IDi, ID1, ID2	Scalar point identification number (Integer > 0; **ID1** < **ID2**)

Remarks:

1. If the alternate form is used, all scalar points **ID1** through **ID2** are defined.

Input Data Entry: **SUPORT** Fictitious Support

Description: Defines coordinates at which the user desires determinate reactions to be applied to a free body during analysis.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
SUPORT	SETID	ID	C	ID	C	ID	C		
SUPORT	1000	16	215						

Field	Contents
SETID	Solution control **SUPPORT** set identification (Integer > 0)
ID	Grid or scalar point identification number (Integer > 0)
C	Component number (zero or blank for scalar points; any unique combination of the digits 1 through 6 for grid points)

Remarks:

1. Coordinates specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets.
2. From one to three support coordinates may be defined on a single entry.
3. Continuation entries are not allowed.

Input Data Entry: **TABDMP1** Modal Damping Table

Description: Defines modal damping as a tabular function of frequency.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TABDMP1	ID	TYPE	F_1	G_1	F_2	G_2	F_3	G_3	CONT
CONT	F_4	G_4	F_5	G_5	F_6	G_6	-etc-		

TABDMP1	3	G	0.0	0.005	1.0	0.008	2.0	0.001	ABC
+BC	2.5	0.01057	2.6	0.01362					

Field	Contents
ID	Table identification number (Integer > 0)
TYPE	Data word which indicates the type of damping units, **G**, **CRIT**, **Q**, or blank. Default is **G**.
F_i	Frequency value in cycles per unit time (Real ≥ 0.0).
G_i	Damping value (Real).

Remarks:

1. The **Fi** must be in either ascending or descending order but not both.
2. Jumps between two points (**Fi = Fi+1**) are allowed, but not at the end points.
3. At least two entries must be present.
4. Any **Fi, Gi** entry may be ignored by placing the **BCD** string **SKIP** in either of two fields used for that entry.
5. The **TABDMP1** mnemonic infers the use of the algorithm

   $g = g_t(F)$

   where F is input to the table and g is returned. The table look-up $g_T(F)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. At jump points the average $g_T(F)$ is used. There are no error returns from this table look-up procedure.
6. If **TYPE** is **G** or blank, the damping values are in structural damping units, that is, the value of g in (1+ig)K. If **TYPE** is **CRIT**, the damping values are in the units of fraction of critical damping, C/C0. If **TYPE** is **Q**, the damping values are in the units of the amplification or quality factor, **Q**. These constants are related by the following equations:

   C/C0 = g/2,

   $$Q = \left\{ \begin{array}{c} \dfrac{1}{\left(\dfrac{2C}{C_0}\right)} \\ \\ \dfrac{1}{g} \end{array} \right\}$$

Input Data Entry: **TABLED1**

Description: Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TABLED1	ID								CONT
CONT	x_1	y_1	x_2	y_2	x_3	y_3	-etc-		

TABDMP1	32								ABC
+BC	-3.0	6.9	2.0	5.6	3.0	5.6			

Field	Contents
ID	Table identification number (Integer > 0)
x_i, y_i	Tabular entries (Real)

Remarks:

1. The **xi** must be in either ascending or descending order but not both.
2. Jumps between two points (**xi = xi+1**) are allowed, but not at the end points.
3. At least two entries must be present.
4. Any x-y entry may be ignored by placing the string **SKIP** in either of the two fields used for that entry.
5. The generated function is:

   $y = y_T(X)$

   where X is input to the table and Y is returned. The table look-up $y_T(x)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. At jump points the average $y_T(x)$ is used. There are no error returns from this table look-up procedure.

Input Data Entry: **TEMP** Grid Point Temperature Field

Description: Defines temperature at grid points for determination of (1) Thermal Loading; and (2) data recovery.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G	T	G	T	G	T		
TEMP	3	94	316.2	49	219.8				

Field	Contents
SID	Temperature set identification number (Integer > 0)
G	Grid point identification number (Integer > 0)
T	Temperature (Real)

Remarks:

1. From one to three grid point temperatures may be defined on a single entry.
2. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined.
3. For each thermal load, temperatures must be specified for all grid points using either **TEMP** or **TEMPD** entries.

Input Data Entry: **TEMPD** Grid Point Temperature Field Default

Description: Defines a temperature value for all grid points of the structural model which have not been given a temperature on a **TEMP** entry.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID	T	SID	T	SID	T	SID	T	

TEMPD	1	215.3							

Field	Contents
SID	Temperature set identification number (Integer > 0)
T	Default temperature value (Real)

Remarks:

1. From one to four default temperatures may be defined on a single entry.
2. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined.
3. For each thermal load, temperatures must be specified for all grid points using either **TEMP** or **TEMPD** entries.

Input Data Entry: **TF** Dynamic Transfer Function

Description:

1. Used to define a transfer function of the form

$$(\mathbf{B0} + \mathbf{B1}\ p + \mathbf{B2}\ p^2)\ u_d + \sum_i (\mathbf{A0}(i) + \mathbf{A1}\ (i)\ p + \mathbf{A2}\ (i)\ p^2)\ u_i = 0$$

2. May also be used as a means of direct matrix input. See Remark 3.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TF	SID	GD	CD	B0	B1	B2			CONT
CONT	G(1)	C(1)	A0(1)	A1(1)	A2(1)				

TF	1	2	3	4.0	5.0	6.0			+ABC
+ABC	13	4	5.0	6.0	7.0				

Field	Contents
SID	Set identification (Integer > 0).
GD,G(i)	Grid, scalar or extra point identification numbers (Integer > 0).
CD,C(i)	Component numbers (null or zero for scalar or extra points, any **one** of the digits 1 through 6 for a grid point).
B0, B1, B2, A0(i),A1(i), A2(i)	Transfer function coefficients (Real).

Remarks:

1. The matrix elements defined by this entry are added to the dynamic matrices for the problem.
2. Transfer function sets must be selected in Solution Control (**TFL = SID**) to be used.
3. The constraint relation given in Equation 1 will hold only if no structural elements or other matrix elements are connected to the dependent coordinate, u_d. In fact, the terms on the left side of Equation 1 are simply added to the terms from all other sources in the row for u_d.
4. Any number of continuations are allowed.

Input Data Entry: **TIMELIST**

Description: Defines a list of times at which outputs are desired.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TIMELIST	SID	TIME	TIME	TIME	TIME	TIME	TIME	TIME	CONT
CONT	TIME	TIME	-etc-						

TIMELIST	100	0.1	0.2	0.5	1.0				

Field	Contents
SID	Set identification number referenced by Solution Control (Integer > 0)
TIME	Time, (in consistent time unit) at which outputs are desired. (Real)

Remarks:

1. In order to be used, the **SID** must be referenced by Solution Control.
2. The nearest time to **TIME**, either above or below, which was used in the Transient Response analysis will be used to satisfy the output requests.
3. Any number of continuations is allowed.

Input Data Entry: **TLOAD1**

Description: Defines a time dependent function of the form:

$$\boldsymbol{P}(t) = A\boldsymbol{F}(t - \tau)$$

for use in a transient response problem.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	DLAGID	TID						

TLOAD1	10	8	13						

Field	Contents
SID	Set identification number (Integer > 0)
DLAGID	Identification number of **DLAGS** set which defines A and τ (Integer > 0)
TID	Identification number of a **TABLED1** entry which gives $F(t-\tau)$ (Integer > 0)

Remarks:

1. **SID** must be unique for all **TLOAD1**, **TLOAD2**, **RLOAD1**, and **RLOAD2** entries.

Input Data Entry: **TLOAD2**

Description: Defines a time-dependent dynamic load of the form:

$$P(t) = \begin{cases} 0 & \text{when } \bar{t} < 0 \text{ or } \bar{t} > \mathbf{t2} - \mathbf{t1} \\ A\,\bar{t}^{\,\mathbf{B}}\, e^{\,\mathbf{C}\,\bar{t}} \cos(2\pi\,\mathbf{f}\,\bar{t} + \theta) & \text{when } 0 \le \bar{t} \le \mathbf{t2} - \mathbf{t1} \end{cases}$$

$$\text{where } \bar{t} = t - \mathbf{t1} - \tau$$

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	DLAGID	T1	T2	FREQ	PHASE	CTEXP	GROWTH	
TLOAD2	10	6	2.1	4.7	12.0	30.0	2.0	3.0	

Field	Contents
SID	Set identification number (Integer > 0)
DLAGID	Identification number of the **DLAGS** entry set which define the time invariant load A and the time delay (Integer >0)
T1	Time constant (Real ≥ 0.0)
T2	Time constant (Real, **T2 > T1**)
FREQ	Frequency in cycles per unit time (Real ≥ 0.0)
PHASE	Phase angle in degrees (Real)
CTEXP	Exponential coefficient (Real)
GROWTH	Growth coefficient (Real)

Remarks:

1. **TLOAD**2 loads may be combined with **TLOAD1** loads only by specification on a **DLOAD** entry.
2. **SID** must be unique for all **TLOAD1**, **TLOAD2**, **RLOAD1** and **RLOAD2** entries.

Input Data Entry **TRIM** Trim Variable Specification

Description: Specifies conditions for steady aeroelastic trim or nonplanar steady aerodynamic analysis.

Format and Example:

1	2	3	4	5	6	7	8	9	10
TRIM	TRIMID	MACH	QDP	TRMTYP	EFFID	VO			CONT
CONT	LABEL1	VAL1	LABEL2	VAL2	LABEL3	VAL3	LABEL4	VAL4	-etc-

TRIM	1001	0.90	1200.	LIFT	100	926.3			+ABC
+ABC	NZ	8.0	QRATE	0.243	ELEV	FREE	ALPHA	FREE	

Field	Contents
TRIMID	Trim set identification number (Integer>0)
MACH	Mach number (Real ≥ 0.0)
QDP	Dynamic pressure (Real>0.0)
TRMTYP	Type of trim required (Character or blank) (See Remark 3) blank **SUPORT** controlled trim. **ROLL** Axisymmetric roll trim (1 DOF) **LIFT** Symmetric trim of lift forces (1 DOF) **PITCH** Symmetric trim of lift and pitching moment (2 DOF)
EFFID	Identification number of **CONEFFS** Bulk Data entries which modify control surface effectiveness values (Integer ≥ 0, or blank)(Remark 2)
VO	True velocity (Real>0.0, or blank) (See Remark 12)
LABELi	Label defining aerodynamic trim parameters.
VALi	Magnitude of the specified trim parameter (Real) or the character string **FREE**.

Remarks:

1. The **TRIM** entry is selected in Solution Control in the **SAERO** and **NPSAERO** disciplines with the **TRIM** option.
2. All aerodynamic forces created by the control surface will be reduced to the referenced amount. For example, an **EFF1** of 0.70 indicates a 30% reduction in the forces.
3. The **TRMTYP** field has the following interpretation:

   LIFT Implies that the vertical acceleration will be trimmed by one **FREE** symmetric control parameter or surface, or, the acceleration computed for some set of symmetric parameters/surfaces.

ROLL	implies that the roll acceleration, **PACCEL**, will be trimmed by some one **FREE** antisymmetric control parameter or surface — OR — the acceleration computed for some set of antisymmetric parameters/surfaces. Any number of antisymmetric parameters may be fixed, but the **FREE** parameters are limited to **PACCEL** — OR — any one antisymmetric parameter or surface. For example, **PACCEL=0.0; AILERON=1.0; PRATE=FREE**
PITCH	implies that the vertical acceleration, **NZ**, and the pitch acceleration, **QACCEL**, will be trimmed by no more than two **FREE** symmetric control parameters or surfaces — **OR** — the accelerations computed for some set of symmetric parameters/surfaces. Any number of symmetric parameters may be fixed, but the **FREE** parameters are limited to **QACCEL** and **NZ** — OR — up to two symmetric parameters or surfaces — **OR** — some combination. For example, **NZ=8.0g's; QACCEL=0.0; ALPHA=FREE; ELEV=FREE**
blank	implies that the support DOFs are equal to the number of free parameters. Appropriate trim equations are assembled and solved.

4. Units for **QDP** are force per unit area.

5. Allowable options for **LABEL**i for symmetric trim (the symmetry option is selected in Solution Control) are:

Structural Accelerations

NX	Longitudinal acceleration (Remark 10)
NZ	Vertical acceleration (load factor) (Remark 10)
QACCEL	Pitch acceleration (Remark 11)

Aerodynamic Parameters

ALPHA	Angle of attack in degrees
QRATE	Pitch rate (Remark 12)
THKCAM	Thickness and camber (Remark 13)
and	Control surfaces in degrees

6. Allowable options for **LABELi** for antisymmetric trim (the symmetry option is selected in Solution Control) are:

Structural Accelerations

NY	Side-slip acceleration (Remark 10)
PACCEL	Roll acceleration (Remark 11)
RACCEL	Yaw acceleration (Remark 11)

Aerodynamic Parameters

BETA	Yaw angle in degrees (Remark 14)
PRATE	Roll rate (Remark 12)
RRATE	Yaw rate (Remark 14)
and	Antisymmetric control surfaces in degrees

7. If **VALUEi** is a real number, the associated aerodynamic parameter or structural acceleration is set to that value. If **VALUEi** is the character string **FREE**, then the associated parameter will be determined as part of the trim analysis.

8. The number of **FREE** Values of **VALUEi** must correspond exactly to the number of unknowns in the trim analysis. If **TRMTYP** is blank, the number of **SUPORT** DOF.
9. If **TRIMID** is referenced by an **NPSAERO** discipline, **TRMTYP** must be blank and **FREE** is not allowed for **VALUEi**.
10. For **NX**, **NY** and **NZ**, units are length per second in consistent units unless a **CONVERT/MASS** Bulk Data entry is provided. In this case, the values are dimensionless.
11. The angular accelerations, **QACCEL**, **PACCEL**, and **RACCEL**, are entered in units of radians per second per second.
12. **QRATE**, **PRATE**, and **RRATE**, are entered in units of radians per second. The velocity must be input if any of the "rate" parameters are given since its value is needed to dimensionalize the forces computed for a unit rate per velocity in the aerodynamic preface.
13. The **THKCAM** label refers to thickness and camber effects and its corresponding value is usually set to 1.0. Non-unit values of the **THKCAM** parameter are available only to provide added generality.
14. Any control surfaces, trim parameters, or structural accelerations not specified on the **TRIM** entry will not participate in the analysis: they will be given fixed values of 0.0. This includes **THKCAM**.
15. Refer to the **STATIC AEROELASTIC TRIM** Application Note for more information.

Input Data Entry: **TSTEP**

Description: Defines time step intervals at which a solution will be generated and output in transient analysis.

Format and Examples:

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N(1)	DT(1)	N0(1)					CONT
CONT		N(2)	DT(2)	N0(2)					

TSTEP	2	10	.001	5					+ABC
+ABC		9	0.01	1					

Field	Contents
SID	Set identification number (Integer > 0)
N(i)	Number of time steps of value **DT(i)** (Integer ≥ 2)
DI(i)	Time increment (Real > 0.0)
N0(i)	Skip factor for output (every **N0(i)th** step will be saved for output) (Integer > 0)

Remarks:

1. **TSTEP** entries must be selected in the Solution Control (**TSTEP=SID**).
2. Note that the entry permits changes in the size of the time step during the course of the solution. Thus, in the example shown, there are 10 time steps of value 0.001 followed by 9 time steps of value .01. Also, the user has requested that output be recorded for t = 0.0, 0.005, 0.01, 0.02, 0.03, etc.

Input Data Entry: **VELOLIST**

Description: Defines a list of velocity values.

Format and Example:

1	2	3	4	5	6	7	8	9	10
VELOLIST	SID	VELO1	VELO2	VELO3	VELO4	VELO5	VELO6	VELO7	CONT
CONT	VELO8	VELO9	-etc-						

VELOLIST	201	100.0	80.0	200.0					

Field	Contents
SID	Velocity set identification number (Integer > 0)
VELOi	Velocity value (Real > 0.0)

Remarks:

1. **VELOLIST** Bulk Data entries are selected in the Function Packet.

Input Data Entry: **VSDAMP**

Description: Specifies values of g and/or ω3 to generate either viscous damping that has the same damping forces as structural damping of magnitude g at the frequency ω3 or to specify the structural damping g (see Remarks 3 and 4)

Format and Examples:

1	2	3	4	5	6	7	8	9	10
VSDAMP	SID	G	ω3	SID	G	ω3			

VSDAMP	100	0.005	15.0						

Field	Contents
SID	Set identification number (Integer > 0)
G	Damping value (Real)
ω3	Frequency value in Hertz (Real ≥ 0.0)

Remarks:

1. The setid is selected by the **DAMPING**=n command in Solution Control.
2. Up to two values of g and ω3 can be defined on a single entry.
3. If ω3 is zero, g will be used to generate a complex stiffness matrix 3 of the form $\boldsymbol{K} = (1 + ig)\boldsymbol{K}$
4. If ω3 is nonzero, a viscous damping matrix of the form $\boldsymbol{B} = \left[\frac{g}{2\pi\omega_3}\right]\boldsymbol{K}$ is generated.

This page is intentionally blank.

Chapter 8.

OUTPUT FEATURES

In a software system the magnitude of ASTROS, the amount of data that may be of interest to you is very large. In multidisciplinary optimization, the quantity of data is even larger and the expense involved in its computation even more critical. It is worthwhile, therefore, to limit the amount of output to a minimum and to provide a mechanism for you to select those data that are of importance in each particular case. Chapter 4 of this manual described one mechanism provided to select particular iterations, disciplines, subcases and response quantities: that of the Solution Control output request. This Chapter endeavors to present the totality of output options available. The system controlled outputs from the engineering modules are described in order to establish a familiarity with an ASTROS output listing. This is followed by a more complete description of output from each Solution Control request than is contained in Chapter 4, with different disciplines, elements, design constraints and node types accounted for in some detail. These represent the outputs that are fully supported by the ASTROS software and require little or no user intervention to obtain. The presentation of these features assumes that the standard executive sequence is used. If the user substantially modifies the standard sequence (to the point where certain modules are not called), some or all of the presented output features may no longer be available.

The more advanced forms of user output requests are also presented in this section. The most basic of these forms involve changing the engineering module print control levels through the use of the DEBUG packet. Then, the MAPOL addressable print utilities are presented. The use of these utilities, in conjunction with the general versatility of the MAPOL language, provides the user with the capacity both to look at existing data and to compute and view additional data. In fact, these options enable the user to obtain virtually any data that reside on the data base or that can be computed and stored on the data base. Finally, a quick overview of the Interactive eBASE Environment (eSHELL) is given. The eSHELL program provides for complete Standard Query Language interactive queries on the eBASE entities.

8.1. SYSTEM CONTROLLED OUTPUT

Many of the engineering and executive system program units write data to the ASTROS output listing automatically. As enumerated in the introduction to this section, output of this nature in ASTROS is very limited, but sufficient amounts exist to justify a brief presentation of the data and their formats. It is also useful to present the basic ASTROS listing in order to facilitate contrasting it to listings containing user selected output quantities. The first page of ASTROS output is the title page showing the version number, date and host machine. Each page of output following the solution control listing is labeled with six lines of header information including the user selected title, subtitle and label. The version number, date and, if applicable, the design iteration number will also appear in the header of each page.

8.1.1. Default Output Printed by Modules

The DEBUG packet echo and the **ASSIGN DATABASE** entries, shown in Table 8-1, are the first output following the title page. Immediately following these, the solution control commands are echoed to the output listing. This listing is helpful in identifying the particular disciplines and cases selected in the run. The multidisciplinary nature of ASTROS requires further output labeling. Therefore, in addition to the solution control summary, the **BOUND** module writes a summary of selected disciplines for each boundary condition at the top of the boundary condition loop, as shown in Table 8-2. It indicates all disciplines and most discipline options in the current boundary condition to assist you in determining the particular path that will be taken through the standard MAPOL sequence. A similar printout, Table 8-3, from the **ABOUND** module appears at the top of the sensitivity phase boundary condition loop to indicate the nature of the active boundary conditions and active design constraints.

The next set of output, Table 8-4, comes from the bandwidth minimizer. It details the method selected, numbers of grids and elements in the model and the values of the measures of merit in the resequencing of grid points.

Active constraint information is provided in the Active Constraint Summary from the **ACTCON** module. It indicates the total number of constraints considered active according to the current constraint deletion criteria. You may select a complete listing of the active constraints with the **PRINT DCONSTRAINT** solution control option, but you may not suppress the table header indicating the number of constraints retained of the total number applied. This number is computed even if the current design is considered to be the converged optimum. A summary of the convergence criteria and of the critical constraint value is included in the Active Constraint Summary header, illustrated in Table 8-5, if the approximate problem was considered converged following the preceding redesign step.

Each redesign step is summarized in a small table, shown in Table 8-6, entitled the Approximate Optimization Summary. It indicates the optimization method used in resizing and the changes in three measures of convergence. The first measure is the change in the value of the objective function during the solution of the approximate optimization problem. The second is the change in the Euclidean norm of the design variable vector and finally, the maximum absolute change in any component of the design variable vector. Each of the values are computed as an absolute change and a percentage change. These values are then printed. You may compare the first two percentage values against your input convergence limit, denoted **UPPER BOUND PERCENT MOVE**, to determine which (if either) is greater than the limit. If either

Table 8-1. DEBUG and ASSIGN DATABASE Output

```
::::::::::::::::::::::::::::::::::::::::::::::::::
::                                              ::
::   AUTOMATED STRUCTURAL OPTIMIZATION SYSTEM   ::
::                                              ::
::   *****  *****  *****  *****  *****  *****   ::
::   *   *  *        *    *   *  *   *  *       ::
::   *****  *****    *    *****  *   *  *****   ::
::   *   *      *    *    *  *   *   *      *   ::
::   *   *  *****    *    *   *  *****  *****   ::
::                                              ::
::                VERSION 9.0                   ::
::                                              ::
::            IBM RISC SYSTEM/6000              ::
::                JUL 01, 1992                  ::
::                                              ::
::::::::::::::::::::::::::::::::::::::::::::::::::


***** ASTROS DEBUG PACKET ECHO  *****

DEBUG KEY "LOGBEGIN  " HAS BEEN SELECTED
DEBUG KEY "LOGMODULE " HAS BEEN SELECTED
DEBUG KEY "MATRIX    " HAS BEEN SELECTED

*****  ASTROS ASSIGN DATABASE COMMAND ECHO  *****

*...10...**...20...**...30...**...40...**...50...**...60...**...70...**...80...*

ASSIGN DATABASE COMB SHAZAM NEW

*...10...**...20...**...30...**...40...**...50...**...60...**...70...**...80...*

DATA BASE NAME     = COMB
DATA BASE PASSWORD = SHAZAM
DATA BASE STATUS   = NEW
USER PARAMETERS ARE:
** NONE GIVEN **
```

Table 8-2. Boundary Condition Summary

```
B O U N D A R Y   C O N D I T I O N   S U M M A R Y   F O R   B O U N D A R Y   C O N D I T I O N  2


MATRIX REDUCTION SUMMARY:
     THE PHYSICAL SET CONTAINS   3948 DEGREES OF FREEDOM (DOFS)
                                 3948 PHYSICAL DOFS ARE STRUCTURAL
                             AND     0 PHYSICAL DOFS ARE EXTRA POINTS

     THERE ARE     12 DEPENDENT MULTIPOINT CONSTRAINT DOFS LEAVING
                 3936 INDEPENDENT DOFS

     THERE ARE   1563 SINGLE POINT CONSTRAINT DOFS LEAVING
                 2373 FREE DOFS

     THE FREE DOFS ARE REDUCED USING STATIC CONDENSATION
     THERE ARE   2221 OMITTED DOFS LEAVING
                  152 ANALYSIS SET DOFS
                    0 OF WHICH ARE "SUPPORTED" LEAVING
                  152 DOFS LEFT OVER


DISCIPLINE-SUBCASE SUMMARY:
     *** STATICS HAS BEEN SELECTED ***
          5 SUBCASE(S) ARE DEFINED BY SOLUTION CONTROL
     *** FLUTTER HAS BEEN SELECTED ***
         A REAL EIGENANALYSIS WILL ALSO BE DONE IF NOT ALREADY SELECTED
          1 SUBCASE(S) ARE DEFINED BY SOLUTION CONTROL
```

Table 8-3. Active Boundary and Constraint Summary

```
S E N S I T I V I T Y   S U M M A R Y   F O R   B O U N D A R Y   C O N D I T I O N   : 1

     4 FLUTTER CONSTRAINTS
 ------
     4 TOTAL ACTIVE CONSTRAINTS FOR THIS BOUNDARY CONDITION.


S E N S I T I V I T Y   S U M M A R Y   F O R   B O U N D A R Y   C O N D I T I O N   : 2

     2 TSAI-WU STRESS CONSTRAINTS ON STATIC SUBCASE   1
     6 TSAI-WU STRESS CONSTRAINTS ON STATIC SUBCASE   2
     4 FLUTTER CONSTRAINTS
 ------
    12 TOTAL ACTIVE CONSTRAINTS FOR THIS BOUNDARY CONDITION.
```

Table 8-4. Resequencing Summary

```
      ***   S U M M A R Y   O F   A U T O M A T I C   R E S E Q U E N C I N G   ***

                  METHOD SELECTED                              CM
                  CRITERION                          RMS WAVEFRONT


                                                  BEFORE     AFTER
                  BANDWIDTH                          648        60
                  PROFILE                          25960     23960
                  MAXIMUM WAVEFRONT                   50        60
                  AVERAGE WAVEFRONT                39.453    36.413
                  RMS WAVEFRONT                    40.632    38.159


                  NUMBER OF GRID POINTS                         658
                  MAXIMUM NODAL DEGREE                           14

                  NUMBER OF MPC EQUATIONS PROCESSED              12

                  ELEMENTS PROCESSED
                                          CSHEAR                565
                                          CTRMEM                 40
                                          CONROD                305
                                          CONM2                  35
                                          CBAR                  152
                                          CQUAD4                662

                  TOTAL ELEMENTS                               1759
```

Table 8-5. Active Constraint Summary

```
          S U M M A R Y   O F   A C T I V E   C O N S T R A I N T S
                      AFTER ANALYSIS 4 OF A MAXIMUM 16
                    12 CONSTRAINTS RETAINED OF 60 APPLIED

 THE APPROXIMATE OPTIMIZATION PROBLEM WAS CONVERGED WITH
 FEASIBLE CONSTRAINT CRITERIA (CTLMIN)...:    5.00000E-04 AND
 ACTIVE CONSTRAINT CRITERIA   (CTL)......:   -7.50000E-04

 CURRENT MAXIMUM CONSTRAINT VALUE...:   -4.97155E-04

 TO TERMINATE                    ...:    -2.25000E-03   <  -4.97155E-04   <=  1.00000E-03

 *** ASTROS OPTIMIZATION HAS CONVERGED ***

 ************************************************************
 *         CONSTRAINT RETENTION ALGORITHM SUMMARY           *
 *         RFAC =    3.000, EPS =    -.100, NDV =      4    *
 *                                                          *
 *  # OF CONSTRAINTS RETAINED BY RFAC     =        12       *
 *       CUTOFF CONSTRAINT VALUE          =     -.981       *
 *                                                          *
 *  # ADDED WITH VALUES GREATER THAN EPS  =         0       *
 *                                                          *
 *  # OF ADDITIONAL MINIMUM THICKNESS                       *
 *    CONSTRAINTS RETAINED ONLY FOR                         *
 *    CONTROLLING MOVE LIMITS (DCONTHK)   =         0       *
 ************************************************************
```

COUNT	CONSTRAINT VALUE	CONSTRAINT TYPE	TYPE COUNT	BOUNDARY ID	SUBCASE	ELEMENT TYPE	EID/LAYR
1	-7.09291E-02	UPPER BND LIFT EFFECT	1	1	1	N/A	N/A
2	-4.97155E-04	DISPLACEMENT	1	1	1	N/A	N/A
3	-4.20226E-01	VON MISES STRESS	1	1	1	QDMEM1	13
4	-8.19597E-01	VON MISES STRESS	2	1	1	QDMEM1	14
5	-8.42767E-01	VON MISES STRESS	3	1	1	QDMEM1	16
6	-5.62036E-01	VON MISES STRESS	4	1	1	QDMEM1	17
7	-4.38053E-01	VON MISES STRESS	5	1	1	QDMEM1	20
8	-8.14886E-01	VON MISES STRESS	6	1	1	QDMEM1	21
9	-8.52344E-01	VON MISES STRESS	7	1	1	QDMEM1	23
10	-5.76223E-01	VON MISES STRESS	8	1	1	QDMEM1	26
11	-9.77223E-01	VON MISES STRESS	10	1	1	ROD	2
12	-9.81333E-01	VON MISES STRESS	28	1	1	SHEAR	29

Table 8-6. Approximate Optimization Summary

```
 ****          ASTROS APPROXIMATE OPTIMIZATION SUMMARY          ****
 ***                        ITERATION  1                        ***
 **    RESIZING METHOD          = MATHEMATICAL PROGRAMMING       **
 **    DESIGN VAR. MOVE LIMIT   =     2.000000                    *
 *     UPPER BOUND PERCENT MOVE =     1.000000 PERCENT            *
 *     CRITERION 1:  OBJECTIVE CHANGE                             *
 *                   CURRENT VALUE        =   4.3531E+01          *
 *                   PREVIOUS VALUE       =   2.7840E+01          *
 *                   DELTA                =   1.5691E+01          *
 *                   PERCENT MOVE         =     56.3603           *
 *     CRITERION 2:  DESIGN VECTOR MOVE                           *
 *                   NORM OF X-X0         =   1.3076E+00          *
 *                   EUCLIDEAN NORM OF X0 =   2.3200E+00          *
 *                   PERCENT MOVE         =     56.3603           *
 *     CRITERION 3:  DESIGN VARIABLE MOVE                         *
 *                   MAXIMUM MOVE         =   3.6390E-01          *
 *                   AT DESIGN VARIABLE   =        3              *
 *                   CURRENT VALUE        =   1.2339E+00          *
 *                   PREVIOUS VALUE       =   8.7000E-01          *
 *                   PERCENT MOVE         =     41.8275           *
 *     THE APPROXIMATE PROBLEM IS NOT CONVERGED                   *
 ****************************************************************
```

value is greater, the approximate problem will not be considered converged, otherwise it will be. A message indicating the state of convergence closes the Approximate Optimization Summary.

The last default design print, Table 8-7, is generated by the **ACTCON** module on the final design iteration. The **ACTCON** module prints out the design iteration history. The iteration history includes statistics summarizing each ***approximate*** optimization problem and shows the increments in the objective function. All values in this table are associated with the approximate problem. Since weight in ASTROS is explicitly linear in the design variables, the objective function values are exact.

The final default outputs are a trailer indicating the status of the termination (either with or without errors), the date and the time the run was completed and an execution timing summary. The timing summary, shown in Table 8-8, indicates the CPU time spent in each phase of the execution. The elapsed clock time is shown upon leaving each phase of the MAPOL execution. This summary is useful in determining where a problem may have occurred and in confirming that the proper path was taken through the MAPOL sequence. It is, of course, also useful as an indication of the relative CPU costs of each phase of execution.

8.1.2. Error Message Output

Error messages can be printed from virtually all the modules of the ASTROS system as well as from the data base management software. Database errors should not occur unless you have modified or otherwise written a special MAPOL sequence, incorrectly assigned file names or used other incorrect or inconsistent database information. Typically, database errors cause immediate termination of the execution. The system administrator should be able to assist in solving such problems which, it is felt, will most likely be

Table 8-7. Design Iteration History

A S T R O S D E S I G N I T E R A T I O N H I S T O R Y

ITERATION NUMBER	OBJECTIVE FUNCTION VALUE	NUMBER FUNCTION EVAL	NUMBER GRADIENT EVAL	NUMBER RETAINED CONSTRAINTS	NUMBER ACTIVE CONSTRAINTS	NUMBER VIOLATED CONSTRAINTS	NUMBER LOWER BOUNDS	NUMBER UPPER BOUNDS	APPROXIMATE PROBLEM CONVERGENCE
1	1.25894E+04	(INITIAL FUNCTION VALUE)							
2	7.12705E+03	31	4	18	1	0	0	0	NOT CONVERGED
3	6.36273E+03	30	6	18	1	0	1	1	NOT CONVERGED
4	6.08681E+03	39	8	18	1	0	3	1	NOT CONVERGED
5	5.89348E+03	47	11	18	1	0	4	0	NOT CONVERGED
6	5.74943E+03	56	13	18	1	0	4	0	NOT CONVERGED
7	5.62364E+03	38	9	18	1	0	4	1	NOT CONVERGED
8	5.50224E+03	40	10	18	1	0	4	1	NOT CONVERGED
9	5.38496E+03	24	7	18	1	0	4	1	NOT CONVERGED
10	5.27604E+03	36	8	18	2	0	4	1	NOT CONVERGED
11	5.18694E+03	43	11	18	2	0	4	1	NOT CONVERGED
12	5.14224E+03	33	5	18	2	0	4	1	NOT CONVERGED
13	5.13861E+03	10	2	18	2	0	4	1	NOT CONVERGED
14	5.13618E+03	10	1	18	2	0	4	1	NOT CONVERGED
15	5.11049E+03	18	3	18	2	0	4	1	CONVERGED

THE FINAL OBJECTIVE FUNCTION VALUE IS:

FIXED =	0.00000E+00
+ DESIGNED =	5.11049E+03
TOTAL =	5.11049E+03

Table 8-8. ASTROS Execution Summary

```
                              ************************************
                              ***                              ***
                              *** A S T R O S  T E R M I N A T E D ***
                              ***     02/16/93   11:45:50      ***
                              ***                              ***
                              ************************************


                    A S T R O S   T I M I N G   S U M M A R Y
 ELAPSED   TOTAL                                                    STEP
  TIME      CPU                                                     CPU
-------- ----------                                               ----------
00:00:00 00:00:00.0 *** BEGIN ASTROS ***
00:00:02 00:00:01.9 BEGIN PREFACE MODULES                        00:00:08.4
00:00:12 00:00:10.4 ELEMENT MATRIX GENERATION                    00:02:56.9
00:04:43 00:03:07.4 NON-PLANAR STEADY AERODYNAMICS               00:00:00.0
00:04:43 00:03:07.4 PHASE 1 ELEM. MATRIX ASSEMBLY                00:00:30.4
00:05:54 00:03:37.8 PHASE 1 STATIC LOADS GENER.                  00:00:00.2
00:05:55 00:03:38.0 STEADY AERODYNAMICS                          00:00:00.0
00:05:55 00:03:38.1 UNSTEADY AERODYNAMICS                        00:01:36.9
00:07:56 00:05:15.1 ****************************                 00:00:00.0
00:07:56 00:05:15.1 BEGIN OPTIMIZATION                           00:00:00.0
00:07:56 00:05:15.1 ----------------------------                 00:00:00.0
00:07:56 00:05:15.1    DESIGN ITERATION   1                      00:00:08.1
00:08:22 00:05:23.3       BOUNDARY CONDITION   1                00:00:02.8
00:08:28 00:05:26.1          MPC REDUCTION                       00:00:28.7
00:09:03 00:05:54.9          SPC REDUCTION                       00:00:03.6
00:09:07 00:05:58.5          STATIC CONDENSATION                 00:00:54.3
00:10:12 00:06:52.9          >>>DISCIPLINE: NORMAL MODES         00:00:01.9
00:10:15 00:06:54.8          >>>DISCIPLINE: FLUTTER              00:00:09.1
00:10:33 00:07:04.0          DATA RECOVERY                       00:00:00.0
00:10:33 00:07:04.1          STATIC CONDENSATION RECOVERY        00:00:00.7
00:10:34 00:07:04.8          SPC RECOVERY                        00:00:00.2
00:10:34 00:07:05.0          MPC RECOVERY                        00:00:00.3
00:10:34 00:07:05.4          CONSTRAINT EVALUATION               00:00:00.0
00:10:34 00:07:05.4          OUTPUT PROCESSING                   00:00:00.1
00:10:35 00:07:05.5       BOUNDARY CONDITION   2                00:00:00.7
                          .........
                          .........
                          .........
00:12:34 00:08:48.0          CONSTRAINT EVALUATION               00:00:19.7
00:13:20 00:09:07.8          OUTPUT PROCESSING                   00:00:00.1
00:13:20 00:09:07.9          SENSITIVITY ANALYSIS                00:00:44.7
00:14:47 00:09:52.7          DESIGN MODULE                       00:00:03.3
00:14:52 00:09:56.1 --------------------------                   00:00:00.0
00:14:52 00:09:56.1    DESIGN ITERATION   2                      00:00:08.2
                       .........
                       .........
                       .........
01:30:33 00:45:23.8 *** END ASTROS ***
```

caused by incorrect use of the system or by incorrect system installation. The ASTROS Programmer's Manual contains further information on the causes of particular database errors.

The standard ASTROS error messages are printed by the **UTMWRT** utility module and represent error checks that the modules are coded to perform or errors that may cause problems in the current module's algorithm. As much as possible, these error messages are intended to be standalone in that the user should be able to interpret the message without referring to the Programmer's Manual. There are four different levels of errors that can occur in ASTROS, each labeled differently when printed:

(1) **System Fatal Message**

These messages come about due to errors or inconsistencies in the system definitions. Usually, these relate to erroneous input to the system generation utility, SYSGEN, or are a result of using an outdated system data base. You should contact your system adminis-

trator to effect a correction. Hopefully, these errors will rarely occur and should never occur in an unmodified ASTROS system.

(2) **User Information Message**

These messages are written when the system encounters data that may represent an input error or may later generate a problem but that may only be a special user input that falls outside the expected range. Usually, these messages can be ignored. This is the least serious type of user message in ASTROS.

(3) **User Warning Message**

These messages are written when the system encounters data that are incorrect but which may not cause termination. In some cases, this level of error is issued to signify that the system will continue to search for errors but will terminate abnormally following the search.

(4) **User Fatal Message**

These messages are written when the system encounters data that are in error to the extent that continuation is impossible. The system will terminate execution either immediately or after some minor clean up.

If the user is unable to decipher the error message, the following steps can be helpful in determining the source of the error:

(1) Check the timing summary with the LOGBEGIN and LOGMODULE options in the DEBUG packet against the MAPOL sequence path to determine which module generated the error message. Also, check the SYSGEN output to determine the module that wrote the message. Note that the "message number" is included in the error message print if the message is a standard one and the message number can be used to trace the module that uses the message.

(2) Check the Programmer's Manual documentation for the relevant module to determine the error checks it performs and to get further information on the source of the error.

8.2. SOLUTION CONTROL OUTPUT OPTIONS

This Section presents a detailed description of the output quantities that can be selected through the solution control packet. These quantities fall into five categories: (1) element; (2) nodal; (3) design; (4) eigenvalues for flutter and normal modes; and (5) aeroelastic trim quantities. Each of these categories is presented in the separate subsections that follow.

The **PRINT** and **PUNCH** solution control commands are used to request the desired output quantities. These commands have three groups of options: subset options, quantity options and form options. These options are fully described in Chapter 4 of this manual, but one point must be stressed: subset options play an extremely important role in ASTROS output requests. Subset options allow you to identify the set of iterations or subcases to which the print selection will apply. This selection is necessary because many disciplines (**MODES**, for example) generate more than one subcase (eigenvector) with a single solution control directive. The critical point is that the default selection for subcases is that there be no output. In other words, if there are no subcases selected, ASTROS will, by default, ***print nothing***.

Unlike NASTRAN, ASTROS has no options to reorder the output. The multidisciplinary nature of ASTROS completely negates the utility of the **SORT1** and **SORT2** options found in NASTRAN variants, and any other sort options become impossibly complex very quickly. Instead, a reasonable, fixed sort is

established in which each boundary condition is treated separately and in the order given in the solution control packet. If the standard sequence is used, the response quantities will appear in the following order within each optimize or analyze boundary condition:

(1) Steady aerodynamic trim parameters.
(2) Flutter roots and flutter mode shape modal participation factors -- note that the mode shape is only available if flutter has occurred and if the **FLUTTER** discipline is within an **ANALYZE** boundary condition.
(3) Applied **LOAD** print requests.
(4) The "displacement" nodal response quantities: **DISPLACEMENT**s, **VELOCITY**s, and **ACCELERATION**s.
(5) Element response quantities in the order **STRESS**, **STRAIN**, **FORCE** and **STRAIN ENERGY** for each subcase, elements are processed alphabetically within each quantity type.

In the **OPTIMIZE** subpacket, these data are followed by the selected design and resizing prints in the following order:

(7) Active constraint summary (either the default abbreviated print or the full print if the **DCONSTRAINT** print option is selected).
(8) The print of the global and then local design variables representing the current design depeding on the **GDESIGN** and **LDESIGN PRINT** requests. On the final design iteration, the iteration history precedes the design variable output by default.

Within each response quantity's print module, the disciplines are not treated in the order given in the solution control packet; instead, they are treated, where applicable, in the following order:

(A) **STATICS**
(B) **MODES**
(C) **SAERO**
(D) **TRANSIENT**
(E) **FREQUENCY**

The subcases within each discipline are treated in the order given in the solution control packet. In the case of **MODES**, the eigenvectors are ordered in increasing eigenvalue order. **TRANSIENT** and **FREQUENCY** subcases are ordered in increasing time or frequency step.

8.2.1. Element Response Quantities

ASTROS has two basic forms of elements: aerodynamic elements and structural elements. An aerodynamic element is defined as a "box" of an aerodynamic macroelement, e.g., wing component or fuselage segment. The nature of the macroelement varies among both aerodynamic models and among aerodynamic components within each model. In general, however, a box is the smallest subdivision of the aerodynamic component for which data (e.g., pressures, forces, and moments) are computed. Structural elements are either metric elements, which connect structural node points (grids); scalar elements, which connect pairs of degrees of freedom or pairs of scalar points; or mass elements. Table 8-9 shows the list of aerodynamic and structural elements in ASTROS for which element output exist. The following subsections document the quantities that are available as output for each of these elements. The structural mass elements are not included in this table since they have no element response quantities. The NASTRAN User's Manual (Reference 2) was used as a major resource in writing this section and you are referred to it for additional information on the structural elements.

Table 8-9. ASTROS Aerodynamic and Structural Elements

AERODYNAMIC	STRUCTURAL
CAERO1 CAERO2 CAERO6 PAERO6	CBAR CELAS1, CELAS2 CIHEX1. CIHEX2, CIHEX3 CROD, CONROD CQDMEM1, CTRMEM CTRIA3, CQUAD4

Structural element output is available for all disciplines that result in a real displacement field. This includes **STATICS**, **MODES**, **TRANSIENT**, and **SAERO** analyses. Complex displacement fields (from **FLUTTER** and **FREQUENCY** analyses) result in computation of the selected (complex) element response quantities, but their formatted print is not available except through executive sequence print utilities described in Subsection 3.4. For all disciplines in ASTROS, the solution control print options **STRESS**, **STRAIN**, **FORCE**, and **ENERGY** are used to select print of the structural element quantities. The **AIRDISP** and **TPRESSURE** options are used for aerodynamic element quantities. Each of these print options selects either **ALL**, **NONE** or an integer set identification number that refers to one or more **ELEMLIST** bulk data entries specifying which elements are to have output computed and printed. Chapter 4 contains the complete description of the solution control print command. Each output is carefully labeled as to its boundary condition number, discipline generating the response field and load condition, mode number, time step, frequency step or flight condition represented by the output.

8.2.1.1. Aerodynamic Element Output

The solution control **PRINT** option **TPRESSURE** provides the trimmed pressures on the aerodynamic boxes for **SAERO**. The trimmed pressures are computed and stored in the relational entity **OAGRDLOD**. The **AIRDISP** print option is available for the **SAERO** discipline and provides the out-of-plane displacements and streamwise slopes of the aerodynamic boxes that coorespond to the structural displacements. These data are computed and stored on the relational entity **OAGRDDSP**.

Aerodynamic geometry data are computed and stored by default to a set of relational entities that parallel the structural model. These data forms are designed primarily for model checkout of the **SAERO** model using existing FE preprocessors that support NASTRAN-style input data. These relations are: **AEROGEOM** which supplies the GRID-like data and **CAROGEOM** which provides connectivity data for the boxes in a ROD or QUAD4 form. The ROD is used to model the outline of the airfoils and the QUAD4 elements are used to model the boxes.

For unsteady aerodynamics, the box-on-box aerodynamic forces are only available through the **DEBUG/UNSTEADY** and **DEBUG/AMP** options (see Chapter 2). The geometry data are not available.

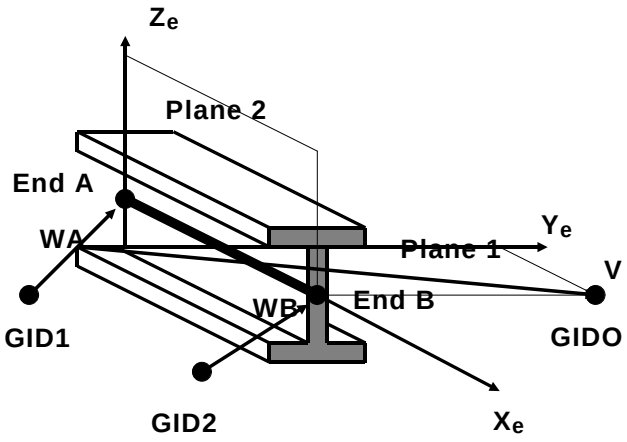


Figure 8-1. BAR Element Coordinate System

8.2.1.2. Bar Element Output

The BAR element includes extension, torsion, bending in two perpendicular planes and the associated shears. The shear center is assumed to coincide with the elastic axis. The BAR element coordinate system is shown in Figure 8-1. The orientation of the BAR element is described in terms of two reference planes defined through the use of the orientation vector, v, as shown in that figure. The positive directions for the element forces are shown in Figure 8-2. Additional information on the structural elements is contained in Chapter 5 of the ASTROS Theoretical Manual.

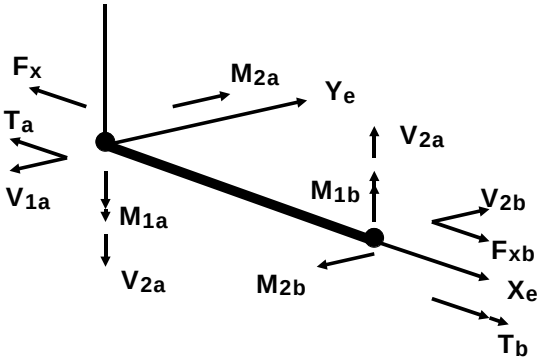


Figure 8-2. BAR Element Forces Sign Conventions

Stresses, strains, forces and strain energies are available as output for the BAR element through the **STRESS**, **STRAIN**, **FORCE**, and **ENERGY** solution control print options. The following element forces are output on request:

(1) Bending moments at each end in both reference planes.
(2) Shear forces in each reference plane.
(3) Average axial force.
(4) Torque about the bar axis.

The following element stresses and strains in the element coordinate system are output on request:

(1) Average axial stress or strain.
(2) Extensional stress or strain due to bending at 4 points on the cross-section at each end.
(3) Maximum and minimum stress or strain at each end.
(4) Stress margins of safety for the element in both tension and compression.

Tensile stresses and strains are given a positive value while compressive stresses and strains are given a negative value. The bending contribution to the stresses are always computed at the four points on the element cross-section that were specified on the connectivity entry for the BAR element. This means that the safety margins are computed using all eight stress values even if all four stress points at each end are

Table 8-10. BAR Element Output Quantities

```
1                                                                                    ASTROS VERSION 9.0   03/03/93   P.   16
                                                                                     FINAL ANALYSIS SEGMENT
                                                                    TRANSIENT ANALYSIS: BOUNDARY 1, TIME = 4.9999997E-02
                          S T R E S S E S   I N   B A R   E L E M E N T S      ( B A R )
     ELEMENT       SA1           SA2           SA3           SA4          AXIAL         SA-MAX        SA-MIN    M.S.-T
       ID.         SB1           SB2           SB3           SB4          STRESS        SB-MAX        SB-MIN    M.S.-C
         106 -2.810616E+03  0.000000E+00 -2.810616E+03  2.810616E+03  0.000000E+00  2.810616E+03 -2.810616E+03  1.7E+38
             7.498176E-02  0.000000E+00  7.498176E-02 -7.498176E-02                7.498176E-02 -7.498176E-02  1.7E+38

1                                                                                    ASTROS VERSION 9.0   03/03/93   P.   23
                                                                                     FINAL ANALYSIS SEGMENT
                                                                    TRANSIENT ANALYSIS: BOUNDARY 1, TIME = 9.9999998E-03
                          S T R A I N S   I N   B A R   E L E M E N T S      ( B A R )
     ELEMENT       SA1           SA2           SA3           SA4          AXIAL         SA-MAX        SA-MIN
       ID.         SB1           SB2           SB3           SB4          STRAIN        SB-MAX        SB-MIN
         106 -1.963023E-04  0.000000E+00 -1.963023E-04  1.963023E-04  0.000000E+00  1.963023E-04 -1.963023E-04
             5.857950E-10  0.000000E+00  5.857950E-10 -5.857950E-10                5.857950E-10 -5.857950E-10

1                                                                                    ASTROS VERSION 9.0   03/03/93   P.   31
                                                                                     FINAL ANALYSIS SEGMENT
                                                                    TRANSIENT ANALYSIS: BOUNDARY 1, TIME = 9.9999998E-03
                          F O R C E S   I N   B A R   E L E M E N T S      ( B A R )
   ELEMENT       BEND-MOMENT END-A          BEND-MOMENT END-B              - SHEAR -            AXIAL
     ID.       PLANE 1       PLANE 2       PLANE 1       PLANE 2        PLANE 1       PLANE 2        FORCE         TORQUE
       106  0.000000E+00  3.195801E+00  0.000000E+00 -9.536743E-06  0.000000E+00  6.391621E-01  0.000000E+00  0.000000E+00
1                                                                                    ASTROS VERSION 9.0   03/03/93   P.   31
                                                                                     FINAL ANALYSIS SEGMENT
                                                                    TRANSIENT ANALYSIS: BOUNDARY 1, TIME = 9.9999998E-03

                                   E L E M E N T   S T R A I N   E N E R G I E S

              BAR      ELEMENTS                 TOTAL ENERGY OF ALL ELEMENTS IN THE SUBCASE  =      4.388508E-02

                                    ELEMENT ID           STRAIN ENERGY   PERCENT OF TOTAL
                                          101           1.430531E-03          3.259722
                                          102           4.057172E-04           .924499
                                          103           4.288774E-03          9.772738
                                          104           1.405888E-02         32.035667
                                          105           1.847305E-02         42.094154
                                          106           5.228124E-03         11.913216
              BAR      ELEMENTS   SUBTOTAL            4.388508E-02        100.000000
```

the same and/or coincide with the element axis. Also, margins of safety are printed even if no stress limits were given on the material entry. In these cases, a very large value for the margin of safety is used to indicate that no limits were specified. In addition, ASTROS fully supports strain output for the BAR element. Strain energies may also be requested for the BAR element. The strain energy print (which is identical for all ASTROS structural elements) is patterned after that in NASTRAN. It shows the total strain energy for the given displacement field, the strain energy in each selected element and the total strain energy for all the elements of a given type, e.g., all the BAR elements. Examples of each of these outputs are shown in Table 8-10.

8.2.1.3. ELAS Element Output

The ELAS element is a scalar spring element which relates the displacements at a pair of scalar points or degrees of freedom or that relates a single degree of freedom to a ground state. The element force and strain energy are directly available for the element and the user can, if desired, input a scalar quantity that relates the "stress" in the element to the displacement(s) of the connected degree(s) of freedom. On output, these values will be printed for each output request for each selected ELAS element. Strains have no meaning for the scalar spring element and any such requests will be ignored without warning. Element strain energies, however, are available for the element and are computed from the spring constant and the nodal displacement(s). The strain energy print for the ELAS is identical to that for the BAR element and includes a breakdown by element and by element type. If no scalar value is given for the element stress but the stress value is requested, a value of zero will be computed and printed for the response quantity with no warnings given.

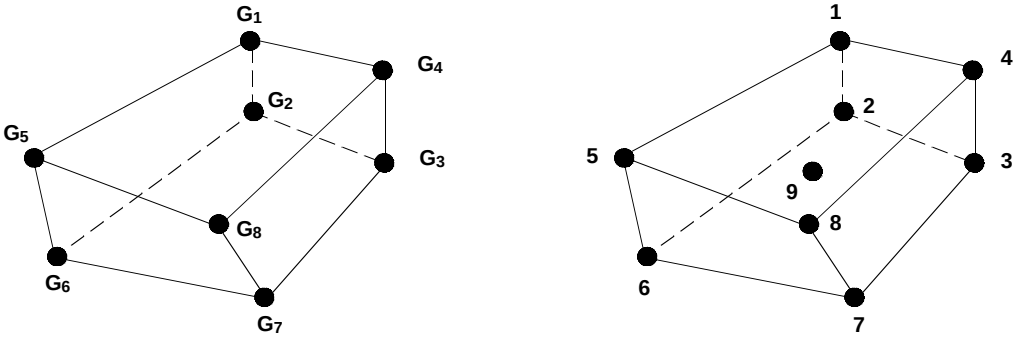
8.2.1.4. IHEX1 Element Output

The IHEX1 element is a linear isoparametric solid hexahedron element with three extensional degrees of freedom for each of its eight nodes.

Stresses, strains, and strain energies are available as output for the IHEX1 element through the **STRESS**, **STRAIN**, and **ENERGY** solution control print command options. Force output is not available for the IHEX1 element. On request, the following stresses and strains are output in the basic coordinate system at the center and at each corner grid point:

(1) Normal stresses or strains in all three directions.
(2) Shear stresses or strains in all three planes.
(3) Principal stresses or strains in all three directions with associated direction cosines.
(4) Mean stress or strain.
(5) Octahedral shear stress or strain.

The stress and strain output at each of the nine points is identified by a stress or strain point ID. The stress and strain point IDs are numbered 1 through 9, with the first eight ordered as on the associated **CIHEX1** input data entry, and the ninth located at the element center, as illustrated in Figure 8-3. All output is provided in the basic coordinate system, since there is no naturally occurring element coordinate system for the IHEX1. An example of the output for the IHEX family of elements is shown in Table 8-11. The IHEX1 element is shown with the IHEX2 and IHEX3 elements differing only in the number of data recovery points.



(a) ELEMENT GRID POINTS **(b) ELEMENT STRESS POINTS**

Figure 8-3. IHEX1 Element Geometry

Strain energy output may be requested for the IHEX1 element. The strain energy print for the IHEX1 is identical to that for the BAR element and includes a breakdown by element and by element type.

Table 8-11. IHEX1 Element Solution Quantities

```
                    S T R E S S E S   I N   8 - N O D E D   S O L I D   E L E M E N T     ( I H E X 1 )
ELEMENT  STRESS       -----CENTER AND CORNER POINT STRESSES------      DIRECTION  COSINES         MEAN         OCTAHEDRAL
  ID.     POINT       NORMAL             SHEAR           PRINCIPAL          A      B      C        STRESS       SHEAR STRESS
   123      1    X -7.617902E+01  XY -2.852164E+01  A -6.942204E+00  LX    .52    .85    .00   4.715845E+01   4.009525E+01
                 Y -3.264816E+01  YZ  0.000000E+00  B -1.018850E+02  LY   -.58    .35    .74
                 Z -3.264816E+01  ZX -3.108560E+01  C -3.264816E+01  LZ   -.63    .38   -.68

   123      2    X  7.238755E+01  XY -2.909098E+01  A  1.015376E+02  LX    .84    .54    .00  -4.481134E+01   4.183963E+01
                 Y  3.102324E+01  YZ  0.000000E+00  B  1.873196E+00  LY   -.35    .54    .77
                 Z  3.102324E+01  ZX  3.477372E+01  C  3.102325E+01  LZ    .41   -.65    .64
                    ............
                    ............
                    ............
   123      9    X  0.000000E+00  XY -2.000000E+01  A  2.535997E+01  LX    .62    .77    .15  -2.955829E+00   1.710619E+01
                 Y  9.217892E+00  YZ -3.080879E+00  B -1.615026E+01  LY   -.78    .62    .00
                 Z -3.504066E-01  ZX  0.000000E+00  C -3.422195E-01  LZ    .09    .12   -.99


                    S T R A I N S   I N   8 - N O D E D   S O L I D   E L E M E N T     ( I H E X 1 )
ELEMENT  STRAIN       ------CENTER AND CORNER POINT STRAINS------      DIRECTION  COSINES         MEAN         OCTAHEDRAL
  ID.     POINT       NORMAL             SHEAR           PRINCIPAL          A      B      C        STRAIN       SHEAR STRAIN
   123      1    X -5.659012E-06  XY -7.415626E-06  A  8.498356E-06  LX    .61    .79    .00   1.886337E-06   9.344843E-06
                 Y  0.000000E+00  YZ  0.000000E+00  B -1.415737E-05  LY   -.53    .41    .00
                 Z  0.000000E+00  ZX -8.082256E-06  C  0.000000E+00  LZ   -.58    .45    .00

   123      2    X  5.377361E-06  XY -7.563655E-06  A  1.477920E-05  LX    .78    .62    .00  -1.792454E-06   9.952897E-06
                 Y  0.000000E+00  YZ  0.000000E+00  B -9.401835E-06  LY   -.40    .50    .00
                 Z  0.000000E+00  ZX  9.041167E-06  C  0.000000E+00  LZ    .48   -.60    .00
                    ............
                    ............
                    ............
   123      9    X -2.660247E-07  XY -5.200001E-06  A  5.628016E-06  LX    .66    .74    .15  -1.182329E-07   4.334298E-06
                 Y  9.323016E-07  YZ -8.010304E-07  B -4.962797E-06  LY   -.75    .67    .00
```

8.2.1.5. IHEX2 Element Output

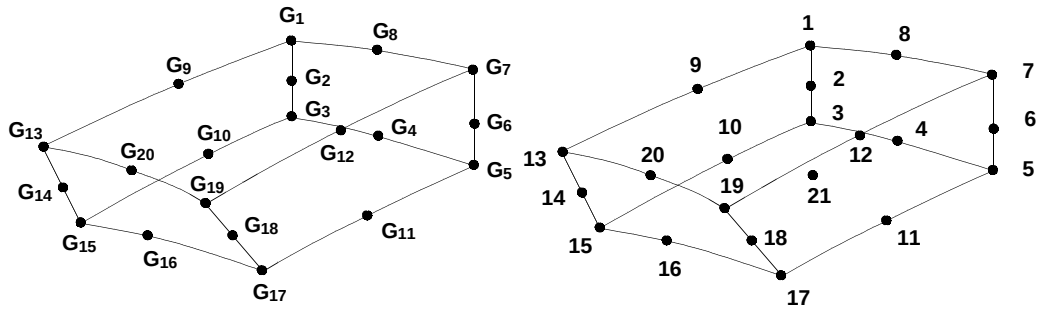
The IHEX2 element is a quadratic isoparametric solid hexahedron element with three extensional degrees of freedom for each of its 20 nodes.

Stresses, strains, and strain energies are available as output for the IHEX2 element through the **STRESS**, **STRAIN**, and **ENERGY** solution control print command options. Force output is not available for the IHEX2 element. On request, the following stresses and strains are output in the basic coordinate system at the twenty-one points located at the center, corners, and mid-edges of the element:

(1) Normal stresses or strains in all three directions.
(2) Shear stresses or strains in all three planes.
(3) Principal stresses or strains in all three directions with associated direction cosines.
(4) Mean stress or strain.
(5) Octahedral shear stress or strain.

The stress and strain output at each of the 21 points is identified by a stress or strain point ID. The stress and strain point IDs are numbered 1 through 21, with the first 20 ordered as on the associated **CIHEX2** input data entry, and the 21st located at the element center. Although the corner stress and strain points are located at the corner grid points of the element, the mid-edge stress and strain points may or may not be located at the mid-edge grid points, depending on the location of those grid points. The stress/strain points for the IHEX2 are illustrated in Figure 8-4. All output is given in the element coordinate system for the IHEX2.

Strain energy output may be requested for the IHEX2 element. The strain energy print for the IHEX2 is identical to that for the BAR element and includes a breakdown by element and by element type.



(a) ELEMENT GRID POINTS **(b) ELEMENT STRESS POINTS**

Figure 8-4. IHEX2 Element Geometry

8.2.1.6. IHEX3 Element Output

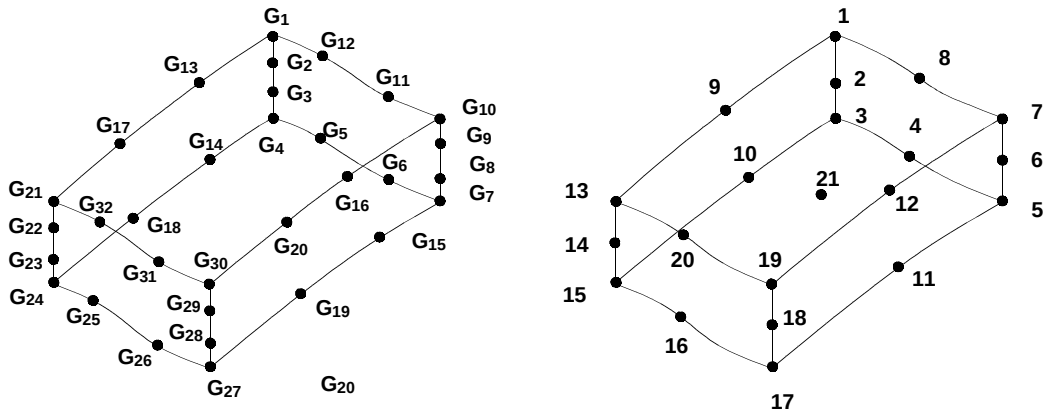
The IHEX3 element is a cubic isoparametric solid hexahedron element with three extensional degrees of freedom for each of its 32 nodes.

Stresses, strains, and strain energies are available as output for the IHEX3 element through the **STRESS**, **STRAIN**, and **ENERGY** solution control print command options. Force output is not available for the IHEX3 element. On request, the following stresses and strains are output in the basic coordinate system at the 21 points located at the center, corners, and mid-edges of the element:

(1) Normal stresses or strains in all three directions.
(2) Shear stresses or strains in all three planes.
(3) Principal stresses or strains in all three directions with associated direction cosines.
(4) Mean stress or strain.
(5) Octahedral shear stress or strain.

The stress and strain output at each of the 21 points is identified by a stress or strain point ID. The stress and strain point IDs are numbered 1 through 21. The first 20 points are ordered as on the associated **CIHEX3** input data entry, except that there is only one mid-edge point per edge, instead of two, and the 21st point is located at the element center. Although the corner stress and strain points are located at the corner grid points of the element, the mid-edge stress and strain points may or may not be located at a grid point, depending on the location of the mid-edge grid points. The stress/strain points for the IHEX3 are illustrated in Figure 8-5. All output is provided in the basic coordinate system, since there is no naturally occurring element coordinate system for the IHEX3.

Strain energy output may be requested for the IHEX3 element. The strain energy print for the IHEX3 is identical to that for the BAR element and includes a breakdown by element and by element type.



(a) ELEMENT GRID POINTS **(b) ELEMENT STRESS POINTS**

Figure 8-5. IHEX3 Element

8.2.1.7. Rod Element Output

The ASTROS ROD element supports both extensional and rotational properties. The element coordinate system and sign conventions are shown in Figure 8-6. ASTROS supports stress, strain, force and strain energy output for the ROD. The forces that are computed are:

(1) Axial force.
(2) Torque about the element axis.

The torque and force are both computed even if the particular element does not support torsional or extensional forces, respectively. In these cases, a value of zero will be printed for the appropriate response quantity. The stresses and/or strains that are available are:

(1) Axial stress or strain.
(2) Torsional stress or strain.
(3) Margin of safety for axial stress.
(4) Margin of safety for torsional stress.

The margins of safety for strain are not available and the stress margins are computed even if there are no limits specified on the material property entry. In these cases, a large safety margin value is used to signify that no limits were imposed. An example of the ROD element output prints is shown in Table 8-12. The strain energy print for the ROD is identical to that for the BAR element and includes a breakdown by element and by element type.

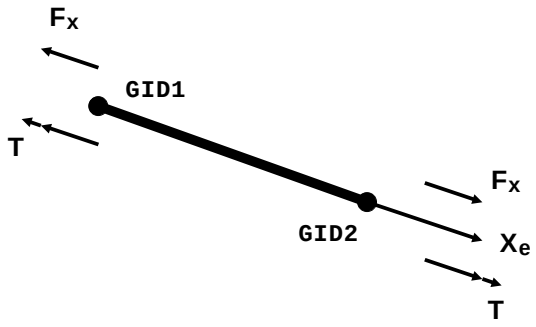


Figure 8-6. ROD Element Coordinate System

8.2.1.8. QDMEM1/TRMEM Element Output

The QDMEM1 isoparametric element and the TRMEM constant strain triangular element are membrane elements which support isotropic, orthotropic and composite membrane properties. If the element is composite, the individual layers are treated as independent, stacked elements in which each "layer," as defined on the **PCOMP** bulk data entry, represents an element. In the case of composite elements, the layers are numbered sequentially starting with the first layer appearing on the **PCOMP** entry. Non-composite elements will show a layer number of zero.

Table 8-12. ROD Element Solution Quantities

```
                          S T R E S S E S   I N   R O D   E L E M E N T S      ( R O D )
     ELEMENT      AXIAL      SAFETY    TORSIONAL    SAFETY      ELEMENT      AXIAL      SAFETY     TORSIONAL    SAFETY
       ID.       STRESS      MARGIN     STRESS      MARGIN        ID.       STRESS      MARGIN      STRESS      MARGIN
           1  6.512166E+03  2.8E+00  0.000000E+00  1.7E+38            2  1.337487E+03  1.8E+01  0.000000E+00  1.7E+38
           3 -6.821167E+03  2.7E+00  0.000000E+00  1.7E+38            4 -1.995846E+03  1.2E+01  0.000000E+00  1.7E+38
1TEN BAR TRUSS                                                                          ASTROS VERSION 9.0   03/03/93   P.   11
                                                                                        FINAL ANALYSIS SEGMENT
 FINAL STATIC ANALYSIS                                                                STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1
                          S T R A I N S   I N   R O D   E L E M E N T S      ( R O D )
     ELEMENT      AXIAL                TORSIONAL                ELEMENT      AXIAL                TORSIONAL
       ID.       STRAIN                 STRAIN                   ID.       STRAIN                 STRAIN
           1  6.512166E-04          0.000000E+00                    2  1.337487E-04          0.000000E+00
           3 -6.821167E-04          0.000000E+00                    4
1TEN BAR TRUSS                                                                          ASTROS VERSION 9.0   03/03/93   P.   12
                                                                                        FINAL ANALYSIS SEGMENT
 FINAL STATIC ANALYSIS                                                                STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1
                          F O R C E S   I N   R O D   E L E M E N T S      ( R O D )
     ELEMENT      AXIAL                                         ELEMENT      AXIAL
       ID.        FORCE                  TORQUE                   ID.        FORCE                  TORQUE
           1  1.953650E+05          0.000000E+00                    2  4.012462E+04          0.000000E+00
           3 -2.046350E+05          0.000000E+00                    4
```

Stresses, strains, forces and strain energies are available for each element or layer of a composite element. Since the stresses, strains, and forces vary within a **CQDMEM1** element, the intersection point of the diagonals projected onto the mean plane of a warped element has been chosen as the point at which the stresses, strains, forces and strain energies for the element are computed. The stresses, strains and element forces are computed in the element coordinate system. The element coordinate system and the stress computation point for the QDMEM1 element are shown in Figure 8-7 and those for the TRMEM in Figure 8-8.

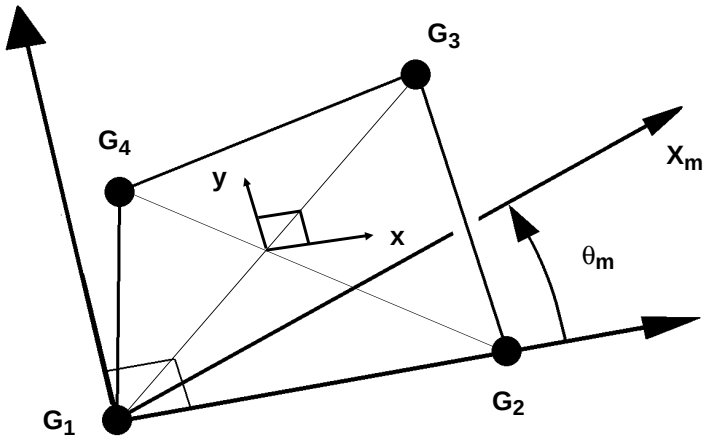


Figure 8-7. QDMEM1 Element Coordinate System

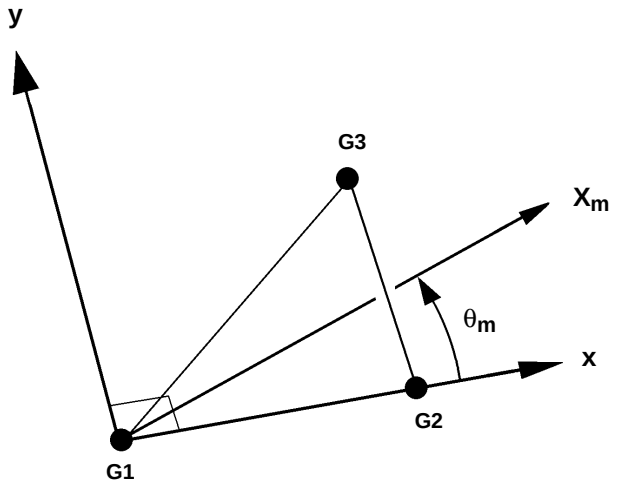


Figure 8-8. TRMEM Element Coordinate System

ASTROS computes the running loads associated with the stresses for the QDMEM1 element. These forces are:

(1) The force components in the element coordinate system at the stress computation point.

The QDMEM1 stress and strain print includes the following:

(2) The normal stresses or strains at the stress point in the element x- and y-directions.
(3) The shear stress or strain on the element x face in the element y-direction.
(4) The angle in degrees between the element x-axis and the major principal axis.
(5) The major and minor principal (zero shear) stresses or strains.
(6) The maximum shear stress or strain.

An example of the printed output for the QDMEM1 is shown in Table 8-13. The output for the TRMEM is identical except for the titling. The strain energy print for the QDMEM1 is identical to that for the BAR element and includes a breakdown by element and by element type.

8.2.1.9. QUAD4/TRIA3 Element Output

The QUAD4 and TRIA4 isoparametric quadrilateral and triangular plate elements include both membrane and bending behavior. Transverse shear flexibility may be requested, as can the coupling of membrane and bending behavior. The QUAD4 element coordinate system and node numbering are shown in Figure 8-9. The TRIA3 element coordinate system and node numbering are shown in Figure 8-10. These elements may be assigned general anisotropic or composite material properties. For designed composites, the layers are treated as stacked membrane elements similar to the QDMEM1 element. In this case, the layers are identified by number in the order specified on the **PCOMP**, **PCOMP**1 or **PCOMP**2 entry. For design invariant composite laminates, the output always refers to the aggregate laminate properties and refers to layer number zero. The reference plane of the QUAD4/TRIA3 elements may be offset from the plane of the grid points and variation in the element thickness may be modeled by

Table 8-13. QDMEM1 Solution Quantities

SIMPLIFIED FIGHTER WING ASTROS VERSION 9.0 03/03/93 P. 13
FINAL ANALYSIS SEGMENT
STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

S T R E S S E S I N Q U A D R I L A T E R A L M E M B R A N E S (Q D M E M 1)

ELEMENT ID.	LAYER NO.	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESS ANGLE	PRINCIPAL STRESSES MAJOR	MINOR	MAX SHEAR
1	0	5.748721E+02	-2.858559E+03	-1.106961E+03	-16.4072	9.008202E+02	-3.184507E+03	2.042664E+03
7	0	-5.322678E+02	-7.550425E+03	-2.563776E+03	-18.0762	3.045232E+02	-8.387216E+03	4.345870E+03
13	0	-6.414443E+02	3.114750E+03	1.165665E+03	74.0868	3.447088E+03	-9.737824E+02	2.210435E+03

SIMPLIFIED FIGHTER WING ASTROS VERSION 9.0 03/03/93 P. 15
FINAL ANALYSIS SEGMENT
STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

S T R A I N S I N Q U A D R I L A T E R A L M E M B R A N E S (Q D M E M 1)

ELEMENT ID.	STRAINS IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRAIN ANGLE	PRINCIPAL STRAINS MAJOR	MINOR	MAX SHEAR
1	1.518198E-04	-3.048268E-04	-2.944048E-04	-16.4051	1.951582E-04	-3.481652E-04	5.433233E-04
7	1.959372E-04	-7.374777E-04	-6.818553E-04	-18.0740	3.071982E-04	-8.487387E-04	1.155937E-03
13	-1.669312E-04	3.326433E-04	3.100170E-04	74.0889	3.768310E-04	-2.111190E-04	5.879501E-04

SIMPLIFIED FIGHTER WING ASTROS VERSION 9.0 03/03/93 P. 17
FINAL ANALYSIS SEGMENT
STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

F O R C E S I N Q U A D R I L A T E R A L M E M B R A N E S (Q D M E M 1)

ELEMENT ID.	- M E M B R A N E - FX	FY	FXY
1	1.149744E+02	-5.717117E+02	-2.213922E+02
7	-1.064536E+02	-1.510085E+03	-5.127553E+02
13	-1.282889E+02	6.229500E+02	2.331330E+02

assigning different element thicknesses at each of the grid points. The reader is referred to Appendix A of the ASTROS Theoretical Manual for additional information on these plate bending elements.

Stresses, strains, forces, and strain energies are available as output for the QUAD4/TRIA3 elements through the **STRESS**, **STRAIN**, **FORCE** and **ENERGY** solution control print command options. The following element stresses and strains are output on request:

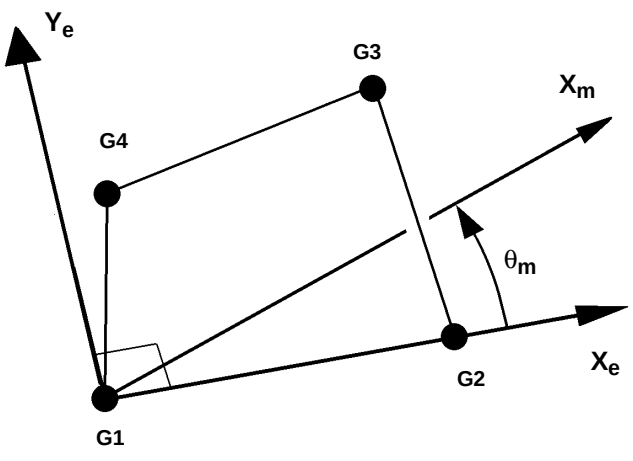


Figure 8-9. QUAD4 Element Coordinate System

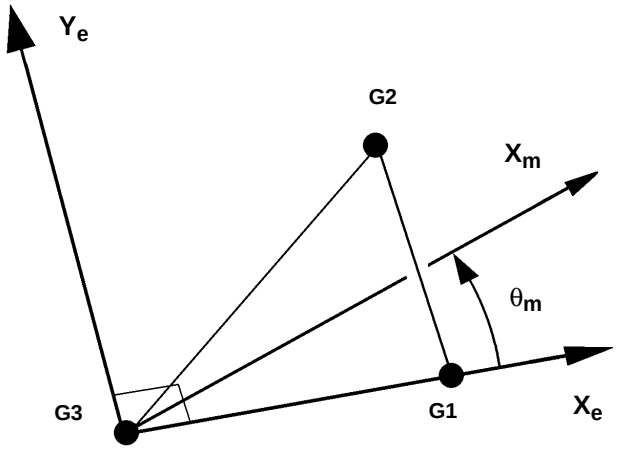


Figure 8-10. TRIA3 Element Coordinate System

(1) Combined extensional and bending stresses and strains computed at the element center in the element coordinate system.

(2) Principal stresses and strains computed at the element center including the angle between the element x-axis and the principal axis.

The following forces are output on request:

(1) Element forces computed at the center of the element in the mean plane in the element coordinate system.

For composite materials, all output quantities are computed using the aggregate laminate properties. Hence, output of stresses or strains at the ply or laminae level is currently not an available print option for the QUAD4/TRIA3 elements in ASTROS. An example of the printed output is shown in Table 8-14.

Table 8-14. QUAD4 and TRIA3 Solution Quantities

1SIMPLIFIED FIGHTER WING ASTROS VERSION 9.0 03/03/93 P. 13
FINAL ANALYSIS SEGMENT
STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

S T R E S S E S I N Q U A D R I L A T E R A L P L A T E S (Q U A D 4)

ELEMENT ID	LAYER NO.	FIBER DISTANCE	STRESSES IN STRESS COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR
3	0	-1.00000E-01	6.91372E+02	-8.33210E+03	-2.21727E+03	-13.0858	1.20677E+03	-8.84749E+03
		1.00000E-01	6.91372E+02	-8.33210E+03	-2.21727E+03	-13.0858	1.20677E+03	-8.84749E+03
7	0	-1.00000E-01	-5.43684E+02	-7.64178E+03	-2.58719E+03	-18.0457	2.99228E+02	-8.48469E+03
		1.00000E-01	-5.43683E+02	-7.64178E+03	-2.58719E+03	-18.0457	2.99228E+02	-8.48469E+03
13	0	-1.00000E-01	-6.16923E+02	3.07066E+03	1.17650E+03	73.7292	3.41404E+03	-9.60305E+02
		1.00000E-01	-6.16923E+02	3.07066E+03	1.17650E+03	73.7292	3.41404E+03	-9.60305E+02

1SIMPLIFIED FIGHTER WING ASTROS VERSION 9.0 03/03/93 P. 14
FINAL ANALYSIS SEGMENT
STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

S T R A I N S I N Q U A D R I L A T E R A L P L A T E S (Q U A D 4)

ELEMENT ID	LAYER NO.	FIBER DISTANCE	STRAINS IN STRESS COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRAINS (ZERO SHEAR) ANGLE	MAJOR	MINOR
3	0	-1.00000E-01	3.50886E-04	-8.56139E-04	-5.91205E-04	-13.0479	4.19391E-04	-9.24645E-04
		1.00000E-01	3.50886E-04	-8.56139E-04	-5.91205E-04	-13.0479	4.19391E-04	-9.24645E-04
7	0	-1.00000E-01	1.93048E-04	-7.46862E-04	-6.85105E-04	-18.0443	3.04642E-04	-8.58457E-04
		1.00000E-01	1.93048E-04	-7.46862E-04	-6.85105E-04	-18.0443	3.04642E-04	-8.58457E-04
13	0	-1.00000E-01	-1.64593E-04	3.26487E-04	3.13462E-04	73.7247	3.72245E-04	-2.10351E-04
		1.00000E-01	-1.64593E-04	3.26487E-04	3.13462E-04	73.7247	3.72245E-04	-2.10351E-04

1SIMPLIFIED FIGHTER WING ASTROS VERSION 9.0 03/03/93 P. 15
FINAL ANALYSIS SEGMENT
STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

F O R C E S I N Q U A D R I L A T E R A L P L A T E S (Q U A D 4)

ELEMENT ID	- MEMBRANE FORCES - FX	FY	FXY	- BENDING MOMENTS - MX	MY	MXY	-TRANSVERSE SHEAR FORCES- QX	QY
3	2.10645E+02	-1.73879E+03	-2.46591E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
7	5.97852E+01	-1.69688E+03	1.03084E+01	2.03451E-07	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
13	-1.89788E+02	6.80536E+02	-4.45374E+01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

8.2.1.10. Shear Panel Output

The shear panel is an element which resists the action of tangential forces applied to its edges. In ASTROS, the shear panel supports only isotropic material properties and makes use of the shear flow distribution approximation of Garvey (Reference 4) with special handling for warped, parallel edge and general trapezoidal geometries. The element force sign convention is shown in Figure 8-11. The stresses, strains, forces and strain energies are available for the shear panel. The element forces that are computed include the following:

(1) The eight forces between each pair of nodes; each force is directed along the line connecting the adjacent nodes (the element edge).
(2) The four "kick" forces at each node, normal to the plane formed by the two adjacent element edges.
(3) The shear flows (forces/unit length) along each edge.

When stresses or strains are requested, they are computed at the node points in skewed coordinates parallel to the adjacent edges. Both the average and maximum shear stress or strain are then printed. A safety margin based on the maximum stress value is computed for stress output. A large safety margin is printed if not limits were specified on the material property entry. Table 8-15 contains a sample of the SHEAR panel element output.

The strain energy print for the SHEAR panel is identical to that for the BAR element and includes a breakdown by element and by element type.

8.2.2. Nodal Response Quantities

ASTROS has two basic forms of node point: the structural node and the extra point. The structural node is defined as either a "grid" point having 6 degrees of freedom (three translations and three rotations) or a "scalar" point having a single degree of freedom. These node points can be used to connect metric and scalar structural elements. The extra point is similar to the scalar point in that it has a single degree of freedom, but differs in that extra points included in the model are selected in the boundary condition rather than being implicitly included in the model. Further, they cannot be connected directly to either metric or scalar structural elements; instead, these elements are connected through terms introduced by direct matrix input or by transfer functions. Extra points are used in dynamic analyses for modeling control systems and other nonstructural mechanisms in the system under analysis. These degrees of freedom do not appear in the system matrices until after the dynamic matrix assembly and do not appear in any but the dynamic response disciplines (**FLUTTER**, **TRANSIENT** and **FREQUENCY**). When nodal output is requested for dynamic analyses, any extra point results may be selected using the **GRIDLIST** entry just as are grid and scalar point results.

Nodal output is available for all disciplines in ASTROS, although particular nodal response quantities may not be available for all disciplines. The solution control print options **VELOCITY**, **DISPLACEMENT**, **GPFORCE**, **LOAD**, **SPCFORCE**, and **ACCELERATION** are used to select print of the nodal response quantities. Each of these print options selects either **ALL**, **NONE** or an integer set identification number that refers to one or more **GRIDLIST** bulk data entries. Chapter 3 contains the complete description of the solution control print command. Each output is carefully labeled as to its boundary condition number, which discipline generated the output quantities and which load condition, mode shape, time step, frequency step or flight condition is represented by the output.

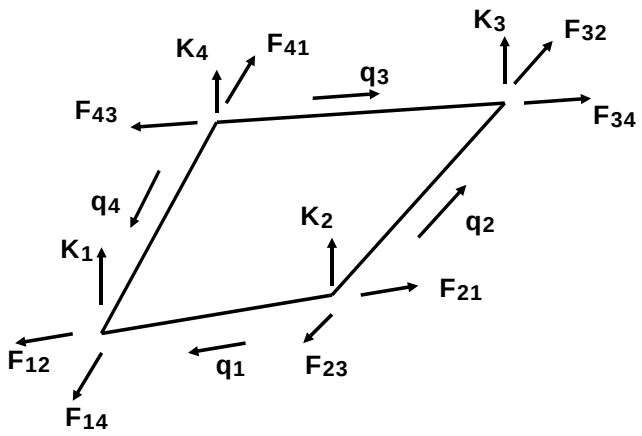


Figure 8-11. Shear Panel Forces

Table 8-15. SHEAR Solution Quantities

```
SIMPLIFIED FIGHTER WING                                                    ASTROS VERSION 9.0   03/03/93   P.   14
                                                                            FINAL ANALYSIS SEGMENT
                                                                  STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1
                              S T R E S S E S   I N   S H E A R   P A N E L S   ( S H E A R )
       ELEMENT        MAX           AVERAGE       SAFETY          ELEMENT        MAX           AVERAGE       SAFETY
         ID.         SHEAR            SHEAR        MARGIN            ID.         SHEAR            SHEAR        MARGIN
           17    4.728344E+03    3.414915E+03    4.9E+00              21    1.600469E+03    1.250366E+03    1.6E+01
           25    5.703891E+03    4.119477E+03    3.9E+00              29    2.716250E+02    2.716250E+02    1.0E+02
           32    7.276465E+02   -7.276465E+02    3.7E+01              36    3.413748E+03   -3.413748E+03    7.2E+00
SIMPLIFIED FIGHTER WING                                                    ASTROS VERSION 9.0   03/03/93   P.   16
                                                                            FINAL ANALYSIS SEGMENT
                                                                  STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1
                              S T R A I N S   I N   S H E A R   P A N E L S   ( S H E A R )
           ELEMENT        MAX           AVERAGE             ELEMENT        MAX           AVERAGE
             ID.         SHEAR            SHEAR               ID.         SHEAR            SHEAR
                  17    1.257538E-03    9.082221E-04              21    4.256566E-04    3.325442E-04
                  25    1.516992E-03    1.095605E-03              29    7.224069E-05    7.224069E-05
                  32    1.935230E-04   -1.935230E-04              36    9.079117E-04   -9.079117E-04
SIMPLIFIED FIGHTER WING                                                    ASTROS VERSION 9.0   03/03/93   P.   18
                                                                            FINAL ANALYSIS SEGMENT
                                                                  STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1
                                F O R C E S   I N   S H E A R   P A N E L S   ( S H E A R )

            ====== POINT 1 ======     ====== POINT 2 ======     ====== POINT 3 ======     ====== POINT 4 ======
  ELEMENT    F-FROM-4     F-FROM-2     F-FROM-1     F-FROM-3     F-FROM-2     F-FROM-4     F-FROM-3     F-FROM-1
    ID.              KICK-1      SHEAR-12      KICK2      SHEAR-23      KICK-3      SHEAR-34      KICK-4      SHEAR-41

       17  6.30889E+03  1.41850E+03 -1.41850E+03 -6.30889E+03  6.30889E+03  9.45669E+02 -9.45669E+02 -6.30889E+03
                 0.00000E+00  9.45669E+02  0.00000E+00  6.30446E+02  0.00000E+00  4.20297E+02  0.00000E+00  6.30446E+02

       32 -2.91059E+03 -6.54882E+02  6.54882E+02  2.91059E+03 -2.91059E+03 -6.54882E+02  6.54882E+02  2.91059E+03
                 0.00000E+00 -1.45529E+02  0.00000E+00 -1.45529E+02  0.00000E+00 -1.45529E+02  0.00000E+00 -1.45529E+02
```

The form of nodal output in ASTROS is very similar for all nodal output quantities; therefore, only general descriptions will be given rather than individually describing each response quantity in turn. In general, the nodal output includes the node point identification number (sorted by external identification number) and node type: (G)rid, (S)calar point or (E)xtra point. This is followed by either one or six quantities associated with the node point. The columns of the print are labeled Ti for the translations and Ri for the rotation where i = 1, 2 or 3. An example is shown in Table 8-16. Complex nodal quantities are generated by **FLUTTER** and **FREQUENCY** disciplines and can be printed in either polar coordinates or cartesian coordinates through the form option on the **PRINT** or **PUNCH** command. Cartesian print is the default. Complex quantities are printed using the same columns as real nodal data but use two lines of output. The first line contains either the real part or the magnitude and the second line either the imaginary part or the phase angle in degrees. An example of POLAR complex print is shown in Table8-17.

All structural disciplines generate **DISPLACEMENT** output except some **FLUTTER** analyses. Flutter mode shapes are generated only if a flutter condition occurs in the selected range of velocities and then only if the **FLUTTER** discipline occurs in the **ANALYZE** subpacket of the solution control. **VELOCITY**s are only available for **TRANSIENT** and **FREQUENCY** analyses. **ACCELERATION**s are available for **STATICS** with inertia relief, **SAERO**, **TRANSIENT** and **FREQUENCY** analyses.

The **LOAD** option selects output of externally applied loads at the nodal points. For **STATICS**, the applied mechanical, thermal and/or gravity loads are output for the selected nodes and subcases. Steady aeroelastic loads output prints, for each trim condition, those trimmed forces applied to the structure following transformation from the aerodynamic model. The **SAERO** "applied" load is the sum of the trimmed rigid loads and the flexible correction. Each component is stored independently in the relation **OGRIDLOD**, but

Table 8-16. Displacement Vector

TEN BAR TRUSS ASTROS VERSION 9.0 03/03/93 P. 7
FINAL ANALYSIS SEGMENT
FINAL STATIC ANALYSIS STATICS ANALYSIS: BOUNDARY 1, SUBCASE 1

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	2.82588E-01	-1.26504E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
2	G	-3.17412E-01	-1.31319E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
3	G	2.34438E-01	-5.58118E-01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
4	G	-2.45562E-01	-6.00705E-01	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
5	G	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
6	G	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

Table 8-17. Complex Displacement Vector

ASTROS VERSION 9.0 03/03/93 P. 8
FINAL ANALYSIS SEGMENT
FREQUENCY ANALYSIS: BOUNDARY 1, FREQ = 3.2179463E+00

C O M P L E X D I S P L A C E M E N T V E C T O R
P O L A R F O R M

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
7	G	0.00000E+00	0.00000E+00	1.17367E+01	0.00000E+00	5.63383E-01	0.00000E+00
		0.00000E+00	0.00000E+00	3.47650E+02	0.00000E+00	1.67894E+02	0.00000E+00

the **APPLIED** component is printed on request. **TRANSIENT** and **FREQUENCY** disciplines compute loads at user specified time or frequency steps which may be printed. The **FLUTTER** and **MODES** disciplines have no loads output.

The print of single point forces of constraint, the **PRINT SPCF** option, has been implemented in a computational sense with the **PRINT** or **PUNCH** request generating the **SPCFORCE** forces for all disciplines (including **SAERO**) and storing the terms in the relation **OGRIDLOD**. In **SAERO**, the "applied" load that is used is the sum of the trimmed rigid load and the flexible correction. The actual printing of the **SPCFORCES** to the output file is not available.

The **GPFORCE PRINT/PUNCH** request does not send data to the output file or to the punch file. Instead, a **PRINT** or **PUNCH** request will result in the storage of the data on the database. The relation used to store **GPFORCE** data is called **GPFDATA** and is loaded in module **EDR**. The format of the relational tuple is:

ATTRIBUTE	DESCRIPTION
NITER	Iteration Number
BC	Boundary Condition id
DISC	Discipline Type (as in CASE relation)
SUBCASE	Subcase number
EID	Element id
ETYPE	Element Type
CMPLX	Complex Flag (1=Real, 2=Complex)
SIL	Internal DOF number
FLAG	Type of DOF (6=GRID, 1=SCALAR)
RFORCE	Real Part of Forces
IFORCE	Imaginary Part of Forces

where **BC**, **NITER**, **DISC** and **SUBCASE** identify the ASTROS analysis; **EID** and **ETYPE** identify the element; **SIL** identifies which degree(s) of freedom these forces are associated with (obviously it is one of those attached to the element **EID/ETYPE**) and the forces are stored in **RFORCE** (and **IFORCE**) with scalar points using only word 1 of each array. Notice that there will be one entry for each grid/scalar for each element for each subcase for each discipline for each boundary condition for each iteration for which data are requested in Solution Control.

8.2.3. Design Variables and Design Constraints

There is an important distinction between global design variables and local design variables in ASTROS. A number of linking options relating global variables to local variables are provided and are described in

Section 2 of the Theoretical Manual. Briefly, the local variable is the physical element property (e.g., thickness or cross-sectional area) that is free to change in the design process while the global variables are the actual variables that are modified by the resizing module. The resultant physical variables are then computed based on the user's linking options and the current global design variable values.

The **GDESIGN** solution control print option allows the user to request that a set of the global design variables be printed at some set of iterations. The global variable print displays the user assigned design variable identification number, the current value, the minimum and maximum values allowed for the global variable, the sensitivity of the objective function to the design variable and the linking option used to relate it to local design variables. The linking options are:

(1) Unique Physical. The user has related the global variable to a single local variable through a **DESELM** entry.

(2) Linked Physical. The user has related the global variable to some number of local variables through a combination of **DESVARP** and **ELIST**/**PLIST** entries.

(3) Shape Function. The user has related this and possibly other global variables to some number of local variables through a combination of **DESVARS**/**SHAPE** entries.

The final item in the global design variable print is an eight character user label identifying the design variable. An example of this print for the initial iteration of the ten bar truss problem is shown in Table 8-18 along with the **LDESIGN** print output.

The **LDESIGN** solution control print option allows the user to request that some set of local design variables be printed at some set of iterations. The local design variables are, of course, element dependent. Each element type that has elements connected to global design variables is printed separately. The elements are identified by element identification number and, if appropriate, the layer number. The linking option used to connect them to the global variables is also shown. This print can be very helpful in checking the correctness of the design model. Following these general data are the element dependent local variable value and the allowable range that the primary value can take. Note that the BAR element links the moments of inertia to the cross-sectional area so all three "design variables" are shown but the area is the only independent variable. The local variable print accounts for all scalar factors that might appear in the design variable linking and therefore, indicates the true physical values represented by the current design. Finally, the two-dimensional elements include a print of the ratio of the current thickness to the minimum thickness. This additional item is included as a convenience to allow a quick computation of the number of composite plys represented by a particular design ***if*** the user inputs the ply thickness as the minimum thickness and ***if*** the element has composite material properties.

The solution control print option **DCONSTRAINT** selects that the active constraint summary print should include a table indicating which constraints are active, their current value, the constraint type and other identifying data connecting the constraints to a particular element, subcase and/or discipline. Table 8-18 shows the **DCONSTRAINT** print in addition to the default **ACTCON** summary. The identifying data for each constraint in the print includes the **TYPE COUNT**, which is a running count (by constraint type) of all active and inactive constraints. This allows the user to identify exactly which constraint is active; e.g., the fourth flutter constraint or the 3000th Von Mises stress constraint. Additionally, if the constraint is associated with a particular boundary condition, the associated boundary condition identification is shown. Similar connections are made for subcase and element dependent constraints. If the constraint is not boundary condition, subcase or element dependent, zeros, blanks or the string **N/A** will appear in the corresponding columns of the active constraint summary. The user is cautioned that the constraint

Table 8-18. Design Variable Values

```
1TEN BAR TRUSS                                                    ASTROS VERSION 9.0   03/03/93   P.    7
                                                                  ASTROS ITERATION 1
 STATIC ANALYSIS

                          A S T R O S   D E S I G N   V A R I A B L E   V A L U E S


   DESIGN       DESIGN          MINIMUM         MAXIMUM        OBJECTIVE          LINKING         LAYER     LAYER     USER
  VARIABLE     VARIABLE
    ID          VALUE            VALUE           VALUE         SENSITIVITY        OPTION          NUMBER    LIST      LABEL

        1    2.00000E+00     6.66700E-03     1.00000E+03     5.40000D+02  UNIQUE PHYSICAL         N/A       N/A       ROD1
        2    2.00000E+00     6.66700E-03     1.00000E+03     5.40000D+02  UNIQUE PHYSICAL         N/A       N/A       ROD2
        3    2.00000E+00     6.66700E-03     1.00000E+03     5.40000D+02  UNIQUE PHYSICAL         N/A       N/A       ROD3
        4    2.00000E+00     6.66700E-03     1.00000E+03     5.40000D+02  UNIQUE PHYSICAL         N/A       N/A       ROD4
        5    2.00000E+00     6.66700E-03     1.00000E+03     5.40000D+02  UNIQUE PHYSICAL         N/A       N/A       ROD5

1TEN BAR TRUSS                                                    ASTROS VERSION 9.0   03/03/93   P.    8
                                                                  ASTROS ITERATION 1
 STATIC ANALYSIS
            S U M M A R Y   O F   L O C A L   D E S I G N   V A R I A B L E S   --   I T E R A T I O N   1
                                        R O D   E L E M E N T S
                                EID       LINKING OPTION       AREA        MINIMUM     MAXIMUM
                                    1  UNIQUE PHYSICAL   3.00000000E+01   1.000E-01   1.500E+04
                                    2  UNIQUE PHYSICAL   3.00000000E+01   1.000E-01   1.500E+04
                                    3  UNIQUE PHYSICAL   3.00000000E+01   1.000E-01   1.500E+04
                                    4  UNIQUE PHYSICAL   3.00000000E+01   1.000E-01   1.500E+04
                                    5  UNIQUE PHYSICAL   3.00000000E+01   1.000E-01   1.500E+04
```

Table 8-19. Design Constraint Summary

```
TEN BAR TRUSS                                                     ASTROS VERSION 11.0  04/06/95   P.   56
                                                                  ASTROS ITERATION 13
STATIC ANALYSIS
                          S U M M A R Y   O F   A C T I V E   C O N S T R A I N T S
                                    AFTER ANALYSIS 13 OF A MAXIMUM 16
                                  18 CONSTRAINTS RETAINED OF 18 APPLIED

                      ****************************************************************
                      *         CONSTRAINT RETENTION ALGORITHM SUMMARY               *
                      *         RFAC =    3.000, EPS =    -.100, NDV =     10        *
                      *                                                              *
                      *  # OF CONSTRAINTS RETAINED BY RFAC     =        18           *
                      *       CUTOFF CONSTRAINT VALUE          =    -2.000           *
                      *                                                              *
                      *  # ADDED WITH VALUES GREATER THAN EPS  =         0           *
                      *                                                              *
                      *  # OF ADDITIONAL MINIMUM THICKNESS                           *
                      *    CONSTRAINTS RETAINED ONLY FOR                             *
                      *    CONTROLLING MOVE LIMITS (DCONTHK)   =         0           *
                      ****************************************************************

     COUNT   CONSTRAINT VALUE     CONSTRAINT TYPE     TYPE COUNT  BOUNDARY ID  SUBCASE  ELEMENT TYPE     EID/LAYR/DIMENSION

         1     -1.99999E+00     DISP/DCID = 1              1           1         1       N/A          N/A
         2     -1.99999E+00     DISP/DCID = 2              2           1         1       N/A          N/A
         3     -1.36685E+00     DISP/DCID = 3              3           1         1       N/A          N/A
         4     -1.73042E+00     DISP/DCID = 4              4           1         1       N/A          N/A
         5     -8.14951E-06     DISP/DCID = 5              5           1         1       N/A          N/A
         6     -6.63099E-06     DISP/DCID = 6              6           1         1       N/A          N/A
         7     -6.33147E-01     DISP/DCID = 7              7           1         1       N/A          N/A
         8     -2.69576E-01     DISP/DCID = 8              8           1         1       N/A          N/A
         9     -7.30348E-01     VON MISES STRESS           1           1         1     ROD            1
        10     -9.99997E-01     VON MISES STRESS           2           1         1     ROD            2
        11     -6.70627E-01     VON MISES STRESS           3           1         1     ROD            3
        12     -7.28571E-01     VON MISES STRESS           4           1         1     ROD            4
```

ordering in the active constraint summary is not necessarily the order that constraints appear in the sensitivity matrices, the **DESIGN** module or other discipline dependent output.

Finally, in interpreting the constraint values, the user must be aware of some features of ASTROS design constraints. The constraints in ASTROS are all formulated such that a value greater than zero represents a violated constraint. Also, all the constraints are normalized in some manner by the design allowable. The normalization has been formulated in such a way as to provide the best behavior under the linear approximations used in the approximate optimization problem but this has the effect of obscuring the physical meaning of the constraint. The user is referred to the Theoretical Manual for the exact form of each constraint.

8.2.4. Flutter/Normal Modes Response Quantities

The solution control print option **ROOTS** for flutter selects that the root extraction summary for flutter analyses be printed. In addition, if the flutter analyses appear in the **ANALYZE** subpacket of the solution control packet, the modal participation factors of any flutter conditions will be printed. The roots are ordered such that the lowest frequency root at each velocity is associated with the lowest frequency normal mode and so on in increasing frequency order. For each normal mode, the corresponding velocity value, damping ratio, frequency and complex eigenvalue are shown. For **OPTIMIZE** flutter analyses, only the user's input velocities are used in the root extraction algorithm. **ANALYZE** flutter analyses may generate additional velocities in the process of converging to a flutter crossing. Further, **OPTIMIZE** flutter analyses assume that constraints are imposed and print out the **TYPE COUNT** and **CONSTRAINT VALUE** as shown in Table 8-20. These columns do not appear for analysis cases.

The complex modal participation factors for each of the normal modes in the modal representation of the structure are printed if the **ROOTS PRINT** option is selected in **ANALYZE** flutter disciplines and a flutter crossing is found. A flutter crossing can occur for each Mach number and density ratio combination in the flutter analysis. Therefore, the flutter condition is identified by velocity, Mach number and density ratio to distinguish among multiple flutter conditions in the same analysis. An example is given in Table 8-21 in which the **INDEX** is the normal mode and **REAL/IMAG** are the complex factor. Note that a zero participation factor will be shown for normal modes that the user omitted from the flutter analysis.

Table 8-20. Flutter Solution Results

1 ASTROS VERSION 9.0 03/03/93 P. 20
ASTROS ITERATION 4

S U M M A R Y O F P - K F L U T T E R E V A L U A T I O N

MODE = 1 MACH NUMBER = .8000 DENSITY RATIO = 1.0000E+00

VEL NO.	TYPE COUNT	CONSTRAINT VALUE	VELOCITY EQUIVALENT	VELOCITY TRUE	DAMPING RATIO	FREQUENCY CYC/SEC	FREQUENCY RAD/SEC	COMPLEX EIGENVALUE REAL	COMPLEX EIGENVALUE IMAGINARY
1	1	-1.890E+00	1.01150E+04	1.01150E+04	-3.78041E-01	2.28032E+01	1.43276E+02	-6.42583E-02	3.39954E-01
2	7	-1.008E+01	1.51725E+04	1.51725E+04	-2.01653E+00	2.49806E+01	1.56958E+02	-2.50330E-01	2.48278E-01
3	13	-2.202E+00	1.71955E+04	1.71955E+04	-4.40401E-01	0.00000E+00	0.00000E+00	-1.52631E-01	5.78939E-08
4	19	-1.711E+00	1.82070E+04	1.82070E+04	-3.42128E-01	0.00000E+00	0.00000E+00	-1.18573E-01	2.59419E-11
5	25	-1.536E+00	1.87128E+04	1.87128E+04	-3.07225E-01	0.00000E+00	0.00000E+00	-1.06476E-01	1.89918E-14

MODE = 2 MACH NUMBER = .8000 DENSITY RATIO = 1.0000E+00

VEL NO.	TYPE COUNT	CONSTRAINT VALUE	VELOCITY EQUIVALENT	VELOCITY TRUE	DAMPING RATIO	FREQUENCY CYC/SEC	FREQUENCY RAD/SEC	COMPLEX EIGENVALUE REAL	COMPLEX EIGENVALUE IMAGINARY
1	2	-2.292E+00	1.01150E+04	1.01150E+04	-4.58386E-01	4.39390E+01	2.76077E+02	-1.50133E-01	6.55051E-01
2	8	-1.017E+00	1.51725E+04	1.51725E+04	-2.03466E-01	2.88941E+01	1.81547E+02	-2.92149E-02	2.87173E-01
3	14	-3.128E-01	1.71955E+04	1.71955E+04	-6.25583E-02	2.99157E+01	1.87966E+02	-8.20597E-03	2.62346E-01
4	20	-2.659E-02	1.82070E+04	1.82070E+04	-5.31809E-03	3.02772E+01	1.90237E+02	-6.66799E-04	2.50766E-01
5	26	1.055E-01	1.87128E+04	1.87128E+04	2.11023E-02	3.04440E+01	1.91285E+02	2.58853E-03	2.45332E-01

Table 8-21. Modal Participation Factors

```
                                                                    ASTROS VERSION 9.0   03/03/93   P.   31
                                                                     FINAL ANALYSIS SEGMENT
                                                                  MODES ANALYSIS: BOUNDARY 2


                              MODAL PARTICIPATION FACTORS FOR CRITICAL FLUTTER SPEED OF:

                                        MACH          =        .8000
                                        V(TRUE)       =   18306.8594
                                        V(EQ)         =   18306.8594
                                        DENSITY RATIO =     1.000000
                                        FREQUENCY     =    30.310659 HZ,   190.447495 RAD/S


   INDEX     REAL          IMAG        INDEX     REAL          IMAG        INDEX     REAL          IMAG

      1    9.8031E-01    0.0000E+00       2    6.4565E-02   -1.8495E-01       3   -1.5952E-02   -9.2147E-03
      4    1.1690E-02   -8.5903E-03       5    5.0619E-03   -2.7755E-03       6    5.2087E-03    8.6489E-04
```

The **ROOTS** print option for normal modes, illustrated in Table 8-22, selects that the eigenvalue extraction table be printed. It will appear immediately ahead of any eigenvectors, if any were selected. The table is patterned after that in NASTRAN and includes the eigenvalues (in sorted order), the extraction order, the cyclic and radian frequency and generalized mass and generalized stiffness for each eigenvector computed. The table is prefaced by data identifying the eigenvalue extraction method and some self-explanatory method dependent data.

8.2.5. Aeroelastic Trim Quantities

The TRIM solution control print option select that the aeroelastic trim parameters and stability coefficients be printed. There are two types of aeroelastic trim analyses in ASTROS: (1) **SYMMETRIC** and (2) **ANTISYMMETRIC**. The number of degrees of freedom **SUPORT**ed at the support point determine the number of trim degrees of freedom. **SYMMETRIC** analyses may have DOF's 1, 3 and/or 5 (thrust, lift, pitch) or any combination. **ANTISYMMETRIC** analyses may have 2, 4, and/or 6 (side-force, roll, yaw) or any

Table 8-22. Real Eigenanalysis Results

```
1INTERMEDIATE COMPLEXITY WING                                                ASTROS VERSION 9.0   03/03/93   P.   29
 QDMEM1 ELEMENTS WITH 153 DESIGN VARIABLES                                    FINAL ANALYSIS SEGMENT
 NORMAL MODES                                                              MODES ANALYSIS: BOUNDARY 2


                                    S U M M A R Y   O F   R E A L   E I G E N   A N A L Y S I S


                                58 EIGENVALUES AND 6 EIGENVECTORS EXTRACTED USING METHOD GIVENS

                                MAXIMUM OFF DIAGONAL MASS TERM IS 2.608506147E-15 AT ROW 5 AND COLUMN 4

                    MODE     EXTRACTION      EIGENVALUE              FREQUENCY                        GENERALIZED
                             ORDER           (RAD/S)**2      (RAD/S)          (HZ)          MASS          STIFFNESS

                       1         57        1.80997E+04     1.34535E+02     2.14119E+01     7.09235E-02     1.28370E+03
                       2         56        9.14126E+04     3.02345E+02     4.81197E+01     7.63528E-02     6.97961E+03
                       3         58        1.86623E+05     4.31999E+02     6.87547E+01     1.11454E-01     2.07998E+04
                       4         55        3.21398E+05     5.66920E+02     9.02281E+01     7.53867E-02     2.42291E+04
                       5         54        6.23876E+05     7.89858E+02     1.25710E+02     1.49350E-01     9.31757E+04
                       6         53        8.25231E+05     9.08422E+02     1.44580E+02     8.80090E-02     7.26278E+04
                       7         52        1.11603E+06     1.05642E+03     1.68135E+02     0.00000E+00     0.00000E+00
                       8         51        1.64091E+06     1.28098E+03     2.03875E+02     0.00000E+00     0.00000E+00
```

combination. The thrust DOF should never be free since ASTROS has no mechanism to input thrust and the drag computations based on potential aerodynamics are invariably poor. The code does not impose any restriction, however. Each TRIM print is labeled with the Mach number, dynamic pressure, reference grid point, and the appropriate normalization parameters. These parameters are the reference area and chord length for longitudinal coefficients and reference area and span for lateral coefficients.

The **SYMMETRIC** trim print includes, in the most general case, the drag, lift and pitching moment stability coefficients for:

$C_{D_{o,}}$ $C_{L_{o,}}$ C_{M_o} - Thickness and camber effects

C_{D_α}, C_{L_α}, C_{M_α}, - Angle of attack (α) in both radians and degrees

C_{D_δ}, C_{L_δ}, C_{M_δ} - User defined control surface deflection(s) (δ) (both radians and degrees)

C_{D_q}, C_{L_q}, C_{M_q} - Pitch rate (q) in both radians and degrees.

These nondimensional factors are implicitly defined in the following equations:

$$\text{Drag} = \bar{q}\, S \left[C_{D_o} + C_{D_\alpha}\, \alpha + C_{D_q}\, \frac{qc}{2V} + C_{D_{\delta_i}}\, \delta_i \right] \quad \text{for i} = 1, \ldots\ n_{SYM}$$

$$\text{Lift} = \bar{q}\, S \left[C_{L_o} + C_{L_\alpha}\, \alpha + C_{L_q}\, \frac{qc}{2V} + C_{L_{\delta_i}}\, \delta_i \right] \quad \text{for i} = 1, \ldots\ n_{SYM}$$

$$\text{Pitching Moment} = \bar{q}\, S\, c \left[C_{M_o} + C_{M_\alpha}\, \alpha + C_{M_q}\, \frac{qc}{2V} + C_{M_{\delta_i}}\, \delta_i \right] \quad \text{for i} = 1, \ldots\ n_{SYM}$$

where,

$\bar{q}$ = Dynamic Pressure

S = Reference Area

c = Reference Chord

V = Reference Velocity

n_{SYM} = The number of symmetric control surfaces

These definitions are the standard forms used in aircraft stability and control (see Reference 5).

Each of these three quantities (drag, lift and pitching moment coefficients) is shown in up to three forms (Table 8-23):

Table 8-23. Symmetric Trim Results

```
1SIMPLIFIED WING STRUCTURE DESIGN                                                    ASTROS VERSION 9.0   03/03/93   P.    9
 STRESS, DISP, LIFT AND AILERON EFFECTIVENESS CONSTRAINTS                            ASTROS ITERATION 1
 SYMMETRIC CONDITION

                              NONDIMENSIONAL LONGITUDINAL STABILITY DERIVATIVES
                        COMPUTED AT THE AERODYNAMIC REFERENCE GRID AND INCLUDING ANY CONTROL EFFECTIVENESS

                                 TRIM IDENTIFICATION   =      100          REFERENCE GRID   =       20
                                 REFERENCE AREA        =   2.4000E+03      REFERENCE CHORD  =   2.0000E+01


                                <<<        DRAG          >>> <<<         LIFT          >>> <<<    PITCHING MOMENT    >>>
                                  RIGID     RIGID   FLEXIBLE    RIGID     RIGID   FLEXIBLE    RIGID     RIGID   FLEXIBLE
     PARAMETER      LABEL        DIRECT    SPLINED              DIRECT    SPLINED             DIRECT    SPLINED
     ---------      -----        ------    -------  --------    ------    -------  --------   ------    -------  --------
  THICKNESS/CAMBER "THKCAM  "      -.0012    N/A       N/A        .0099     .0099     .0173     .0057     .0057     .0069

  ANGLE OF ATTACK  "ALPHA   " 1/DEG     .0010    N/A       N/A        .1173     .1173     .1928    -.0062    -.0062     .0080
  ANGLE OF ATTACK  "ALPHA   " 1/RAD     .0582    N/A       N/A       6.7223    6.7223   11.0489    -.3552    -.3552     .4573

  PITCH RATE       "QRATE   " S/DEG    -.0015    N/A       N/A        .0923     .0923     .1004    -.2034    -.2034    -.1999
  PITCH RATE       "QRATE   " S/RAD    -.0870    N/A       N/A       5.2878    5.2878    5.7517  -11.6513  -11.6513  -11.4535

  CONTROL SURFACE  "ELEV    " 1/DEG    -.0012    N/A       N/A        .0118     .0118     .0128    -.0431    -.0431    -.0420
  CONTROL SURFACE  "ELEV    " 1/RAD    -.0670    N/A       N/A        .6775     .6775     .7346   -2.4704   -2.4704   -2.4069
 ------------------------------------------------------------------------------------------------------------------------------
 COMPUTED DRAG VALUES ARE INCLUDED FOR COMPLETENESS AND MODEL CHECK-OUT ONLY
          USE CAUTION IN INTERPRETING THEIR PHYSICAL VALIDITY
 VALUES MARKED "N/A" CANNOT BE COMPUTED UNLESS THE CORRESPONDING DOF IS SUPPORTED
 ------------------------------------------------------------------------------------------------------------------------------


        TRIM RESULTS FOR TRIM SET   100 OF TYPE PITCH
        ---------------------------------------------

        MACH NUMBER       8.00000E-01
        DYNAMIC PRESSURE  6.50000E+00
        VELOCITY          9.86400E+03

        TRIM PARAMETERS:
               DEFINITION             LABEL          FLEXIBLE          RIGID
               ----------             -----          --------          -----
        LOAD FACTOR                 "NZ      "      3.09119E+03       3.09119E+03            (USER INPUT)
        PITCH RATE                  "QRATE   "      1.56990E+01       1.56990E+01  DEG/S     (USER INPUT)
        ANGLE OF ATTACK             "ALPHA   "      1.26089E+00       2.17052E+00  DEG       (COMPUTED)
        CONTROL SURFACE ROTATION    "ELEV    "     -2.11388E+00      -2.63485E+00  DEG       (COMPUTED)
        THICKNESS/CAMBER            "THKCAM  "      1.00000E+00       1.00000E+00            (USER INPUT)
```

(1) The stability derivative for the rigid aerodynamic model as computed directly from the forces acting on the aerodynamic boxes (termed **DIRECT** in the output). This output always appears since it comes directly from the aerodynamic model.

(2) The stability derivative for the rigid aerodynamic model as computed from the forces transformed to the structural degrees of freedom (termed **SPLINE**d in the output). This output only appears if the associated DOF is **SUPORT**ed.

(3) The flexible derivative which includes corrections for the flexibility and inertia relief effects. This output only appears if the associated DOF is **SUPORT**ed.

If the first two forms do not agree closely (within 1-2 percent), the spline transformation may be incorrect or some of the applied load is being reacted by model **SPC**s or **MPC**s before reaching the **SUPORT** points. This latter is the most common occurrence so that **SPCFORCE** and **GPFORCE** data are the first place to look to correct the problem. The **SPLINE**d and **FLEXIBLE** forms are used in the stability coefficient constraint calculations (**DCONALE**, **DCONCLA**, **DCONSCF**).

Finally, the trim parameters that were computed for the current flight condition are shown. In general, these are the angle of attack in degrees, the pitch rate in deg/s, and the **SYMMETRIC** control surface deflection angle(s) in degrees. In each case, the rigid and flexible "trim" state is shown (the rigid is informational only) and the parameter is labeled as **COMPUTED** if it was a free parameter in the trim analysis or **USER INPUT** if it was a fixed user input trim parameter. Only those parameters explicitly called out on the **TRIM** bulk data entry are listed.

The **ANTISYMMETRIC** trim print is similar except that the degrees of freedom that are available result in coefficients for side force, rolling moment and yawing moment.

$C_{Y_\beta}, C_{l_\beta}, C_{N_\beta}$ = Yaw angle (β), in both radians and degrees.

$C_{Y_r}, C_{l_r}, C_{N_r}$ = Yaw rate (r), in both radians and degrees.

$C_{Y_p}, C_{l_p}, C_{N_p}$ = Roll rate (p), in both radians and degrees.

$C_{Y_\delta}, C_{l_\delta}, C_{N_\delta}$ = User defined control surface deflection(s) (δ), in both radians and degrees.

These nondimensional factors are implicitly defined in the following equations:

Side Force $= \bar{q}S\left[C_{Y_\beta}\beta + C_{Y_r}\frac{rb}{2V} + C_{Y_p}\frac{pb}{2V} + C_{Y_{\delta_i}}\delta_i \right]$ for i = 1 ,..., n_{ANTI}

Roll Moment $= \bar{q}Sb\left[C_{l_\beta}\beta + C_{l_r}\frac{rb}{2V} + C_{l_p}\frac{pb}{2V} + C_{l_{\delta_i}}\delta_i \right]$ for i = 1 ,..., n_{ANTI}

Yaw Moment $= \bar{q}Sb\left[C_{N_\beta}\beta + C_{N_r}\frac{rb}{2V} + C_{N_p}\frac{pb}{2V} + C_{N_{\delta_i}}\delta_i \right]$ for i = 1 ,..., n_{ANTI}

where,

b = reference semispan

n_{ANTI} = The number of antisymmetric control surfaces

These quantities are shown in three forms as shown in Table 8-24:

(1) The stability derivative for the rigid aerodynamic model as computed directly from the forces acting on the aerodynamic boxes (termed **DIRECT** in the output). This output always appears since it comes directly from the aerodynamic model.

(2) The stability derivative for the rigid aerodynamic model as computed from the forces transformed to the structural degrees of freedom (termed **SPLINE**d in the output). This output only appears if the associated DOF is **SUPORT**ed.

(3) The flexible derivative which includes corrections for the flexibility and inertia relief effects. This output only appears if the associated DOF is **SUPORT**ed.

Table 8-24. Antisymmetric Trim Results

```
1SIMPLIFIED WING STRUCTURE DESIGN                                          ASTROS VERSION 9.0   03/03/93   P.   26
 STRESS, DISP, LIFT AND AILERON EFFECTIVENESS CONSTRAINTS                  ASTROS ITERATION 2
 ANITSYMMETRIC CONDITION

                                  NONDIMENSIONAL LATERAL STABILITY DERIVATIVES
                        COMPUTED AT THE AERODYNAMIC REFERENCE GRID AND INCLUDING ANY CONTROL EFFECTIVENESS

                                    TRIM IDENTIFICATION   =      200          REFERENCE GRID   =       20
                                    REFERENCE AREA        =   2.4000E+03      REFERENCE SPAN   =   6.0000E+01


                                  <<<      SIDE FORCE       >>> <<<    ROLLING MOMENT     >>> <<<     YAWING MOMENT     >>>
                                    RIGID     RIGID   FLEXIBLE    RIGID     RIGID   FLEXIBLE    RIGID     RIGID   FLEXIBLE
     PARAMETER      LABEL           DIRECT    SPLINED             DIRECT    SPLINED             DIRECT    SPLINED
     ---------      -----           ------    -------  --------   ------    -------  --------   ------    -------  --------
  YAW ANGLE        "BETA    " 1/DEG     .0000    N/A       N/A        .0000     .0000     .0000     .0000    N/A       N/A
  YAW ANGLE        "BETA    " 1/RAD     .0000    N/A       N/A        .0000     .0000     .0000     .0000    N/A       N/A

  YAW RATE         "RRATE   " S/DEG     .0000    N/A       N/A        .0000     .0000     .0000     .0000    N/A       N/A
  YAW RATE         "RRATE   " S/RAD     .0000    N/A       N/A        .0000     .0000     .0000     .0000    N/A       N/A

  ROLL RATE        "PRATE   " S/DEG     .0000    N/A       N/A       -.0418    -.0418    -.0510    -.0002    N/A       N/A
  ROLL RATE        "PRATE   " S/RAD     .0000    N/A       N/A      -2.3951   -2.3951   -2.9248    -.0112    N/A       N/A

  CONTROL SURFACE  "AILERON " 1/DEG     .0000    N/A       N/A        .0166     .0166     .0160     .0000    N/A       N/A
  CONTROL SURFACE  "AILERON " 1/RAD     .0000    N/A       N/A        .9508     .9508     .9191     .0018    N/A       N/A
 ------------------------------------------------------------------------------------------------------------------------------
 VALUES MARKED "N/A" CANNOT BE COMPUTED UNLESS THE CORRESPONDING DOF IS SUPPORTED
 ------------------------------------------------------------------------------------------------------------------------------


         TRIM RESULTS FOR TRIM SET   200 OF TYPE ROLL
         -----------------------------------------------

         MACH NUMBER       8.00000E-01
         DYNAMIC PRESSURE  6.50000E+00
         VELOCITY          9.86400E+03

         TRIM PARAMETERS:
                DEFINITION             LABEL           FLEXIBLE           RIGID
                ----------             -----           --------           -----
         CONTROL SURFACE ROTATION    "AILERON "      1.00000E+00       1.00000E+00  DEG        (USER INPUT)
         ROLL RATE                   "PRATE   "      1.03324E+02       1.30519E+02  DEG/S      (COMPUTED)
```

As in the longitudinal case, discrepancies between the data for the first two forms may indicate an error in the spline transformations or, more likely, the boundary conditions to allow the applied load to reach the **SUPORT** point unreacted. The effectiveness calculations are performed using the third form.

Finally, the trim parameters that were computed for the current flight condition are shown. In general, these are the yaw angle in degrees, the yaw rate in deg/s, the roll rate in deg/s, and the **ANTISYMMETRIC** control surface deflection angle(s) in degrees. In each case, the rigid and flexible "trim" state is shown (the rigid is informational only) and the parameter is labeled as **COMPUTED** if it was a free parameter in the trim analysis or **USER INPUT** if it was a fixed user input trim parameter. Only those parameters explicitly called out on the **TRIM** bulk data entry are listed.

8.3. SUMMARY OF SOLUTION RESULTS

Some of the solution results of ASTROS are written to the print file while others are placed on the CADDB database. In the latter case, they can be accessed using the ICE interactive program. Table 8-25

provides a summary of each quantity and indicates whether the data are printed or stored. In each case, if the **PUNCH** request results in storage of the data, the **PRINT** request also stores the data even if the actual printing of the data occurs.

Table 8-25. Summary of Output Quantities

QUANTITY	IF PRINT IS REQUESTED	IF PUNCH IS REQUESTED
ACCEL	**PRINT** File	Relation **OGRDDISP**
AIRDISP	Relation **OAGRDDSP**	Relation **OAGRDDSP**
BUCK	**PRINT** File	Relations **OPNLBUCK/OEULBUCK**
CGRAD	Relation **GRADIENT**	Relation **GRADIENT**
DCON	**PRINT** File	**Relation CONST**
DISP	**PRINT** File	**Relation OGRIDDSP**
ENERGY	**PRINT** File	Relation **EOxxxx**[1]
FORCE	**PRINT** File	**Relation EOxxxx**[1]
GDESIGN	**PRINT** File	**Relation GLBDES**
GPFORCE	Relation **GPFDATA**	Relation **GPFDATA**
GPWG	**PRINT** File	**Relation OGPWG**
KSNS	Unstructured **DKVI**	Unstructured **DKVI**
LDESIGN	**PRINT** File	**Relation OLOCALDV**
LOAD	Relation **OGRIDLOD**	Relation **OGRIDLOD**
MASS	Matrix **MGG**	Matrix **MGG**
MODEL		**PUNCH** File
MSNS	Unstructured **DMVI**	Unstructured **DMVI**
OGRADIENT	Relation **GRADIENT**	Relation **GRADIENT**
QHH	Matrix **QHHL/QHHLFL**	Matrix **QHHL/QHHLFL**
QHJ	Matrix **QHJL**	Matrix **QHJL**
ROOT	**PRINT** File	**Relations LAMBDA/CLAMBDA**
SPCF	Relation **OGRIDLOD**	Relation **OGRIDLOD**
STIFFNESS	Matrix **KGG**	Matrix **KGG**
STRAIN	**PRINT** File	**Relation EOxxxx**[1]
STRESS	**PRINT** File	**Relation EOxxxx**[1]
TPRESSURE	**PRINT** File	**Relation OAGRDLOD**
VELOCITY	**PRINT** File	**Relation OGRIDDSP**
TRIM	**PRINT** File	

1 - xxxx represents an element name: BAR, ELAS, HEX1, HEX2, HEX3, QDMM1, QUAD4, ROD, SHEAR, TRIA3 or TRMEM.

8.4. OTHER SELECTABLE QUANTITIES

The DEBUG packet has been used to control several low level outputs in a number of ASTROS modules. This subsection documents the outputs generated by those DEBUG parameters that relate to modifying the level or form of output from an ASTROS module.

8.4.1. Intermediate Steady Aerodynamic Matrix Output

The preface aerodynamic module, **STEADY**, has a selectable print-level DEBUGs called **STEADY**. These options will generate output from the **USSAERO** submodule of the preface aerodynamics modules. There are four print levels available:

PRINT	ACTION
1	Prints steady aerodynamic model geometry and a few miscellaneous debugs.
2	Prints the above and stability coefficient data.
3	Prints the above and pressure data from the **USOLVE** submodule.
4	Prints the above and voluminous data from the calculation of velocity components and intermediate matrices from the **USSAERO** submodule.

The user is cautioned that a print level of 4 generates a large amount of data. Most of these prints are vestigial prints from the **USSAERO** code that was adapted for use in the ASTROS system. In cases where the output data is not self-evident, the user is referred to the **USSAERO** documentation (Reference 6).

8.4.2. Intermediate Unsteady Aerodynamic Matrix Output

The secondary unsteady aerodynamic preface module, **AMP**, has an optional print DEBUG called **AMP** and an optional matrix argument as its last argument.

```
CALL AMP ([AJJTL],[D1JK],[D2JK],[SKJ],[QKKL],QKJL],[QJJL],[ajjdc]);
```

The **AMP** option controls the output of several intermediate matrices or of individual matrices from the matrix lists **QKKL**, **QKJL** and **QJJL** that are formed in **AMP** for **FLUTTER** and **GUST** analyses, respectively. The user is referred to the Programmer's Manual for complete documentation of these data base entities. The following matrices are output:

IF PRINT IS:	AND:	AND:	THEN:
1			The SKJ matrix.
	If there is only one aerodynamic group	If flutter entries are in the bulk data packet	The above and the matrix, [X], representing the solution to the equation: [AJJ] * [X] = ([D1JK] + (ik)[D2JK])
		If gust entries are in the bulk data packet	The above and the matrix [QKJ] from the corresponding matrix list.
>1		If flutter entries are in the bulk data packet	The above and the matrices [D1JK] and [D2JK].
			The above and the matrix [AJJT] after extraction from the corresponding matrix list.

The optional matrix **[AJJDC]** is used to store the intermediate matrix **[X]** described in the options shown above. If **[AJJDC]** is blank, a scratch data base entity is used to store **[X]**. In either case, **[X]** may be printed through the **AMP** option. Only the last **[X]** matrix calculated will be returned to the executive sequence in **[AJJDC]** for use in additional processing.

8.4.3. Flutter Root Iteration Output

The flutter analysis module, **FLUTTRAN**, has an optional DEBUG print control called **FLUTTRAN** to generate additional information on the flutter eigenvalue extraction:

The **FLUTTRAN** option in this case pertains to prints that give information on the iterative solution of the flutter matrices. It has the following meaning:

PRINT	ACTION
1	Print the number of iterations required to find each flutter root.
>1	Print the above plus information on each of the estimated roots for each iteration. This voluminous information may sometimes be of use, when the flutter solution goes astray, in determining if a modified set of velocities would give improved results.

8.4.4. Stress Constraint Computation Output

The stress/strain constraint evaluation module, **SCEVAL**, has an optional DEBUG parameter called **SCEVAL**. The **SCEVAL** argument, if non-zero, will generate a listing, by element type, of all the constrained elements, the current value of their stress components and the resultant constraint value for each design load condition. Also included in the print, is the running "type count" for stress and strain constraints that appears in the Active Constraint Summary print described in Subsection 5.2.3. This allows the user to identify exactly which elements and subcases are associated with each particular stress or strain constraint. This print is a remnant from the ASTROS development when the element stresses were not available, but it may still be useful in checking out the constraint modeling for large problems.

8.4.5. Intermediate Optimization Output

The **DESIGN** module for resizing via mathematical programming methods has a DEBUG option called **DESIGN** that selects a print of intermediate data:

The **DESIGN** debug value is passed directly to the MicroDOT optimization package which makes the following intermediate quantities available:

PRINT	ACTION
1	Initial design information and final results.
2	The above and function values at each iteration.
3	The above and internal MicroDOT parameters.
4	The above and search directions.
5	The above and gradient information.
6	The above and scaling information.
7	The above and one-dimensional search information.

These **DESIGN**/DEBUG options allow the user to view the detailed calculations used in the solution of the approximate constrained optimization problem that ASTROS generates at each iteration. The user is cautioned that the data printed from the **DESIGN** module are not necessarily ordered in the same manner as in other design prints and not identified by user supplied design variable identification numbers.

8.5. EXECUTIVE SEQUENCE OUTPUT UTILITIES

In recognition of the inability to provide for the print of all useful response quantities, utilities have been included in the set of MAPOL modules to augment the solution control print options. These utilities may be placed in any MAPOL program where the user desires to see additional information. In general, these utilities print the data contained in either general or specific data base entities. The formats of these prints are more general and therefore, less well identified than the special print options described in the preceding subsections. The generality of these utilities, however, is felt to be a vital addition to the output features of the ASTROS procedure in that almost any data on the user's data base files can be written to the output file. These utilities provide a primitive link between ASTROS and external post-processing systems. ICE now provides a very sophisticated link.

8.5.1. Structural Set Definition Print Utility, USETPRT

The **USETPRT** utility has been provided to print, for each boundary condition in the solution control packet, the structural set definition table stored in the ASTROS data base entity, **USET**. This utility exactly mimics the capabilities provided by the NASTRAN **PARAM/USETPRT** option. The **USETPRT** module has the following calling sequence:

```
CALL USETPRT ( USET(BC), BGPDT(BC) );
```

For the selected boundary condition, **BC**, each degree of freedom in the structural model is listed in a table which shows the structural sets to which the degree of freedom belongs. The reader is referred to Section 4 of the Theoretical Manual for more information on the structural set definitions in ASTROS.

8.5.2. Special Matrix Print Utility, UTGPRT

The print utility, **UTGPRT**, has been provided in order to view particular matrices whose rows correspond to the structural degrees of freedom. In general, these matrices are very large and virtually impossible to interpret without some additional formatting beyond that which is available for more general matrix prints. Therefore, for the supported matrix entities, the matrix columns are printed in a form similar to that used for the nodal response quantities as presented in Section 5.2.2. The **UTGPRT** utility has the following calling sequence:

```
CALL UTGPRT ( BC, USET(BC), [mat1], [mat2], ... [mat10] );
```

where up to ten matrix arguments may be supplied. The **BC** integer argument and the name of the **USET** entity for the **BC**'th boundary condition identifies the associated boundary condition so that the utility can make use of the **USET** entity in formatting the output. The following entities are supported:

```
[DKUG], [DMUG], [DPVJ], [DUG], [DPGV], [DUGV]
[DPTHVI], [DPGRVI], [PG], [DFDU]
```

The utility keys off the entity names, so the above names must be used, although the matrices can be subscripted if the user wishes. The reader is referred to the Programmer's Manual for additional information on the data contained in these entities.

8.5.3. General Matrix Print Utility, UTMPRT

The matrix print utility, **UTMPRT**, has been written such that any data base matrix entity can be printed to the output file. The calling sequence for **UTMPRT** is:

```
CALL UTMPRT ( method, [mat1], [mat2], ... [mat10] );
```

where up to ten matrices can be printed in a single call. The optional integer **METHOD** argument selects from among two formats that are available. If **METHOD** is zero or absent, the entire matrix column, starting with the first non-zero term and ending with the last non-zero term, will be printed, including all intermediate zeros. If **METHOD** is non-zero, only the non-zero terms of each column will be printed.

8.5.4. General Relation Print Utility, UTRPRT

The print utility, **UTRPRT**, has been written such that any data base relational entity can be printed to the output file. The calling sequence for **UTRPRT** is:

```
CALL UTRPRT ( rel1, rel2, ... rel10 );
```

where up to ten relations can be printed in a single call. Relational entities are tables in which the columns are called attributes. The **UTRPRT** utility attempts to print the relation in a format in which each

column represents one attribute and each row represents a single entry in the relation. The utility is not very sophisticated, however, and relations having more attributes than can fit in the width of a page (128 characters or approximately 12 attributes) will have the trailing attributes ignored. Also, string attributes are only printed if they are eight characters long. Despite its limitations, the **UTRPRT** utility can be very useful in viewing ASTROS data base relations.

8.5.5. General Unstructured Print Utility, UTUPRT

The print utility, **UTUPRT**, has been written such that any data base unstructured entity can be printed to the output file. The calling sequence for **UTUPRT** is:

```
CALL UTUPRT ( UNSTRUCT, type );
```

Unlike other data base entities, there is no information in an unstructured entity to identify what type of data is stored in its records. The user, therefore, must supply the **TYPE** argument to select the proper format to use in the print. The following **TYPE**'s are available:

TYPE	<0	prints only the record length (in single precision words) of each record in the entity
TYPE	=0	prints each record using an integer format
TYPE	=1	prints each record using a real single precision format
TYPE	=2	prints each record using a double precision format.

For **TYPE** values greater than or equal to zero, each record will be printed, in its entirety, in the selected format. If, as is typical, the record contains mixed data, e.g., both integer and real data, the user can make multiple calls to **UTUPRT** to view first the integer format and then the real format. No errors will occur, but the real data will give spurious looking integer prints and vice versa.

8.6. THE eSHELL INTERACTIVE PROGRAM

A code like ASTROS is so general that to make all possible response quantities available would take decades of effort, if it could be done at all. In fact, a mature finite element code like NASTRAN is continuously adding new output capabilities as the user community dictates. Naturally, a relatively new code like ASTROS cannot attempt to address all these features. Instead, the ASTROS designers considered it important to focus on design optimization and provide a large, but finite, number of options for post-processing outside this area. To support a powerful and general purpose ability to query solution results, UAI provides the program called eSHELL. This special program allows users to access any ASTROS data, to view it interactively and to generate files which can be moved from one computer to another or used as input to other application programs.

Chapter 9.

MAPOL PROGRAMMING

This chapter contains the programmer's manual for the ASTROS executive language, MAPOL. It presents the syntax and features of the MAPOL language and it contains the general information needed to make syntactically correct modifications to the ASTROS standard MAPOL sequence and to write independent MAPOL programs to direct the ASTROS system. All variable types, statement forms, input/output features and intrinsic functions are presented.

9.1. INTRODUCTION AND USER OPTIONS

The Matrix Analysis Problem Oriented Language (MAPOL) is a high level computer language that has been designed to support the large-scale matrix operations typically encountered in engineering analysis. Its conceptual roots may be traced to the Direct Matrix Abstraction Program (DMAP) capability found in the NASTRAN® system developed by NASA in the late 1960's. The DMAP *language* used to create NASTRAN's solution algorithms is very crude; however, it has been a prime factor in extending the life cycle of the system. It has done this by providing a simple method of installing new code and functional capabilities into the system. It also affords the user an opportunity to interact with the software.

MAPOL has been selected to provide the same advantages to the ASTROS system. Additionally, it extends the primitive DMAP design by assimilating the many advances that were made in computer science over the intervening two decades. MAPOL is a structured procedural language that directly supports high-order matrix operations, the manipulation of database entities and complex data types. The syntax of the language is similar to PASCAL, and it should be easily learned by anyone familiar with Fortran or PASCAL. This Chapter details all of the features of the MAPOL language and gives examples of their use.

9.1.1. USER OPTIONS

In this section, the different kinds of MAPOL programs and their uses are discussed. MAPOL is the control language of the ASTROS system and, as such, the multidisciplinary solution algorithm is simply a MAPOL program that is embedded in the system. the user is free to modify this standard algorithm and can also create individual MAPOL programs or specialized procedures.

9.1.2. MAPOL PROGRAM FORM

If an ASTROS analysis is not using the standard solution, then a MAPOL program is required as the first part of the input data stream. The MAPOL data packet must be formed as shown:

```
MAPOL [<option-list>] ;
...
...
...
END;
```

As introduced in Chapter 1, all bold capitalized words (e.g., **MAPOL**) must appear exactly as they are written. A symbol enclosed in angle brackets (e.g., <option-list>) represents one or more choices to be made. If the symbol is enclosed in square brackets (e.g., [<id>]), the choice is optional.

The MAPOL command, which must be the first statement in the program, selects compiler options. These options are shown in Table 9-1 where the default option options are indicated by boldface.

Table 9-1. MAPOL Command Options

NAME	OPTION
LIST NOLIST	Lists the MAPOL source program
GO NOGO	Selects, or deselects, execution after program compilation.

As an example, the statement:

```
MAPOL NOLIST;
```

will cause the MAPOL program to be compiled and executed with no listings produced, while the statement:

```
MAPOL NOGO;
```

will cause the MAPOL program to be compiled and a listing of the source code produced. After compilation, ASTROS will terminate without executing the program.

9.1.3. THE STANDARD ASTROS SOLUTION

As mentioned earlier, the ASTROS multidiscpliniary solution algorithm is a MAPOL program. The code resides on the ASTROS system database. It is retrieved and used whenever a MAPOL program is not found in the input data stream. While Chapter 2 provides a complete listing of the standard MAPOL algorithm for ASTROS, the actual program changes with each release of the system. Because of this, it is recommended that the user request the current listing if it is needed. This may be done by executing the ASTROS system generation program, SYSGEN. This program provides a listing of the standard solution algorithm as part of its output.

9.1.4. MODIFYING THE STANDARD SOLUTION

In some cases, the user may wish to modify the standard ASTROS solution in order to, for example, perform some auxiliary computations not currently available or to execute only a portion of the solution. Special MAPOL editing commands allow for these modifications:

```
DELETE   a [  ,b]
REPLACE  a [  ,b]
INSERT   a
```

DELETE is used to remove one or more statements starting with line "a" and, optionally ending with line "b" inclusively. REPLACE performs a deletion of the specified line, or lines, and replaces them with any following MAPOL statements. The INSERT command allows any number of MAPOL statements to be inserted after line "a".

For example:

```
EDIT;
INSERT 1
$ MY MODIFICATION $
REPLACE 20,23
A := 2 * B;
DELETE 101,237
```

Note that rather than entering the **MAPOL** command, the special **EDIT** declaration is used.

In the example, a comment is added at the beginning of the algorithm to document the modification. Several lines (20-23) are replaced by a new computational expression, and a larger block of lines (101-237) is removed from the program.

9.1.5. CREATING MAPOL PROGRAMS

If the standard executive sequence is not selected, the MAPOL compiler assumes that a new program is being created. This new program may perform any operations that use any of the matrix and database utilities available in the ASTROS system. All of these are described in subsequent chapters of this manual.

Table 9-2. Summary of MAPOL User Options

STATEMENT	FUNCTION
MAPOL [<option-list>]	Begins a MAPOL program and selects its name and compiler options
END	Terminates the MAPOL program
EDIT [<option-list>]	Modifies the standard solution sequence
DELETE a [,b]	Removes line **a** or lines **a** through **b** when editing
REPLACE a [,b]	Removes old line **a** or lines **a** through **b** and inserts new ones when editing
INSERT a	Inserts new lines after **a** when editing

9.1.6. SUMMARY

Table 9-2 summarizes the MAPOL statements that have been described in this section, along with their uses.

9.2. DATA TYPES AND DECLARATIONS

This section describes the data types that are available in the MAPOL language. It discusses their specifications during programming and how they are represented in the ASTROS machine.

9.2.1. DEFINITIONS AND NOTATION

All programming languages are composed of two kinds of symbols. The first kind of symbol is an explicit part of the language. In MAPOL, such symbols include special characters such as:

```
=  -  *  :=  ;
```

and "reserved words" such as:

```
REAL, RELATION, IF, ELSE, WHILE
```

In this Chapter, reserved words are indicated by bold capitalized names.

The second kind of symbol is an identifier, or variable name, which may be chosen by the programmer. Identifiers are composed of letters and digits, but the first character must always be a letter. This and other definitions in this manual are shown as:

```
<ident> := <letter> | <ident>  <letter> | <ident> <digit>
```

The vertical line "|" is read as "or". This definition clearly specifies all possible legal identifiers, because no matter how many times the rules:

```
<ident> := <ident> <letter>
  or
<letter> := <ident> <digit>
```

are used, the user must finally use the rule:

```
<ident> := <letter>
```

This final rule ensures that the identifier begins with a <letter>. In MAPOL <letter> refers to any of the upper case letters from **A** to **Z**, and digit to the integers from **0** to **9**.

Note that although this open-ended definition of an identifier, which is called recursive, implies that arbitrarily long names may be used, the MAPOL compiler has an implementation limit of eight characters for a variable name. However, for subscripted database entities, the implementation limit is five characters — the subscipt is later appended to this basic name. If such a name is too long to accomodate a subscript, then characters are truncated on the right with warning.

9.2.2. COMMENTARY

Commentary may be included in the MAPOL program by enclosing the text between two dollar signs ($). Comments may be one or more complete lines, or they may be embedded in a line as shown below:

```
   A := 2;
$  SET B TO 4 $
   B := 4;
$  THIS IS A MULTI
   LINE COMMENT
   THAT SHOWS HOW IT MUST ONLY START
   AND END WITH DOLLAR $
   C := A * B; $ THIS IS AN INTERLINE COMMENT $
```

9.2.3. SIMPLE DATA TYPES

The MAPOL language supports five simple data types:

- **INTEGER**
- **REAL**
- **COMPLEX**
- **LOGICAL**
- **LABEL**

MAPOL is a strongly typed language, and as such, all variables must be declared at the beginning of a program unit. This is done with one or more declaration statements. The syntax of a declaration statement is defined by the rules shown below:

```
<decl>     := <type> <var-list>
<type>     := REAL | INTEGER | COMPLEX | LOGICAL | LABEL
<var-list> := <var> | <var>, <var-list>
```

Each simple variable, with the exception of **LABEL**, may be an array with one subscript. This is defined by:

```
<var>  := | <ident> (<sub1>) | <ident> (<sub1>)
<sub1> := INTEGER
```

9.2.3.1. Data Type INTEGER

INTEGERs are whole numbers such as 157, 83, or 22. An **INTEGER** may also have a sign associated with it such as -47 or +1024. The range of integers depends upon the ASTROS host computer.

9.2.3.2. Data Type REAL

REAL data represents floating point numbers. Such numbers include 1.75, 0.00025, 1.78E-6 and -3.00271E+36. **REAL** numbers are represented in a manner determined by the ***machine precision*** of the

host computer automatically. MAPOL, therefore, does not distinguish between the **REAL** and **DOUBLE PRECISION** types such as is found in Fortran.

9.2.3.3. Data Type COMPLEX

COMPLEX numbers are those which may be represented in the form:

```
a+bi
```

Because some host computers automatically handle **COMPLEX** data while others do not, MAPOL and ASTROS handle such data in a manner totally independent of the host computer. In the ASTROS machine, both **a** and **b** are represented as a pair of machine precision floating point numbers. Most available mathematical functions operate on **COMPLEX** data.

9.2.3.4. Data Type LOGICAL

LOGICAL variables have a value of ***true*** or ***false***. The ASTROS machine represents true by the Fortran **.TRUE.** and false by the Fortran value **.FALSE.** Logical constants may only be used in assignment statements.

9.2.3.5. Data Type LABEL

LABELs are used to define statement locations within a MAPOL program. Typically, they are only used with the **GOTO** statement (see Section 9.4).

9.2.4. COMPLEX DATA TYPES

To best support comprehensive engineering analysis capabilities, MAPOL supports five complex, or high level data types:

- **MATRIX**
- **IMATRIX**
- **RELATION**
- **UNSTRUCT**
- **IUNSTRUCT**

All of these types represent database entities. Matrices and unstructured entities may be handled only in their entirety in MAPOL. Relations may be accessed on an entry-by-entry and attribute-by-attribute basis. Use of the **IMATRIX** and **IUNSTRUCT**, ("**I**" for indexed) data types allows for more efficient retrieval of data that are accessed in a random order.

9.2.4.1. Data Types MATRIX and IMATRIX

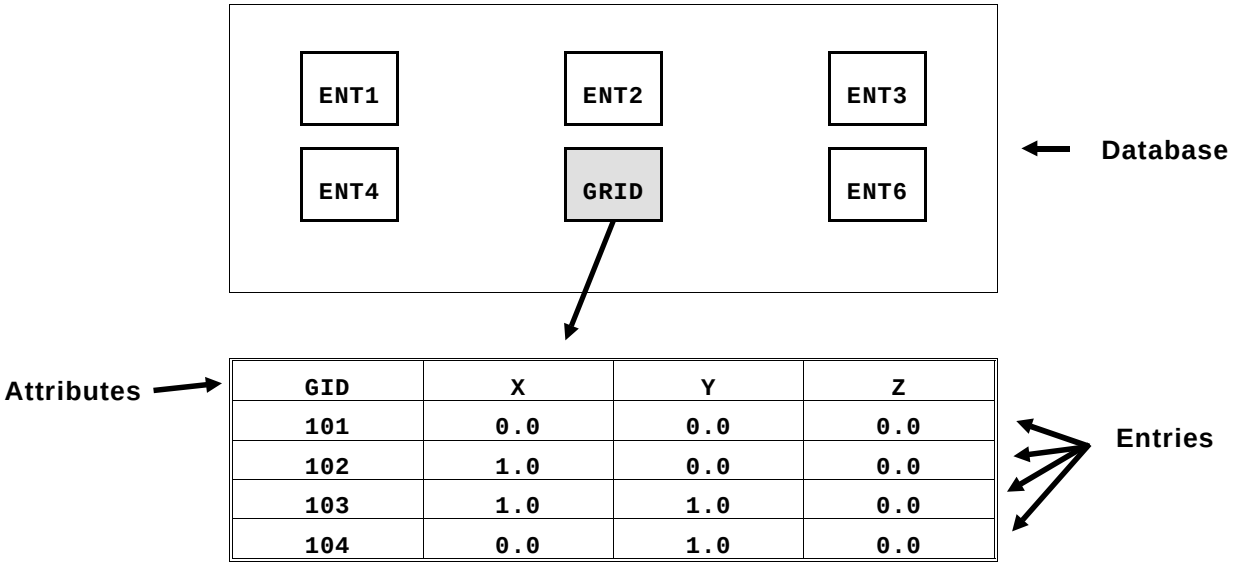
Matrix database entities are declared in a slightly different manner from the remaining data types. The rules for their declaration are:

```
<decl>     := MATRIX <mat-list>
<mat-list> := <mat-list> , <mat-var> | <mat-var>
<mat-var>  := [<ident>] | [<ident> (sub1 , sub 2)]
<sub1>     := INTEGER
<sub2>     := INTEGER
```

Note that the matrix `<ident>` is enclosed in square brackets (i.e., **[ ]**) for clarity and ease-of reading of MAPOL programs. Matrix expressions, then, look as they do written in standard mathematical notation. Matrix variables may also be subscripted to allow multiple entities to be referenced using the same `<ident>` . This feature is used in ASTROS to allow data from multiple boundary conditions to be saved for subsequent evaluation. There is an implementation limit of two subscripts, each of which may take on any integer value from 1 to 1000. However, no more than 1000 entities may result from this declaration. When subscripted matrix entities are used, the executive system generates a CADDB entity name and relates that name to the subscript value. The MAPOL programmer is therefore cautioned that, unlike other high-order variables, subscripted variables, and subscripted matrix entities do not have a corresponding CADDB entity of the same name. Due to the nature of the name generation algorithm, subscripted entity names must be unique in their first five characters.

9.2.4.2. Data Type Relation

The most complex and powerful MAPOL data type is the **RELATION**. Briefly, a relation can be thought of simply as a table. The rows of the table are called ***entries*** and the columns ***attributes*** The CADDB is a collection of such relations as shown in Figure 9-1. In the figure, a single relation, called **GRID**, has been highlighted. The **GRID** relation has four attributes: an identification number, **GID**, and three spatial coordinates (X, Y, and Z). The formats, or ***schemas***, of relations that reside on CADDB are fixed.



GID	X	Y	Z
101	0.0	0.0	0.0
102	1.0	0.0	0.0
103	1.0	1.0	0.0
104	0.0	1.0	0.0

Figure 9-1. Schematic Representation of Relation

All of the relations that are generated by the ASTROS modules that appear in the MAPOL program must be declared. The rules for these declarations are:

```
<decl>   := RELATION <rel-list>
<rel-list>:= <rel-list> , <rel-var> | <rel-var>
<rel-var>    := <ident>
```

If the user wishes to use the individual attributes of a relation, or to define a new relation, the **PROJECT** declaration is used:

```
<decl>   := PROJECT <rel-var> USING <att-list>
<rel-var>    := <ident>
<att-list>:= <att-list> , <attname> | <attname>
<attname>    := <ident>
```

The names of each of the attributes, <attname>, must match those defined in the CADDB schema if the relation already exists; otherwise, they are used to define the schema for the new relation. Note that in MAPOL, the attribute names cannot be shared among relations. This is the pure relational model which is not enforced within ASTROS itself.

As an example, the **GRID** relation of Figure 9-1 would be:

```
INTEGER GID;
REAL X,Y,Z;
PROJECT GRID USING GID,X,Y,Z;
```

Note that each attribute must be declared and be of the appropriate type.

Once a relation and its projection have been declared, specific entries may be retrieved. After a retrieval, any or all of the relation's attributes may be used directly by variables of the form:

```
<relname> . <attname>
```

This is illustrated in the following program segment:

```
INTEGER GID,ID;
REAL X,Y,Z;
REAL C1,C2,C3;
PROJECT GRID USING GID,X,Y,Z;
...
...
ID := GRID.GID;
C1 := GRID.X;
C2 := GRID.Y;
C3 := GRID.Z;
...
```

The value of an attribute within a relation may be modified if an assignment is made and then the entry is written onto CADDB (refer to Chapter 9.8).

9.2.4.3. Data Types UNSTRUCT and IUNSTRUCT

The simplest CADDB data structure is called an **UNSTRUCT**ured entity. The form and content of such an entity is the responsibility of the ASTROS programmer. The only use of the **UNSTRUCT** entity is for inter-module communications: **UNSTRUCT** entities, which may not be subscripted, are declared with:

```
<decl>    := UNSTRUCT <un-list>
<un-list> := <un-list> , <un-var> | <un-var>
<un-var>  := <ident>
```

9.2.4.4. Data Base Entity Declaration Requirements

All of the ASTROS database entities may be divided into three classes: (1) MAPOL entities, (2) HIDDEN entities, and (3) TEMPORARY entities. MAPOL entities are those that are used and appear in the MAPOL program such as matrices or relations used in calculations and any entity appearing as an argument in a functional module call. HIDDEN entities represent data that are used by a functional module but whose contents are generated from required physical data. As an example, the **GRID** Bulk Data are stored in a relation called **GRID**. Many modules might wish to access this **GRID** data. Requiring

the **GRID** relation to appear in the calling list of each such module is more disruptive than it is beneficial. As a result, **GRID** might never explicitly appear in the MAPOL program. It must, however, be declared so that the CADDB will be properly initialized. The last entity type, the TEMPORARY entity, is used mostly as a "scratch" area for intra-module use. As such it is created and deleted by the module needing it. In summary, all of the MAPOL and HIDDEN entities ***must*** be declared in the MAPOL program. TEMPORARY entities are not declared.

9.3. EXPRESSIONS AND ASSIGNMENTS

In this section, the relationships between the various data types are described. Of particular importance is the manner in which data are combined by arithmetic expressions and how values are assigned to the ASTROS machine memory.

9.3.1. ARITHMETIC EXPRESSIONS

Arithmetic expressions are formulae for computing numeric values. An arithmetic expression consists of either a single operand or two or more operands separated by arithmetic operators.

9.3.1.1. Arithmetic Operators

MAPOL supports five arithmetic operators as shown in Table 9-3. Successive operands must be separated by operators, and two operators may not be used in succession.

9.3.1.2. Arithmetic Operands

Arithmetic operands may be constants, symbolic names of constants, variables, (including relational attributes), array elements, or function references. Operands may also be arithmetic expressions and arithmetic expressions enclosed in parentheses. The data type of an arithmetic operand may be **INTEGER**, **REAL** or **COMPLEX**. In some cases, it may also be **MATRIX** or **IMATRIX**. It may never be **LOGICAL**, **LABEL**, **RELATION** (without an attribute specification), **UNSTRUCT**, or **IUNSTRUCT**.

Table 9-3. MAPOL Arithmetic Operators

OPERATOR	DESCRIPTION
+	Addition when connecting two operands. Unary plus when preceding an operand.
-	Subtraction when connecting two operands. Negation when preceding an operand.
*	Multiplication
/	Division
**	Exponentiation

9.3.1.3. Evaluation of Arithmetic Expressions

Expressions are evaluated from left to right according to the following hierarchy of operations:

PRIORITY	OPERATOR
1	**FUNCTION** Evaluation
2	**
3	* and /
4	+ and -

This hierarchy is used to determine which of two sequential operations is to be performed first. If two sequential operations are of unequal rank, the higher ranking operation is performed first. When a unary minus or plus appears in an arithmetic expression, it follows the same hierarchy as a minus or plus used for subtraction or addition. For example:

```
R = -S**T is evaluated as R = -(S**T)
R = -S/T is evaluated as R =  - (S/T)
R = -S+T is evaluated as R = (-S)+T
```

The division of operands in an expression may result in a truncated value for integer operands or a fractional value for non integer operands. Therefore, parentheses should be used when a specific order of evaluation other than left to right is desired for the operands. For example, the expression **8*7/4** has a resultant value of **14**; the expression **8*(7/4)** has a resultant value of **8**; the resultant value of the expression **3.0/(2.0*6.0)** is **0.25**.

9.3.1.4. The Uses of Parentheses

Parentheses may be used in arithmetic expressions to specify the order of operation. This allows an evaluation that is different from the standard hierarchy. Whenever parentheses are used, the enclosed expression is evaluated prior to its use. When such expressions are nested, the innermost expressions are evaluated first. The expression

```
X := A - SQRT(B) / (C-D) * E**2 * (F-G);
```

is therefore evaluated in the following order:

```
SQRT(B)      → TEMP1
(C-D)        → TEMP2
TEMP1/TEMP2  → TEMP3
E ** 2       → TEMP4
TEMP3*TEMP4  → TEMP5
(F-G)        → TEMP6
TEMP5*TEMP6  → TEMP7
A - TEMP7    → X
```

Table 9-4. MAPOL Operation Rules

FOR:	TYPE OF X	INTEGER	REAL	COMPLEX
Binary Operators X op Y	INTEGER	INTEGER	REAL	COMPLEX
	REAL	REAL	REAL	COMPLEX
	COMPLEX	COMPLEX	COMPLEX	COMPLEX
Exponentiation X ** Y	INTEGER	INTEGER	REAL	ILLEGAL
	REAL	REAL	REAL	ILLEGAL
	COMPLEX	COMPLEX	ILLEGAL	ILLEGAL

9.3.1.5. Type and Value of Arithmetic Expressions

Type conversions are performed when mixed expressions are evaluated. The final value of an arithmetic expression may depend upon this type conversion. Table 9-4 shows the conversions that occur when two operands are combined with an arithmetic operator:

Special rules apply to operations when one or more of the operations is of the type **MATRIX**. These rules are discussed in Section 9.5.

9.3.2. LOGICAL EXPRESSIONS

A logical expression produces a logical data type result with a value of **TRUE** or **FALSE**.

9.3.2.1. Logical Operators

Table 9-5 lists the logical operators that may be used in logical expressions.

Table 9-5. MAPOL Logical Operators

OPERATOR	DESCRIPTION	OPERATOR	DESCRIPTION
NOT	Negation (Unary)	OR	Disjunction
AND	Conjunction	XOR	Equivalence

Logical operators must be separated by logical operands except for the following cases:

```
AND NOT
OR NOT
```

9.3.2.2. Logical Operands

Any of the following operands may be used in logical expressions:

- LOGICAL CONSTANTS
- LOGICAL VARIABLES
- LOGICAL ARRAY ELEMENTS
- LOGICAL FUNCTION REFERENCE
- LOGICAL EXPRESSION
- RELATIONAL EXPRESSION

Both logical and relational expressions may be enclosed on parentheses

9.3.2.3. Evaluation of Logical Expressions

Logical expressions are evaluated based on "truth tables" shown in Table 9-6. **L1** and **L2** are logical variables, **T** and **F** signify **TRUE** and **FALSE**:

Table 9-6. Evaluation of MAPOL Logical Expressions

VARIABLES		RESULT			
L1	L2	NOT L1	L1 OR L2	L1 AND L2	L1 XOR L2
T	T	F	T	T	F
T	F	F	T	F	T
F	T	T	T	F	T
F	F	T	F	F	F

Logical operators have a hierarchy similar to the arithmetic operations:

PRIORITY	OPERATOR
1	Logical **FUNCTION**
2	**NOT**
3	**AND**
4	**OR**
5	**XOR**

Any operation in a logical expression may be enclosed in parentheses; the parenthetical expression is evaluated, and the resulting value is used as an operand. Thus, parentheses may be used to alter the order in which operations are to be performed. When parenthetical expressions are nested, evaluation begins with the innermost set of parentheses and proceeds to the outermost set.

9.3.3. RELATIONAL EXPRESSIONS

A relational expression uses relational operators to compare two arithmetic expressions. A relational expression produces a logical data type with a value of **TRUE** or **FALSE**. Thus, a relational expression may be an operand in a logical expression.

9.3.3.1. Relational Operators

Table 9-7 summarizes the relational operators available in MAPOL.

Table 9-7. Relational Operators in MAPOL

OPERATOR	DESCRIPTION
=	Equality
<>	Inequality
>	Greater than
>=	Greater than or equal to
<	Less than
<=	Less than or equal to

9.3.3.2. Relational Operands

Relational operands must be of an arithmetic type integer or real. A complex operand is permitted only when the relational operator is = or <>.

9.3.3.3. Evaluation of Relational Expressions

In a relational expression involving the comparison of arithmetic operands, each of the arithmetic operands is evaluated prior to testing the relation. When the data type of two arithmetic operands differs, one operand is converted to the type of the other before the comparison is made. (See Section 3.2.5 for data type conversions.) The numeric values of the arithmetic operands are compared as specified by the relational operator, and the resulting value is either **TRUE** or **FALSE**.

9.3.4. MATRIX EXPRESSIONS

Matrix expressions are those which combine two or more matrices to yield a matrix result.

9.3.4.1. Matrix Operators

MAPOL allows four computational matrix operators as shown in Table 9-8. All matrices must be conformable in order to perform these operations. In the case of addition and subtraction, this means that the number of rows and columns in A and B must be the same. In the case of multiplications, the number of columns of the premultiplier must equal the number of rows in the post multiplier.

Table 9-8. Matrix Operators in MAPOL

OPERATOR	DESCRIPTION
A+B	$A_{ij} + B_{ij}$
A-B	$A_{ij} - B_{ij}$
A*B	$\sum_k A_{ik} B_{kj}$
-A	$- A_{ij}$

Matrix equations are written with the square brackets just as they are when declared. Examples of these equations are:

```
[A]    := [B] * [C];
[X]    := [Q(I)] * [Z];
[P(2)] := [R] - [S(2 * K + 1)]
```

Matrix operands may also be grouped to direct the order of operation. Instead of the parentheses used in scalar expressions, the square brackets are again used as shown below:

```
[A]    := [ [B] + [C] * [D] ] + [E];
[A(I)] := [ [B] * [[C] + [D] ] * [E]]] * [F];
```

All matrix algebra is optimized to provide the most effective use of computer resources. Matrices may also be multiplied by scalars or scalar expressions which may be **INTEGER**, **REAL**, or **COMPLEX**. These operations are written in the natural way; e.g.:

```
[A]    := (X) [B];
[Q(2)] := (R + S * T) [C] + [D];
```

All scalar multipliers must be placed on the left as shown. Note also that the multiplication operator is implied by the parentheses when multiplying a matrix by a scalar.

In addition to the matrix operations of Table 41, MAPOL allows for matrix transpose and inverse using the syntax of the following example:

```
[A] := TRANS([B]) * [C];
[X] := INV([KGG])*[PG];
[U] := TRANS([A]) * INV([A]*TRANS([A])0*[B];
```

Note that these operations are functions and, as such, the arguments are enclosed in parentheses. Also, the use of **TRANS** is only allowed in expressions. See Section 9.6.9 for a discussion of the **TRANS** function.

9.3.4.2. Matrix Operands and Expressions

Only matrix operands may be used in matrix expressions with the exception noted in Section 9.5.1. The matrix expressions are evaluated with the same hierarchy as that of arithmetic types.

9.3.5. ASSIGNMENT STATEMENTS

Assignment statements are used to compute and assign values to variables and array elements. The syntax of a MAPOL assignment is:

```
<var> := <expr> ;
```

The type of the expression <expr> is converted to the type of the variable <var> based on the rules of Table 9-9 where the following definitions are used.

VAL(X) - value of X
FIX(X) - convert X to an integer value
FLOAT(X) - convert X to a floating point value
REAL(C) - convert to the real part of a complex number

Table 9-9. Assignment Rules in MAPOL

TYPE OF <var>	TYPE OF <expr>	ASSIGNMENT RULE
INTEGER	INTEGER REAL COMPLEX	VAL(<expr>) ⇒ <VAR> FLOAT(VAL(<expr>)) ⇒ <VAR> FIX(REAL(VAL(<expr>))) ⇒ <VAR>
REAL	INTEGER REAL COMPLEX	FLOAT(VAL(<expr>)) ⇒ <VAR> VAL(<expr>) ⇒ <VAR> REAL(VAL(<expr>)) ⇒ <VAR>
COMPLEX	INTEGER REAL COMPLEX	FLOAT(VAL(<expr>)) ⇒ REAL(<VAR>) VAL(<expr>) ⇒ REAL(<VAR>) VAL(<expr>) ⇒ <VAR>

9.4. CONTROL STATEMENTS

9.4.1. INTRODUCTION

Control statements are statements used to alter and control the normally sequential execution of MAPOL instructions. There are five MAPOL control statements:

- **GOTO**
- **FOR...DO**
- **WHILE...DO**
- **IF...THEN...ELSE**
- **END , ENDP**

9.4.2. THE UNCONDITIONAL GOTO STATEMENT

The **GOTO** statement causes MAPOL program to jump unconditionally to the specified statement label. This label must exist in the same program unit (see Section 9.6) as the **GOTO** statement. The label identifier must also have been declared in the program unit's specification statements. The general syntax is:

```
GOTO <label>;
```

where <label> is any legal identifier that has been declared. When used, the label is denoted by <label> : followed by any valid MAPOL statement.

For example:

```
IF A < B GOTO SKIP;
...
...
...
SKIP: C:= B - A;
```

Note that the label must be followed by a colon.

9.4.3. ITERATION

It is often necessary to execute a group of statements repeatedly. Generally, although the statements themselves remain the same, the data on which they operate changes. This iteration or ***looping*** must terminate after a finite number of iterations; therefore, a decision must be made to determine whether to continue or terminate the loop. MAPOL supports two iteration forms: Each is described in this section.

9.4.3.1. The FOR...DO Loop

It is often necessary to perform a set of calculations a specific number of times, and that number does not depend on the statements within the loop. Consider the problem of summing the first 20 integers:

$$SUM = \sum_{n=1}^{20} n$$

Such a problem is ideally suited to the **FOR** loop and could be evaluated using the following MAPOL program:

```
MAPOL
INTEGER N,SUM,TOP;
   TOP  := 20;
   SUM  := 0;
   FOR N = 1 TO TOP DO
      SUM := SUM + N;
   ENDDO;
   PRINT ("(1X,'SUM = ',15)" , SUM );
END;
```

The general syntax of the **FOR** loop is

```
FOR <var> = <expl> TO <exp2> [ BY <exp3> ] DO
   ...
   ...
ENDDO;
```

The loop counter var is called the control variable and may be any integer or real variable. <exp1>, <exp2>, and <exp3> are called the initial, terminal and incremental parameters, respectively. Note the incrementation clause

```
BY <exp3>
```

is optional, as in the example. If it does not appear, the increment is taken to be one.

Each loop terminates with the instruction **ENDDO**. The following rules must be noted:

(1) If <exp1> > <exp2>, then the body of the loop will still be executed once.
(2) The type of the control variable and the three expressions must be the same.
(3) The control variable may not be redefined inside of the the loop.

9.4.3.2. The WHILE...DO Loop

Another way to execute a group of statements repeatedly is with a **WHILE** loop. This type of loop is used to repeat groups of statements that typically modify a more complex condition than the simpler incrementation of the **FOR** loop. As an example, suppose it is desired to compute the cube root of a number X. If **a** is an approximation to the answer, then

$$b = \frac{2a + \frac{x^2}{a}}{3}$$

is an improved guess. The program shown below will compute the cube root of 10 to 3 significant figures:

```
MAPOL
   REAL X,OLD,NEW,TEMP,EPS;
   X := 10;
   OLD := 2.0; $ THE INITIAL GUESS $
   EPS := 0.001 $ THE CONVERGENCE CRITERION $
   WHILE ABS(OLD-NEW) > EPS DO
      TEMP := NEW;
      NEW := (2.0*OLD+X/OLD**2) / 3.0;
      OLD := TEMP;
   ENDDO;
   PRINT ("(1X,'X,CUBERT(X) ',2F15.5)",X,NEW);
END;
```

The general form of the **WHILE** loop is:

```
WHILE <cond> DO
   ...
   ...
ENDDO;
```

The <cond> is any conditional expression that results in a logical outcome.

9.4.4. THE IF STATEMENT

It is often necessary in a program to specify two or more alternatives that must be selected depending upon other program results. The IF statement allows this selection. There are three types of IF statements in MAPOL:

- LOGICAL **IF**
- BLOCK **IF**
- **IF...THEN...ELSE**

9.4.4.1. The Logical IF

The logical **IF** is used if a single expression is to be executed based on a particular condition. The syntax of this statement is

```
IF <cond> <statement> ;
```

where <cond> is any logical expression and <statement> is any legal executable MAPOL statement except:

1. A **WHILE** or **FOR** loop
2. Another logical **IF**
3. An **END**, **ENDP**, **ENDIF**, or **ENDDO** instruction
4. A **PROC** definition

Examples of the logical **IF** are:

```
IF A<B PRINT("1X,'A= ',15)" ,A);
IF ABS(NEW-OLD) >EXP NEW := OLD;
IF A AND B OR C CALL UTMPRT (,[KMAT]);
```

9.4.4.2. The Block IF

It is often necessary to perform a number of instructions based on a given condition. This can be accomplished by a block **IF** statement, the syntax of which is:

```
IF <cond> THEN
   ...
   ...
ENDIF;
```

Rather than a single statement, the body on the block may contain any number of statements:

```
IF A < B THEN
   C := 1.0;
   D := 4.0;
   CALL UTMPRT (, [MMAT]);
ENDIF;
```

9.4.4.3. The IF...THEN...ELSE

The **IF...THEN...ELSE** statement is used to execute one of two separate blocks of code depending on a specific condition. The syntax of this statement is:

```
IF <cond> THEN
...
...      BLOCK 1
...
ELSE
...
...      BLOCK 2
...
ENDIF;
```

If the <cond> is satisfied, the instructions in BLOCK 1 are executed. If <cond> is not satisfied, then BLOCK 2 is executed.

9.4.4.4. Nested IF Statements

IF statements may be nested to any level. That is, each IF or ELSE part may contain another IF statement, as shown below:

```
IF A > B THEN
   A := 100;
ELSE IF C<D THEN
      C := 200;
   ELSE
      C := 0;
   ENDIF;
ENDIF;
```

Note that each **IF...THEN...ELSE** must terminate with its own **ENDIF**. It is helpful to indent code so that the blocks are obvious.

9.4.5. THE END AND ENDP STATEMENTS

The **END** and **ENDP** statements are used to indicate the physical end of a MAPOL program or in-line procedure, respectively.

9.5. INPUT/OUTPUT STATEMENTS

The MAPOL compiler does not have facilities for input in the programming language. All input is handled by the ASTROS executive system. MAPOL does, however, allow direct output to the system print device as defined by the ASTROS host computer. Output is merged with the same file that contains all of the other ASTROS print output.

9.5.1. THE PRINT STATEMENT

Output printing is requested with the **PRINT** statement, the syntax of which is:

```
PRINT (<format> [, <print-list>]);
```

In order to allow maximum power and flexibility while minimizing training, the format specifications used by MAPOL are identical to those used by Fortran. The format is entered as a literal string, enclosed by quotation marks; i.e.,

```
"(1X,5E1.6)"
```

```
"(//1X,' X= ',F15.5)"
```

The <print-list> is a list of one or more defined variables to be printed. If only heading information is being printed, the <print-list> may be omitted. Examples of print statements are:

```
PRINT ("(1X,3I15)",I,J,K);
```

which prints the three integer variables **I**, **J**, and **K** using the indicated format and

```
PRINT ("(1X,'THIS IS A HEADER')");
```

which prints this message **"THIS IS A HEADER"**.

ASTROS does not attempt to check the validity of a format statement with the data types being printed. As a result, it is possible to cause a Fortran run-time error condition.

9.6. PROCEDURES AND FUNCTIONS

9.6.1. INTRODUCTION

One of the most powerful features of a programming language is the ability to define ***procedures***, or subroutines, that perform specialized tasks. Some procedures with special characteristics are called "functions". Each MAPOL main program, procedure or function is called a ***program unit***. This section explains the use of procedures and provides examples of their use.

9.6.2. PROGRAM UNITS AND SCOPE OF VARIABLES

Earlier, a MAPOL program was defined very simply as having the form:

```
MAPOL
   ...
   ...
   ...
END;
```

This form is called a ***main program***. A main program may also contain other program units that may be procedures or functions such as:

```
MAPOL
  PROC A;
  ...
  ...
  ENDP;
  REAL FUNC B;
  ...
  ...
  ENDP;
  ...
  ...
END;
```

All procedures must appear in the main program before any executable statements. Each procedure, or function, may have variable declarations within it. If it does, these variables are called ***local*** to the procedure. Variables defined in the main program prior to the definition of the procedures are called ***global***. The value of a local variable is not available outside of the procedure in which it is defined, while

global variables are available to all procedures that are defined after the declaration of the variable. Note that global variables must be defined in the main program preceding procedure definitions. Declarations following the procedures are *local* to the main program.

9.6.3. DEFINING A PROCEDURE

A procedure is defined in MAPOL by a declaration:

```
PROC <procname> [ <params> ];
```

where <procname> is any identifier. If this name is the same as a run-time procedure, the new procedure will be used. <params> is an optional list of formal parameters that are used to pass information into and retrieve information from the procedure.

```
<params>    := ( <paramlist> )
<paramlist> := <ident> | <paramlist> , <ident>
```

Examples are:

```
PROC MYPROC ( A, B, C ) ;
PROC GETONE ;
```

The **PROC** statement is called the procedure ***head***. It is followed by the ***body*** and an **ENDP** statement:

```
PROC TEST;
   ...       PROCEDURE BODY
   ...       "
   ...       "
ENDP;
```

This defines the procedure program unit. As an example, to find the square root of a real number

$a = (b)^{\frac{1}{2}}$

a Newton-Raphson iteration technique can be used

$$a_{n+1} = a_n - \frac{(a_n^2 - b)}{2a_n}$$

The iteration proceeds to the desired accuracy as determined by

$| a_n - a_{n+1} | < EPS$

A MAPOL procedure for this is shown below:

```
PROC USQRT(A,SQRTA);
   REAL A,SQRTA,EPS,DELTA,AOLD;
   EPS   := 0.0001;
   SQRTA := 1.0;
   DELTA := 1.0;
   WHILE ABS(DELTA) > EPS DO
      AOLD = SQRTA;
      SQRTA := AOLD - ((AOLD*AOLD-A) / (2.0*AOLD));
      DELTA := SQRTA - AOLD;
   ENDDO;
END;
```

9.6.4. INVOKING A PROCEDURE

Once procedures are defined, they may be used anywhere within the main program or in a subsequent procedure. This is done with the MAPOL statement:

```
CALL <procname> [ <userparm> ];
```

where <procname> is one of the defined procedures. The optional <userparm> are the actual user-defined variables to be passed to the procedure. They must agree in number and type with the PROC definition. Parameters are passed by name. For example, a program segment using the square root procedure of the last Section is:

```
MAPOL;
REAL X,Y;
...
...
X := 5;
CALL USQRT(X,Y);
...
...
END;
```

The parameters **X** and **Y** are the actual variables that will be used in place of the formal parameters in the procedure definition. Note that procedures may call other procedures ***if*** the called procedure has already been defined.

9.6.5. FUNCTION PROCEDURES

A special kind of procedure that can have only one output value is called a FUNCTION. Because it is a value, the type of the function must be declared. Valid types are integer, real, complex or logical. Therefore, the function head differs slightly from that of the procedure:

```
<type> FUNC <funcname> [ <params> ]
```

Again, <type> must be included and all other rules are the same as those for a regular procedure.

Unlike procedures, functions are invoked with their name and arguments as in Fortran and they can, therefore, be used directly in assignment statements and expressions, e.g.,

```
A := SIN(X);
B := X + Y * SQRT(Z);
```

9.6.5.1. Examples of Variable Scope

To clarify the concept of variable scope, consider the following example:

```
MAPOL;
   INTEGER A;
   PROC MYPROG(B,C);
      INTEGER B,C;
      ...
      ...
      C := B*A; $ A is available, E and F are not
   ENDP;
   PROC YOURPG(H,I);
      REAL H,I;
      ...
      ...
   ENDP;
   REAL E,F;
   RELATION FOO,BAR;
   ...
   ...
END;
```

In this example, the variable **A** is global to all procedures because its declaration precedes the **PROC** declarations. **B** and **C** are local to **MYPROG** because their declarations appear in the body of that procedure. Finally, **E**, **F**, **FOO** and **BAR** are local to the main program and cannot be used by either procedure. Variables may be global to all **PROC**s or local to the main program. All **PROC** definitions must appear contiguously in the program with no intervening declarations.

9.6.6. INTRINSIC FUNCTION PROCEDURES AND INTRINSIC PROCEDURES

In addition to the user defined procedures and functions within a MAPOL main program unit, MAPOL provides a set of predefined functions and procedures to perform certain tasks in a similar manner to other high level languages such as Fortran. These *intrinsic* procedures are in addition to the engineering modules defined as part of the ASTROS system generation process. The set of intrinsic procedures within the MAPOL language can be broken into three groups: intrinsic mathematical functions, intrinsic relational procedures and general intrinsic procedures. Each group is discussed separately in the following sections.

9.6.7. INTRINSIC MATHEMATICAL FUNCTIONS

Table 57 shows the list of intrinsic mathematical functions available in MAPOL. These functions make up the mathematical function library within the MAPOL language and provide the user with the capacity

to perform a wide variety of tasks within the MAPOL program units. With very few exceptions, the MAPOL mathematical functions are identical in form to those in the Fortran language; the exceptions are noted in Table 9-10. Trigonometric functions in MAPOL use radian angles as arguments and result in radian angles just as in Fortran. All MAPOL functions are "generic" in the sense that they support multiple data types (**INTEGER**, **REAL**) as arguments and perform the appropriate conversions.

9.6.8. INTRINSIC RELATIONAL PROCEDURES

As discussed in Section 9.4, MAPOL has provided a means by which individual relational entries (row/attribute combinations) may be accessed directly. Table 9-11 shows the argument lists to the set of intrinsic procedures provided to enable the MAPOL programmer to open relations, to retrieve particular rows, to update or add rows and to close the relation. In combination with the **RELATION** and **PROJECT** declarations, these procedures provide a direct database interface that neatly matches the full relational application programming interface in CADDB. There is an implementation maximum of five open relational variables at any time during the execution of a MAPOL program.

Figure 9-2 shows a simple MAPOL procedure that manipulates a relation called GPOINT. Two rows are placed in the relation followed by a conditional retrieval to obtain one of the tuples for use in an additional operation.

9.6.9. GENERAL INTRINSIC PROCEDURES

Two other intrinsic procedures have been provided to enhance the utility of MAPOL: the **EXIT** and **TRNSPOSE** procedures. The first is identical to the common Fortran extension **EXIT**. The MAPOL statement

```
CALL EXIT;
```

will cleanly terminate the ASTROS execution without requiring the user to jump to the end of the MAPOL sequence. This is particularly useful when an edited standard solution sequence is used. The **TRNSPOSE** procedure provides an additional **MATRIX** operation that is otherwise missing from the language. While the operation

```
[A] := TRANS (B) * [C];
```

is available within the syntax of MAPOL expression, the operation

```
[A] := TRANS (B);
```

is not. The intrinsic procedure **TRNSPOSE** allows this matrix operation to be performed. The form of **TRNSPOSE**

```
CALL TRNSPOSE ([A], [TRANSA]);
```

where **[A]** is the matrix to be transposed and **[TRANSA]** is the resultant transposed matrix.

Table 9-10. Intrinsic Mathematical Functions in MAPOL

PROCEDURE	DESCRIPTION	USAGE
ABS	Absolute value	**A := ABS(B);**
ACOS	Cosine	**A := ACOS(B);**
ASIN	Arcsine	**A := ASIN(B);**
ATAN	Arctangent	**A := ATAN(B);**
CMPLX	Complex	**A := CMPLX(B,C);**
COS	Cosine	**A := COS(B);**
COSH	Hyperbolic cosine	**A := COSH(B);**
EXP	Exponential	**A := EXP(B);**
IMAG	Imaginary (Equivalent to FORTRAN AIMAG)	**A := IMAG(B);**
LN	Natural Logarithm (Equivalent to FORTRAN LOG)	**A := LN(B);**
LOG	Common Logarithm (Equivalent to FORTRAN LOG10)	**A := LOG(B);**
MAX	Selects largest value	**A := MAX(B,C,..);**
RE	Real component (Equivalent to FORTRAN REAL)	**A := RE(B);**
SIN	Sine	**A := SIN(B);**
SINH	Hyperbolic sine	**A := SINH(B);**
SQRT	Square root	**A := SQRT(B);**
TAN	Tangent	**A := TAN(B);**
TANH	Hyperbolic tangent	**A := TANH(B);**

Table 9-11. Intrinsic Relational Procedures in MAPOL

PROCEDURE	DESCRIPTION AND CALLING SEQUENCE	
RECEND	To end the definition of relational conditions. (A maximum of 10 may be applied per relation)	
	`CALL RECEND( <rel-var> );`	
RELCND	To define relational conditions	
	`CALL RELCND(<rel-var>, <attr>, <relop>, <value>);`	
	`<attr>`	`is an attribute named in quotation marks`
	`<relop>`	`is one of "GT", "LT", "EQ", "NE", "GE", "LE"`
	`<value>`	`is the conditional value`
	(A maximum of 10 may be applied per relation)	
RELADD	To add a tuple to a relation	
	`CALL RELADD ( <rel-var> );`	
RELEND	To close a relation	
	`CALL RELEND ( <rel-var>);`	
RELGET	To retrieve a tuple form an open relation	
	`CALL RELGET ( <rel-var>, <status> );`	
	`<status>`	is an integer variable that is non-zero if an error occurred
RELUPD	To update the fields in an existing tuple.	
	`CALL RELUPD ( <rel-var> );`	
RELUSE	To open a relation.	
	`CALL RELUSE ( <rel-var>, <ntuple>, <status> );`	
	`<ntuple>`	is an integer variable which contains the number of tuples in the relation on output
	`<status>`	is an integer variable that is non-zero if an error occurred

```
MAPOL
RELATION GPOINT;
INTEGER GID, NTUPLES, ERRSTAT;
REAL X,Y,Z;
PROJECT GPOINT USING GID,X,Y,Z;
     CALL RELUSE( GPOINT, NTUPLES, ERRSTAT );
     PRINT( "(' NTUPLES = ', I6)", NTUPLES );
     IF ERRSTAT <> 0 THEN
        PRINT( "(' ERRSTAT IS ', I6)", ERRSTAT );
        CALL EXIT;
     ENDIF;

     GPOINT.GID:=1;
     GPOINT.X :=5.0;
     GPOINT.Y :=6.0;
     GPOINT.Z :=7.0;
     CALL RELADD( GPOINT );

     GPOINT.GID:=2;
     GPOINT.X :=15.0;
     GPOINT.Y :=16.0;
     GPOINT.Z :=17.0;
     CALL RELADD( GPOINT );

     GPOINT.GID:=3;
     GPOINT.X :=.05;
     GPOINT.Y :=.06;
     GPOINT.Z :=.07;
     CALL RELADD( GPOINT );

     CALL RELEND( GPOINT );
```

Figure 9-2. MAPOL Program Using Relational Procedures

This page is intentionally blank.

Chapter 10.

REFERENCES

1. Herendeen, D. L., Hoesly, R. L. and Johnson, E. H., "Automated Strength-Aeroelastic Design of Aerospace Structures," AFWAL-TR-85-3025, September 1985.
2. *The NASTRAN User's Manual (Level 17.5)*, National Aeronautics and Space Administration, NASA SP-222(05), December 1978.
3. Garvey, S. J., "The Quadrilateral Shear Panel," *Aircraft Engineering*, May 1951, p. 134.
4. Etkin, B., *Dynamics of Flight*, John Wiley and Sons, Inc., New York, May 1967.
5. Woodward, F. S., "USSAERO Computer Program Development, Versions B and C," NASA CR 3227, 1980.
6. Herendeen, D.L. and Ludwig, M.R., "Interactive Computer Automated Design Database (CADDB) Environment User's Manual," AFWAL-TR-88-3060, August 1988.

This page is intentionally blank.