U N A

COMPUTER PROGRAM FOR STATIC AND DYNAMIC

STRUCTURAL ANALYSIS BY FINITE ELEMENT METHOD

USER & VERIFICATION MANUAL

- version 7.24 -

By

Dr Zoran Rudic

2023

www.una-fem.com
info@una-fem.com

U N A

COMPUTER PROGRAM FOR STATIC AND DYNAMIC

STRUCTURAL ANALYSIS BY FINITE ELEMENT METHOD

USER & VERIFICATION MANUAL

- version 7.24 -

By

Dr Zoran Rudic

2023

PREFACE

UNA is the finite element solver for static and dynamic structural analysis. The solver can be used for the following analysis tasks:

- Input Data Check
- Static Analysis
- Modal Analysis
- Elastic Buckling
- Substructuring
- Transient Response
- Frequency Response
- Random Vibrations
- Aeroelastic Response

BOOK STRUCTURE

Sect 1. Executive Control Section reference material for UNA input data preparation.
Sect 2. Bulk Data Section reference material for UNA input data preparation.
Sect 3. Guide through solution sequences, special capabilities and modelling techniques.
Sect 4. Verification examples.

PROGRAM INSTALLATION AND RUNNING

Create the new folder on hard drive. Copy all files from the UNA package to new folder.

Run program by opening Windows Explorer and move to folder where program is installed. Click twice on UNA724.EXE and type file name when asked.

Modelling tasks can be performed by using commercial packages such as FEMAP, PATRAN or I_DEAS, with all of them capable of writing NASTRAN Bulk Data Entry files. The UNA solver is supported by the NASUNA program that provides the following translations:

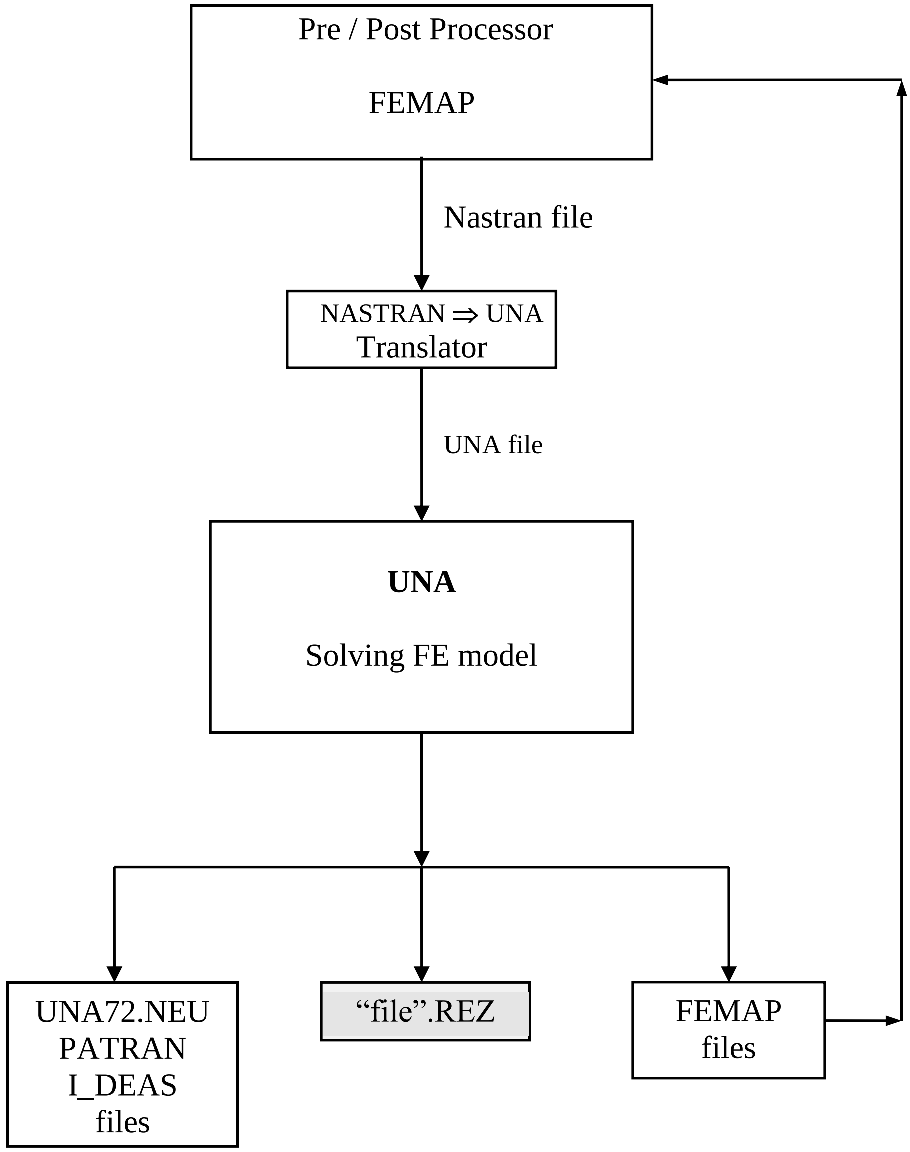
- NASTRAN $\Rightarrow$ UNA (Translate "file".NAS to "file".UNA)
- UNA $\Rightarrow$ NASTRAN (Translate "file".UNA to "file".NAS)

When run UNA creates the following files:

″file″.REZ	Output in UNA format (default).
UNA72.NEU	Output in UNA "NEUTRAL" format. This file is used by third party programs. For details see NEUTRAL Executive Data Entry.
UNA_*.NEU	Output in FEMAP format. These files are used by FEMAP for postprocessing. For details see FEMAP Executive Data Entry.
UNA_SUB*	Output in PATRAN format. These files are used by PATRAN for postprocessing. For details see PATRAN Executive Data Entry.
UNA.UF*	Output in I_DEAS format. These files are used by I_DEAS for postprocessing. For details see UNIVERSAL Executive Data Entry.

The sketch shown on the following page explains the typical modelling, running and postprocessing cycle with FEMAP for pre/postprocessing and UNA for model running and solving.

- FEMAP is a Siemens PLM software.
- I_DEAS is a product of the Structural Dynamic Research Company.
- NASTRAN is a registered trademark of NASA.
- PATRAN is a product of the MSC Software Corporation.


Pre / Post Processor
FEMAP
Nastran file
NASTRAN ⇒ UNA
Translator
UNA file
UNA
Solving FE model
UNA72.NEU
PATRAN
I_DEAS
files
"file".REZ
FEMAP
files

There are two separate blocks in the UNA input file :

1. EXECUTIVE CONTROL SECTION for controlling the FE run.

2. BULK DATA SECTION for FE model definition.

Executive Data Section

This section controls finite element solution process. Section starts form beginning of the file and ends with the following statement

BEGIN BULK

Explanation and command rules are defined in Section 1 of this manual. Commands may be specified in full or free format form. Only first three characters in form of capital letters are required, i.e.

```
SOLUTION    = 1
SOL         = 1
```

Common form for writing Executive Data Entry command is given by:

COMmand <option1>, <option2>, ... ,

Command and options have to be separated by blank or one of the following special characters

= , / { } ( ) "

Comment line is defined by **$** sign in the first place on the command line., i.e.

```
$
$  Model :  Wing Box ver 2.1,  11-Feb-96
$
```

Comment at line is defined by **!** sign. The portion behind sign is comment, i.e.

```
$
FORCE  =  SET 101      !   Print internal loads for elements listed at Bulk Data SET  101
$
```

All Executive Data Entry commands have a default value. Default values are highlighted, i.e.

MASTYP =	CONSISTENT
	LUMPED

Bulk Data Section

This section contains bulk data entry for defining the finite element model. Every Bulk Data Entry must start from first character in the command line, and be titled completely in capital letters. i.e.

1	2	3	4	5	6	7	8	9	10
GRID									

represents grid input data entry. Bulk Data Entries have two formats - normal and large. Here are examples of both :

normal format

1	2	3	4	5	6	7	8	9	10
8	8	8	8	8	8	8	8	8	8

large format

1	2	3	4	5	6
8	16	16	16	16	8

Normal format contains 10 fields each 8 characters long. First field is reserved for entry identification. Fields 2 - 10 are for input data. Entry data are in free format form, and can be of real, integer or character type. For example, real number 7 may be represented as

7.0 .7E1 0.7+1 70.-1 7+0 7.0E0

There cannot be empty spaces between characters in number representation. Large format is defined by sign * immediately after card identification. Sign has to be inside first field (inside first eight characters). Such defined large format entry contains 4 fields each 16 characters long. Last (number 6) field is ignored.

Bulk Data Entry may be continued by additional lines. Additional input is defined by signs

+ or *

in the first position of entry identification field. Sign + defines additional entry in the normal format form, and sign * defines additional entry in the large format form. There are examples of some possible combinations :

Large format data entry with large field continuation data entries

TYPE*					
*					
*					

Normal format card with large and normal field continuation entries

TYPE									
*									
+									
*									

Comment line is defined by **$** sign in the first place of the line , i.e.

$
$ Material : Aluminium 7075/T7351 Plate
$

Comment at line is defined by **!** sign. The portion behind sign is comment, i.e.

GRID 101 0 0.0 0.0 300. ! CBAR orientation grid

Section 1

Executive Control Section

Last entry in the Executive Control Section. After it a Bulk Data Section begins.

Format:

BEGIN BULK

Examples:

BEGIN BULK

Remarks:

Defines start of the BULK DATA section. Must exist.

CHECK

Enables stiffness matrix condition check. This check estimates the numerical round-off error.

Format:

CHECK =	ON OFF

Examples:

CHECK = ON

Describers	**Meaning**
ON	Enables numerical error check
OFF	Disables numerical error check

Remarks:

Numerical error check is based upon the following expression

$$\frac{dU}{U} = 100 \times \frac{L_{max}}{L_{min}} \times 2^{-t}$$

where dU/U is maximal expected numerical error (%), Lmax/Lmin stiffness matrix condition ratio, t number of bytes in one computer word (t=52 for PC version). Condition Lmax/Lmin is calculated by Lanczos numerical procedure.

Requests displacement printout in the output file.

Format:

DISPLACEMENTS	TRA ROT	=	ALL NONE RANGE SET	Num1 Num2	CRITICAL	Real OFF

Examples:

DISP = SET 101
DISP = NONE
DISP TRA = ALL
DISP TRA = RANGE 100 200 CRITICAL 0.5

Describers	Meaning
TRA	Translations selected. If blank both translations and rotations are selected.
ROT	Rotations selected. If blank both translations and rotations are selected.
ALL	Selects all grid points.
NONE	Deselect all grid points.
RANGE	Selects grid points range.
SET	Selects grid points as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Grid points range if coming after RANGE.
CRITICAL	Enables additional selective criterion based on displacement magnitude.
Real	Minimum displacement value if selective output is enabled. Grids with displacement larger than defined value will be printed in output file (selective output is enabled by CRITICAL option) .
OFF	Disabled selective output.

Remarks:

1. The units of translations are the same as the units of the model. Rotations are in units of radians.

2. Displacement results are output in DISPLACEMENT coordinate system (see IDIS on the GRID Bulk Data entry).

DLOAD

Selects a dynamic load to be applied in a frequency response or random vibrations analysis.

Format:

DLOAD = | n |

Examples:

DLOAD = 101

Describers	Meaning
n	Set identification of a RLOAD Bulk Data Entry.

Remarks:

1. A selected RLOAD dynamic load set is applied during the frequency response (SOL=6), or random vibrations analysis (SOL=8).

Requests input data echo printout.

Format:

ECHO	GRIDS EQNUM ELEMENTS LOADS FIXOTO MATTABLES PROPTABLES QPRESS ELOAD COORD INITCOND QLOAD SYSTEM TABLES	=	ON OFF

Examples:

ECHO = ON
ECHO EQNUM = ON

Describers	Meaning
GRIDS	Grid points data
EQNUM	System of equation numbers
ELEMENTS	Elements data
LOADS	Loads data
FIXOTO	FIXOTO generated stiffness data
MATTABLES	Material data
PROPTABLES	Property data
QPRESS	Pressure data
ELOAD	Element pressure and thermal loads
COORD	Coordinate systems data
INITCOND	Initial conditions
QLOAD	Bar / Beam distributed load
SYSTEM	System of equations data
TABLES	Tabular functions
ON /OFF	Activate / deactivate printing

FEMAP

Requests and controls output in FEMAP format for graphic postprocesing.

Format:

FEMAP	DISP VELO ACCE FORCE FREEBODY MODES POST REACT STRESS	=	ON OFF	COMPONENT	Num1	SET	Num2

Examples:

FEMAP = ON
FEMAP FORCE = ON COMPONENT 3 SET 102

Describers	Meaning
DISP	Nodal displacements.
VELO	Nodal velocities.
ACCE	Nodal accelerations.
FORCE	Element output.
FREEBODY	Free body load.
MODES	Modal displacements.
POST	Element post-processing output.
REACT	SPC reactions.
STRESS	Nodal stress output.
ON / OFF	Activate / deactivate output.
COMPONENT	Requests specific number of components for output.
Num1	Number of components that are required.
SET	Select only those elements listed on SET Bulk Data entry to be included in output, or to be used during nodal data averaging for nodal stress output.
Num2	Set number for SET Bulk Data entry.

Remarks:

This is request for additional files with UNA analysis output. The files are written in FEMAP format, and may be used for deformed and contour plots, animation etc.

UNA creates the following files for FEMAP postprocessing:

File Name	Output type	Output request
UNA_FEM.NEU	Analysis run output (except modes)	FEMAP (DISP, ... , STRESS) = ON
UNA_MOD.NEU	Eigenvectors (modal displacements)	FEMAP MODES = ON

REACT : SPC reactions

SPC reactions are output for all load cases in the UNA analysis run. Nodal forces and moments are output in structure BASIC coordinate system as a new load case with a label offset of 100,000, i.e. if the applied load case was 101, reactions are returned as a load case 100,101. Selective output is controlled with REACT = CRITICAL {Real}, see REACT Executive Data entry.

FREEBODY : Free body load

Free body load is output for all load cases in the UNA analysis run. Only the first free body, with lowest FID, is considered (if more than one is specified in the input file). Nodal forces and moments are output in structure BASIC coordinate system as a new load case with a label offset of 200,000, i.e. if the applied load case was 101, free body load is returned as a load case 200,101. Selective output is controlled by Executive Data Entry FREEBODY = CRITICAL {Real}.

DISP, VELO, ACCE : Nodal output (Output vectors from 1 to 8, 401-408, ... etc.)

Values are output in structure BASIC coordinate system. The components are defined as follows:
1 is total translation, 2-4 are X, Y, Z translations, 5 is total rotation, 6-8 are X, Y, Z rotations.

Analysis type		DISP	VELO	ACCE
Static, Buckling, Transient Response		1-8		
Random Vibrations, Aerodynamic Gust	r.m.s. value	401-408	501-508	601-608
	apparent frequency	411-418	511-518	611-618

MODES : Nodal output (Output vectors from 1 to 8)

Modal displacements are output in structure BASIC coordinate system.

STRESS : Nodal output (Output vectors from 1,001 to 1,038)

- By default all 41 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be output in the file. This option allows file size reduction if only a first few components are of the interest.

- In order to find the nodal stresses UNA takes output from plane (CMEMB, CTRIA3, CQUAD4, CSHELL3, CSHELL4) and solid (CSOLID) elements, other element types are ignored. For particular node contributions from **all** adjacent elements mentioned above are averaged by default. This can be altered by specifying SET {Num2} option on the command line. Only output data from elements that are listed on that SET Bulk Data entry will be taken into account during averaging procedure.

- The following page explains component data. Table symbols are as follows:

σvmis	Equivalent Von-Misses stress.
σmax, σmin, τmax	Plane element principal normal and shear stresses.
σmax, σmed, σmin τmax, τmed, τmin	Solid element principal normal and shear stresses.
σx, σy, σz, τxy, τyz, τzx	Components of complete stress tensor. The components are output in BASIC coordinate system.
Tsai-Hill	Tsai-Hill criterion, max. value found through thickness at node.
M.S.	Margin of Safety for composite material. Min. value found through plane element thickness at node.
top, middle, bottom	Plate element top, middle and bottom plane positions.

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared σvmis (max), and reported as Comp. 1.

- For multilayered composite material UNA calculates margins of safety for each ply, see MAT2 Bulk Data entry. Minimal value from all calculated margins for all plies through element thickness is declared M.S. (min), and reported as Comp. 3.

STRESS : Nodal output (Output vectors from 1,001 to 1,038)

Comp.	Meaning	Position
1	σvmis	Max
2	Tsai-Hill	Max
3	M.S.	Min
4		
5		
6	σvmis	top
7	σmax	top
8	σmin	top
9	τmax	top
10	σvmis	middle
11	σmax	middle
12	σmin	middle
13	τmax	middle
14	σvmis	bottom
15	σmax	bottom
16	σmin	bottom
17	τmax	bottom
18	σmax	solid
19	σmed	solid
20	σmin	solid

Comp.	Meaning	Position
21	τmax	solid
22	τmed	solid
23	τmin	solid
24	σx	top
25	σy	top
26	σz	top
27	τxy	top
28	τyz	top
29	τzx	top
30	σx	middle
31	σy	middle
32	σz	middle
33	τxy	middle
34	τyz	middle
35	τzx	middle
36	σx	bottom
37	σy	bottom
38	σz	bottom
39	τxy	bottom
40	τyz	bottom
41	τzx	bottom

FORCE : Element output

- By default all 24 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be written, and output file size reduced.

- By default file contains output data for all elements. It can be altered by specifying SET {Num2} option on the command line. Only elements listed at SET will be included in the file.

- The following page explains component data. Table symbols are as follows:

σa	Axial stress for CROD, CBAR and CBEAM.
$\sigma vmis$	Equivalent Von-Misses stress.
σx, σy, τxy	In-plane stresses, plane element local coordinate system.
σmax, σmin, τmax	In-plane principal stresses, plane element middle plane.
σx, σy, σz, τxy, τyz, τzx	Solid element stresses. SYS is equal to PARAM SOLID 3 = {Num} option that specifies coordinate system for stress components.
Tsai-Hill	Tsai-Hill criterion, max. value found through element thickness.
M.S.	Margin of Safety for composite material. Min. value found through plane element thickness at node.
Pa, Mt	Axial force and torsion moment.
M_{1A}, M_{1B}, V_{1A}, V_{1B}	CBAR plane 1 bending moment and shear force : end-A, end-B.
M_{2A}, M_{2B}, V_{2A}, V_{2B}	CBAR plane 2 bending moment and shear force : end-A, end-B.
Q21-Q41	CSHEAR panel shear flows for edges 1 to 4.
Nx, Ny, Nxy, Qx, Qy Mx, My, Mxy	Plane element unit loads in element local system. (force/moment/transverse shear per unit width).
A, K, B, T	Area, spring stiffness, damping factor, thickness.
Pload	Element distributed load (pressure, force/unit length).
X/C, C, B, A	CSTRIP element control point chorwise location, chord length, panel width, panel area.
Py, My, Z-press	CSTRIP element lift force, pitch moment at control point, element local coordinate system. Z-press is average pressure.
Cla, Cma, Clo, Cmo	CSTRIP element coefficients of lift and pitching moment.
Top, middle, bottom	Plate element top, middle and bottom plane positions.

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared $\sigma vmis$ (max), and reported as Comp. 1.

FORCE : Element output

Static, Buckling, Transient	: vectors	10,001 to 10,030	
Random Response, Aero Gust	: vectors	12,001 to 12,030	r.m.s. value
	vectors	12,101 to 12,130	apparent frequency

Comp.	CELAS2 CDAMP2	CROD CVISC	CBAR CBEAM	CSOLID	CSHEAR	2-D plane (.. CTRIA3…. .. CQUAD4 ..)	CSTRIP
1		σ_a	σ_a	σ_{vmis}	σ_{vmis}	σ_{vmis} (max)	X/C
2						Tsai-Hill (max)	C
3						M.S. (min)	B
4	K (B)	A (B)	A	SYS	T	T	A
5			Pload	Pload	Pload	Pload	
6	Pa	Pa	Pa			Nx	Pz
7	Pa	Pa	Pa			Ny	My
8	Pa	Pa	Pa		Nxy	Nxy	Z-press
9	Mt	Mt	Mt			Mx	
10	Mt	Mt	M_{1A}			My	
11	Mt	Mt	M_{1B}			Mxy	
12		σ_a	V_{1A}		Q21	Qx	Cla
13		σ_a	V_{1B}		Q23	Qy	Cma
14		σ_a	M_{2A}		Q43	σ_{max} (midd)	Clo
15			M_{2B}		Q41	σ_{min} (midd)	Cmo
16			V_{2A}	σ_x		σ_x (top)	
17			V_{2B}	σ_y		σ_y (top)	
18				σ_z	τ_{xy}	τ_{xy} (top)	
19		σ_a	σ_a	τ_{xy}		σ_x (middle)	
20		σ_a	σ_a	τ_{yz}		σ_y (middle)	
21		σ_a	σ_a	τ_{zx}	τ_{xy}	τ_{xy} (middle)	
22						σ_x (bottom)	
23						σ_y (bottom)	
24					τ_{xy}	τ_{xy} (bottom)	

- For multilayered composite material UNA calculates margins of safety for each ply, see MAT2 Bulk Data entry. Minimal value from all calculated margins for all plies through element thickness is declared M.S. (min), and reported as Comp. 3.

POST : Element post-processing output (Output vectors from 20,001 to 20,040)

- By default all 40 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be written, and output file size reduced.

- By default file contains output data for all elements. It can be altered by specifying SET {Num2} option on the command line. Only elements that are listed on SET Bulk Data entry will be included in the file.

- Postprocesing data are based on CROD, CBAR and CBEAM output. UNA searches for all 2-D elements sharing the same line between two grids, and then collects grid point forces from line and all adjacent 2-D elements. For detailed explanation see POST Executive Data Entry.

- The following page explains components data. Table symbols are as follows:

ΔQ	Shear flow balancing axial forces at ends A and B.
Qmax	Maximal shear flow on any of 2-D adjacent elements.
σ_A, σ, σ_B	Axial stresses : end-A, midspan, end-B.
P_A, P, P_B	Axial forces : end-A, midspan, end-B.
M_{1A}, M_1, M_{1B}	Plane 1 bending moment : end-A, midspan, end-B.
M_{2A}, M_2, M_{2B}	Plane 2 bending moment : end-A, midspan, end-B.
V_{1A}, V_1, V_{1B}	Plane 1 shear force : end-A, midspan, end-B.
V_{2A}, V_2, V_{2B}	Plane 2 shear force : end-A, midspan, end-B.
Mt	Torsion moment.
A, L, Le	Cross section area, length between grids, length between ends.

- Output is provided for two locations : grid points and element ends (may not be the coincident positions due to CBAR offset). For definition of the output coordinate systems see PARAM POST Executive Data Entry.

POST : Element post-processing output (Output vectors from 20,001 to 20,040)

Comp.	CROD CBAR CBEAM	
	Output	Loc.
1	A	
2	L	
3	Le	
4	ΔQ	grid
5	Qmax	grid
6	σ_A	element
7	σ	element
8	σ_B	element
9	P_A	grid
10	P	grid
11	P_B	grid
12	M_{1A}	grid
13	M_1	grid
14	M_{1B}	grid
15	V_{1A}	grid
16	V_1	grid
17	V_{1B}	grid
18	M_{2A}	grid
19	M_2	grid
20	M_{2B}	grid

Comp.	CROD CBAR CBEAM	
	Output	Loc.
21	V_{2A}	grid
22	V_2	grid
23	V_{2B}	grid
24	Mt	grid
25	P_A	element
26	P	element
27	P_B	element
28	M_{1A}	element
29	M_1	element
30	M_{1B}	element
31	V_{1A}	element
32	V_1	element
33	V_{1B}	element
34	M_{2A}	element
35	M_2	element
36	M_{2B}	element
37	V_{2A}	element
38	V_2	element
39	V_{2B}	element
40	Mt	element

FIXOTO

Requests grid point local stiffness matrix check and elimination of singularities.

Format:

FIXOTO =	TRA / ROT / ALL / NONE	EPZERO	Real	REACT	ON / OFF	ERROR	GO / STOP

Examples:

FIXOTO = ROT
FIXOTO = ROT EPZERO 1.E-4 REACT ON

Describers	Meaning
TRA	Translation DOF’s will be checked.
ROT	Rotation DOF’s will be checked.
ALL	Translation and rotation DOF’s will be checked.
NONE	Stiffness check disabled. (default).
EPZERO	Tolerance for stiffness matrix singularity test.
Real	Test tolerance (1.E-7 default).
REACT	Request for FIXOTO reactions printout (load exits).
ON / OFF	Activate / deactivate FIXOTO reactions printout.
ERROR	Option for stiffness matrix decomposition error handling.
GO / STOP	Proceed / abort at stiffness matrix decomposition error.

Remarks:

1. Program checks for local nodal stiffness matrix singularities or quasi-singularities. The potential problem is detected when local stiffness minimum eigenvalue is EPZERO times smaller than average value. If that occurs, program includes optimally defined elastic restraint that is directed parallel to the local stiffness matrix eigenvector associated with minimum eigenvalue. Physically, restraint represents spring that connects grid point with the ground (load exit).

2. It is recommended to check rotation DOF’s only, and to use EPZERO default value, i.e.:

   FIXOTO = ROT

3. If stiffness matrix decomposition error occurs, FIXOTO = ERROR STOP / GO controls the continuation of the process.

Output format definition.

Format:

FORMAT = | n |

Examples:

FORMAT = 1

Describers	Meaning	
n = 0	F output format (E if necessary)	(default)
= 1	E output format.	

Remarks:

1. Output printing format is F (default). If it happens that number is larger than available F format, program will print output data using E format automatically.

FORCE

Selects elements for internal load and stress output.

Format:

FORCE	ROD BAR BEAM MEMB SOLID SHEAR PLATE SUPEL ELAS2 STRIP	=	ALL NONE RANGE SET	Num1 Num2

Examples:

FORCE = ALL
FORCE SOLID = SET 101

Describers	**Meaning**
ROD-STRIP	Element type selection. If blank all types are selected.
ALL	Selects all elements.
NONE	Deselect all elements.
RANGE	Selects element range.
SET	Selects elements as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Element range if coming after RANGE.

Remarks:

1. When command does not include any of available options ROD - STRIP, i.e.

   FORCE = SET 102

   then it is related to all element types.

2. To alter element internal load or stress output type, location, and coordinate systems, see PARAM Executive Control Commands.

FREEBODY

Requests and controls free body forces printout.

Format:

FREEBODY	FORCE MOMENT	=	ALL NONE RANGE SET	Num1 Num2	CRITICAL	Real OFF

Examples:

FREEBODY = SET 101
FREEBODY FORCE = ALL
FREEBODY MOMENT = RANGE 100 200 CRITICAL 1.

Describers	Meaning
FORCE	Force selected. If blank both force and moment are selected.
MOMENT	Moment selected. If blank both force and moment are selected.
ALL	Selects all freebody groups.
NONE	Deselect all freebody groups.
RANGE	Selects freebody groups range.
SET	Selects freebody groups as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Freebody range if coming after RANGE.
CRITICAL	Enables additional selective criterion based on force (moment) magnitude.
Real	Minimum force (moment) value if selective output is enabled. Grids with force components larger than defined value will be printed in output file (selective output is enabled by CRITICAL option) .
OFF	Disables selective output.

Remarks:

1. Free body group of elements is defined on FREEBODY Bulk Data Entry. This entry selects groups to be printed.

Selects the set of forcing frequencies to be solved for in frequency response runs.

Format:

FREQUENCY = | n |

Examples:

FREQ = 101
FREQUENCY = 99

Describers	Meaning
n	Set identification number of FREQ, FREQ1 and FREQ2 Bulk Data Entries.

Remarks:

1. A frequency set selection is required for a frequency response problem (SOL=6, 8, 9).

2. All FREQ, FREQ1 and FREQ2 entities with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.

FRESPONSE

Requests and controls frequency response output.

Format:

FRESPONSE	DISP VELO ACCE ROD BAR BEAM MEMB SOLID SHEAR PLATE CELAS2 DAMP2 VISC STRIP	=	ALL NONE RANGE SET	Num1 Num2	COMP	Num1 Num2	PHASE IMAG

Examples:

```
FRESP   BAR   =  SET  101   COMP   7  17
FRESP   STRIP =  ALL  IMAG
```

Describers	Meaning
DISP	Nodal displacements.
VELO	Nodal velocities
ACCE	Nodal accelerations.
ROD - STRIP	Element type selection. If blank all types are selected.
ALL	Selects all entities.
NONE	Deselect all entities.
RANGE	Selects entity range.
SET	Selects entities as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Node range if coming after RANGE.
COMPONENT	Select entity components.
Num1 - Num2	Component range if coming after COMP.
PHASE	Amplitude / Phase output required.
IMAG	Real / Imaginary output required.

Remarks:

1. The nodal displacements, velocities and accelerations are output for all forcing frequencies as specified at selected FREQ, FREQ1 and FREQ2 entries. The components 1 - 3 are translations and components 4 - 6 are rotations in the grid displacement system.

2. Element component data are explained in the table on the following page. The following is description of the table symbols:

σa	Axial stress for CROD, CBAR and CBEAM.
σx, σy, τxy	In-plane stresses, plane element local coordinate system.
σx, σy, σz, τxy, τyz, τzx	Solid element stresses. Components are output in coordinate system specified by PARAM SOLID 3 = {Num}.
Pa, Mt	Axial force and torsion moment.
M_{1A}, M_{1B}, V_{1A}, V_{1B}	CBAR plane 1 bending moment and shear force : end-A, end-B.
M_{2A}, M_{2B}, V_{2A}, V_{2B}	CBAR plane 2 bending moment and shear force : end-A, end-B.
Q21-Q41	CSHEAR panel shear flows for edges 1 to 4.
Nx, Ny, Nxy, Qx, Qy Mx, My, Mxy	Plane element unit loads in element local system. (force/moment/transverse shear per unit width).
Py, My, Z-press	CSTRIP element lift force, pitch moment at control point, element local coordinate system. Z-press is average pressure.
top, middle, bottom	Plate element top, middle and bottom plane positions.

3. PHASE / IMAG selection controls standard UNA output, as well as NEUTRAL and FEMAP output files.

Comp.	CELAS2 CDAMP2	CROD CVISC	CBAR CBEAM	CSOLID	CSHEAR	2-D plane (.. CTRIA3…. .. CQUAD4 ..)	CSTRIP
1							
2							
3							
4							
5							
6	Pa	Pa	Pa			Nx	Pz
7	Pa	Pa	Pa			Ny	My
8	Pa	Pa	Pa		Nxy	Nxy	Z-press
9	Mt	Mt	Mt			Mx	
10	Mt	Mt	M_{1A}			My	
11	Mt	Mt	M_{1B}			Mxy	
12		σa	V_{1A}		Q21	Qx	
13		σa	V_{1B}		Q23	Qy	
14		σa	M_{2A}		Q43		
15			M_{2B}		Q41		
16			V_{2A}	σx		σx (top)	
17			V_{2B}	σy		σy (top)	
18				σz	τxy	τxy (top)	
19		σa	σa	τxy		σx (middle)	
20		σa	σa	τyz		σy (middle)	
21		σa	σa	τzx	τxy	τxy (middle)	
22						σx (bottom)	
23						σy (bottom)	
24					τxy	τxy (bottom)	

Zero frequency tolerance.

Format:

FZERO = | Real |

Examples:

FZERO = 0.01

Describers	Meaning	
Real	The zero frequency tolerance [Hz].	(default = 0.0)

Remarks:

1. The frequencies with absolute value smaller than FZERO will be declared a rigid body frequencies and assigned a value of 0.0 [Hz]

GPFORCE

Requests grid point forces printout in the form of element free body load.

Format:

GPFORCE	FORCE MOMENT	=	ALL NONE RANGE SET	Num1 Num2

Examples:

GPFORCE = SET 101
GPFORCE FORCE = ALL
GPFORCE MOMENT = RANGE 100 200

Describers	**Meaning**
FORCE	Force selected. If blank both force and moment are selected.
MOMENT	Moment selected. If blank both force and moment are selected.
ALL	Selects all elements.
NONE	Deselect all elements.
RANGE	Selects element range.
SET	Selects elements as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Elements range if coming after RANGE.

Remarks:

1. When command does not include any of available options FORCE or MOMENT, i.e.

   GPFORCE = ALL

   then it is related to both forces and moments.

2. Element grid point forces and moments are output in grid DISPLACEMENT coordinate system (see IDIS on the GRID Bulk Data entry). Load represents element free body load (element stiffness matrix is multiplied with deflection vector).

Mass matrix scaling.

Format:

GRAV = | Real |

Examples:

GRAV = 1.E+4

Describers	Meaning
Real	The value used as a mass matrix divisor. (default = 1.0)

Remarks:

1. The terms of the mass matrix are divided by the value of this parameter. This procedure is required for dynamic analysis in order to establish a consistent system of units.

$$M = \frac{\int \rho \cdot dV}{GRAV}$$

2. Compatible system of units is achieved by defining GRAV as an acceleration value correlating force, mass and length, i.e. if the system of units is defined as:

Force	- [daN]	
Length	- [mm]	1 [daN] = 1.E+4 [mm/s**2] x 1 [kg]
Mass	- [kg]	

then GRAV has the following value:

GRAV = 1.E+4

GUST

Gust load selection in an aeroelastic response analysis.

Format:

GUST = | n |

Examples:

GUST = 99

Describers	**Meaning**
n	Set identification of a GUST Bulk Data Entry. (Integer > 0).

Remarks:

1. A gust field selection is required for an aeroelastic response problem (SOL=9).

Specify number of time steps for transient response analysis.

Format:

INCREMENT = | n |

Examples:

INC = 8

Describers	Meaning
n	Number of increments (Integer > 0). (default = 5)

Remarks:

1. This entry specifies number of increments in dynamic transient response analysis. Time interval between starting time T1 and ending time T2 is divided into INC equal parts. For each such interval of time program calculates grid point deflections and element internal loads.

LAMBDA

Specify desired number of modes in modal and elastic stability analysis.

Format:

LAMBDA = | Num |

Examples:

LAMBDA = 10

Describers	**Meaning**
Num	Number of modes (Integer > 0). (default = 5)

Remarks:

1. This entry specify number of free vibration modes in dynamic analysis, or number of modes in buckling analysis.

Specify number of lowest modes in a modal formulation.

Format:

LMODES = | Num |

Examples:

LMODES = 3

Describers	Meaning
Num	Number of lowest modes (Integer > 0). (default LMODES = LAMBDA)

Remarks:

1. LMODES is the number of lowest modes to use in modal formulation (SOL=6,8,9). If LMODES=0 or not specified, the number is equal to LAMBDA calculated modes.

MASTYP

Specify mass matrix type to be consistent or lumped.

Format:

MASTYP	=	CONSISTENT
		LUMPED

Examples:

MASTYP = CONSISTENT
MASTYP = LUMPED

Describers	**Meaning**
CONSISTENT	Selects consistent global mass matrix.
LUMPED	Selects lumped global mass matrix.

Remarks:

1. This entry specify global mass matrix type - consistent in full matrix form, or lumped in diagonal matrix form.

RAM memory request.

Format:

MEMORY = | Num |

Examples:

MEMORY = 8.5

Describers	**Meaning**
Num	Requested RAM memory in MB.

Remarks:

1. This entry specifies request for RAM to be assigned to UNA during the run. By default program will try to assign 40 MB, and if not successful, the memory assigned will be the maximum allowed by operating system.

2. It is recommended to not request more than 75-80 % of computer installed RAM. This will provide operating system with some left space to manage other processes as well.

MGROUP

Selects a substructure (group of elements), to be included in elastic stability and modal analysis.

Format:

MGROUP =	ALL NONE RANGE SET	Num1 Num2

Examples:

MGROUP = SET 12
MGROUP = RANGE 100 200

Describers	**Meaning**
ALL	All elements are used to assemble geometric and mass matrices (default).
NONE	None of the elements are used to assemble geometric and mass matrices.
RANGE	Selects elements range to be used to assemble geometric and mass matrices.
SET	Selects elements as specified at SET Bulk Data entry.
Num1	Set number (after SET).
Num1 - Num2	First and last element number (after RANGE).

Remarks:

1. This Executive Data Entry allows that geometric or mass matrix are assembled only from selected elements. This procedure is used when part of the structure is of interest to be analysed for buckling or natural frequencies. The remaining structure, (elements not specified by MGROUP entry), provides elastic support for the analysed part.

Requests modal displacement printout in the output file.

Format:

MODES = | ALL / NONE / RANGE | | Num1 Num2 | | CRITICAL | | Real / OFF |

MODES =	ALL	Num1 Num2	CRITICAL	Real
	NONE			OFF
	RANGE			

Examples:

MODES = ALL
MODES = NONE
MODES = RANGE 100 200 CRITICAL 0.5

Describers	Meaning
ALL	Selects all modes
NONE	Deselect all modes.
RANGE	Selects modes range.
Num1 - Num2	First and last mode for printing.
CRITICAL	Enables additional selective criterion based on displacement magnitude.
Real	Minimum displacement value for selective output. Grids with modal displacement larger than defined value will be printed in output file (selective output is enabled by CRITICAL option) .
OFF	Disabled selective output.

Remarks:

1. Modal displacements are normalized in order to provide a unit modal mass.

2. Displacement results are output in DISPLACEMENT coordinate system (see IDIS on the GRID Bulk Data entry).

MPC

Selects a multipoint constraint set or coupling set be applied.

Format:

MPC = | n |

Examples:

MPC = 10

Describers	**Meaning**
n	A) Set identification number of a multipoint constraint set that appears on a MPC or MPCADD Bulk Data entry (Integer > 0). B) Set identification number of a coupling set that appears on a COUPG Bulk Data entry (Integer > 0).

Requests and controls output in UNA "NEUTRAL" file, which is written in ASCI textual format. File is intended to provide third party programs with an easy access to the analysis output.

Format:

NEUTRAL	DISP VELO ACCE FORCE FREEBODY GPFORCE MODES POST REACT STRESS	=	ON OFF	COMPONENT	Num1	SET	Num2	ASCI BIN

Examples:

NEUTRAL = ON
NEUTRAL FORCE = ON COMPONENT 3 SET 102

Describers	**Meaning**
DISP	Nodal displacements.
VELO	Nodal velocities.
ACCE	Nodal accelerations.
FORCE	Element output.
FREEBODY	Free body load.
GPFORCE	Element grid point forces.
MODES	Modal displacements.
POST	Element post-processing output.
REACT	SPC reactions.
STRESS	Nodal stress output.
ON / OFF	Activate / deactivate output.
COMPONENT	Requests specific number of components for output.
Num1	Number of components that are required.
SET	Select only those elements listed on SET Bulk Data entry to be included in output, or to be used during nodal data averaging for nodal stress output.
Num2	Set number for SET Bulk Data entry.
ASCI / BIN	ASCI or Binary file.

Remarks:

This is request for a separate textual file with UNA analysis output. The file is written in UNA "NEUTRAL" format, and provides third party programs with an easy access to the analysis output. The output can be directly imported to the spread-sheet programs, or to be used for graphic post-processing (deformed and contour plots, animation etc). UNA creates the following file

UNA72.NEU

in NEUTRAL format. Each line of text represents output for one node or element. This makes output file in the form of table with rows assigned to nodes or elements, and columns are filled with output data. Typical FORTRAN read (write) statement is as follows:

```
      READ(IN,100) (INEU(I),I=1,6),FQ,(COMP(I),I=1,41)
100   FORMAT(6I10,E15.7,41E15.7)
```

Columns 1-6 are integers that define set ID, output nature, node and element numbers, load case number, etc. Output components 1-41 are stored in columns 8-48. The appearance of typical NEUTRAL file is as follows:

```
10000    100    5031    11021     0     0   0.0000000E+00  -1.2345678E+05
10000    100   13013    11021     0     0   0.0000000E+00  -1.2345678E+05
90000    100    5031   121003     2     0   0.0000000E+00  -1.2345678E+05
90000    100    5031   112203     7     0   0.0000000E+00  -1.2345678E+05
```

The following table explains first 7 columns for all output sets. Set ID is given at Column 1.

Output Set	Col. 1 SID	Col. 2 sub I.D.	Col. 3	Col. 4	Col. 5	Col. 6	Col. 7
Comment	0						VER
Displacement	10,000	SOL	CASE	NODE	0	SYS	FQ
Velocity	20,000	SOL	CASE	NODE	0	SYS	FQ
Acceleration	30,000	SOL	CASE	NODE	0	SYS	FQ
Reaction	40,000	SOL	CASE	NODE	0	SYS	FQ
GP Forces	50,000	SOL	CASE	ELEM	NODE	SYS	FQ
Free Body Load	60,000	SOL	CASE	FBODY	NODE	SYS	FQ
Modes	70,000	SOL	MODE	NODE	0	SYS	FQ
Nodal Stress	80,000	SOL	CASE	NODE	0	0	FQ
Internal Loads	90,000	SOL	CASE	ELEM	ELTYPE	0	FQ
Post-Processing	100,000	SOL	CASE	ELEM	ELTYPE	0	FQ

where

SOL : Specifies nature of analysis output (subset number).
CASE / MODE : Load case ID or Mode ID. Total number of analysis sets is equal to sum of total number of load cases and total number of modes.

SOL	Description	CASE / MODE	FQ
= 100	Static analysis	Load case ID	
= 200	Modal analysis	Natural mode ID	Natural frequency [Hz]
= 300	Buckling analysis	Buckling mode ID	Eigenvalue
= 400	Transient response	Time step No.	Time step [sec]
= 500	Frequency response amplitude	Excitation frequency No.	Excitation [Hz]
= 600	Frequency response phase	Excitation frequency No.	Excitation [Hz]
= 700	Random vibration PSD	Excitation frequency No.	Excitation [Hz]
= 800	Random r.m.s. value		
= 900	Random apparent frequency		

VER : UNA version number.
NODE : Node identification number.
SYS : Node displacement system.
ELEM : Element identification number.
FBODY : Free Body identification number.
ELTYPE : Element code
1=CROD 2=CBEAM 3=CMEMB 4=CSOLID 5=CSHEAR
6=CSHELL17=CSHELL28=CMASS 9=CSUPEL 10=CELAS2
11=CBAR 12=CDAMP2 13=CVISC 14=CSTRIP 15=CTRIA3
16=CQUAD4 17=CSHELL3 18=CSHELL4

<u>Output Sets 10,000-70,000 : (Displacements - Modes, columns 8-13)</u>

Output is given in structure DISPLACEMENT coordinate system.

Output SET		Comp. 1	Comp. 2	Comp. 3	Comp. 4	Comp. 5	Comp. 6
10,000	Displacement	X1-tra	X2-tra	X3-tra	R1-rot	R2-rot	X3-rot
20,000	Velocity	X1-tra	X2-tra	X3-tra	R1-rot	R2-rot	X3-rot
30,000	Acceleration	X1-tra	X2-tra	X3-tra	R1-rot	R2-rot	X3-rot
40,000	Reaction	R1-force	R2-force	R3-force	M1-mom	M2-mom	M3-mom
50,000	GP Forces	P1-force	P2-force	P3-force	M1-mom	M2-mom	M3-mom
60,000	Free Body	P1-force	P2-force	P3-force	M1-mom	M2-mom	M3-mom
70,000	Modes	X1-tra	X2-tra	X3-tra	R1-rot	R2-rot	R3-rot

Output Set 80,000 : STRESS (Nodal stresses, columns 8 to 48)

- By default all 41 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be output in the file. This option allows file size reduction if only a first few components are of the interest.

- In order to find the nodal stresses UNA takes output from plane (CMEMB, CTRIA3, CQUAD4, CSHELL3, CSHELL4) and solid (CSOLID) elements, other element types are ignored. For particular node contributions from **all** adjacent elements mentioned above are averaged by default. This can be altered by specifying SET {Num2} option on the command line. Only output data from elements that are listed on that SET Bulk Data entry will be taken into account during averaging procedure.

- The following page explains component data. Table symbols are as follows:

σvmis	Equivalent Von-Misses stress.
σmax, σmin,τmax	Plane element principal normal and shear stresses.
σmax, σmed, σmin τmax, τmed, τmin	Solid element principal normal and shear stresses.
σx, σy, σz, τxy, τyz, τzx	Components of complete stress tensor. The components are output in BASIC coordinate system.
Tsai-Hill	Tsai-Hill criterion, max. value found through thickness at node
M.S.	Margin of Safety for composite material. Min. value found through plane element thickness at node.
top, middle, bottom	Plate element top, middle and bottom plane positions.

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared σvmis (max), and reported as Comp. 1.

- For multilayered composite material UNA calculates margins of safety for each ply, see MAT2 Bulk Data entry. Minimal value from all calculated margins for all plies through element thickness is declared M.S. (min), and reported as Comp. 3.

Output Set 80,000 : STRESS (Nodal stresses, columns 8 to 48)

Comp.	Meaning	Position
1	σvmis	Max
2	Tsai-Hill	Max
3	M.S.	Min
4		
5		
6	σvmis	top
7	σmax	top
8	σmin	top
9	τmax	top
10	σvmis	middle
11	σmax	middle
12	σmin	middle
13	τmax	middle
14	σvmis	bottom
15	σmax	bottom
16	σmin	bottom
17	τmax	bottom
18	σmax	solid
19	σmed	solid
20	σmin	solid

Comp.	Meaning	Position
21	τmax	solid
22	τmed	solid
23	τmin	solid
24	σx	top
25	σy	top
26	σz	top
27	τxy	top
28	τyz	top
29	τzx	top
30	σx	middle
31	σy	middle
32	σz	middle
33	τxy	middle
34	τyz	middle
35	τzx	middle
36	σx	bottom
37	σy	bottom
38	σz	bottom
39	τxy	bottom
40	τyz	bottom
41	τzx	bottom

Output Set 90,000 : FORCE (Element internal loads, columns 8 to 31)

- By default all 24 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be written, and output file size reduced.

- By default file contains output data for all elements. It can be altered by specifying SET {Num2} option on the command line. Only elements listed at SET will be included in the file.

- The following page explains component data. Table symbols are as follows:

σa	Axial stress for CROD, CBAR and CBEAM.
σvmis	Equivalent Von-Misses stress.
σx, σy, τxy	In-plane stresses, plane element local coordinate system.
σmax, σmin,τmax	In-plane principal stresses, plane element middle plane.
σx, σy, σz, τxy,τyz, τzx	Solid element stresses. SYS is equal to PARAM SOLID 3 = {Num} option that specifies coordinate system for stress components.
Tsai-Hill	Tsai-Hill criterion, max. value found through element thickness.
M.S.	Margin of Safety for composite material. Min. value found through plane element thickness at node.
Pa, Mt	Axial force and torsion moment.
M_{1A}, M_{1B}, V_{1A}, V_{1B}	CBAR plane 1 bending moment and shear force : end-A, end-B.
M_{2A}, M_{2B}, V_{2A}, V_{2B}	CBAR plane 2 bending moment and shear force : end-A, end-B.
Q21-Q41	CSHEAR panel shear flows for edges 1 to 4.
Nx, Ny, Nxy, Qx, Qy Mx, My, Mxy	Plane element unit loads in element local system. (force/moment/transverse shear per unit width).
A, K, B, T	Area, spring stiffness, damping factor, thickness.
Pload	Element distributed load (pressure, force/unit length).
X/C, C, B, A	CSTRIP element control point chorwise location, chord length, panel width, panel area.
Py, My, Z-press	CSTRIP element lift force, pitch moment at control point, element local coordinate system. Z-press is average pressure.
Cla, Cma, Clo, Cmo	CSTRIP element coefficients of lift and pitching moment.
top, middle, bottom	Plate element top, middle and bottom plane positions.

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared σvmis (max), and reported as Comp. 1.

Output Set 90,000 : FORCE (Element internal loads, columns 8 to 31)

Comp.	**CELAS2 CDAMP2**	**CROD CVISC**	**CBAR CBEAM**	**CSOLID**	**CSHEAR**	**2-D plane (.. CTRIA3…. .. CQUAD4 ..)**	**CSTRIP**
1		σa	σa	σvmis	σvmis	σvmis (max)	X/C
2						Tsai-Hill (max)	C
3						M.S. (min)	B
4	K (B)	A (B)	A	SYS	T	T	A
5			Pload	Pload	Pload	Pload	
6	Pa	Pa	Pa			Nx	Pz
7	Pa	Pa	Pa			Ny	My
8	Pa	Pa	Pa		Nxy	Nxy	Z-press
9	Mt	Mt	Mt			Mx	
10	Mt	Mt	M_{1A}			My	
11	Mt	Mt	M_{1B}			Mxy	
12		σa	V_{1A}		Q21	Qx	Cla
13		σa	V_{1B}		Q23	Qy	Cma
14		σa	M_{2A}		Q43	σmax (midd)	Clo
15			M_{2B}		Q41	σmin (midd)	Cmo
16			V_{2A}	σx		σx (top)	
17			V_{2B}	σy		σy (top)	
18				σz	τxy	τxy (top)	
19		σa	σa	τxy		σx (middle)	
20		σa	σa	τyz		σy (middle)	
21		σa	σa	τzx	τxy	τxy (middle)	
22						σx (bottom)	
23						σy (bottom)	
24					τxy	τxy (bottom)	

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared σvmis (max), and reported as Comp. 1.

- For multilayered composite material UNA calculates margins of safety for each ply, see MAT2 Bulk Data entry. Minimal value from all calculated margins for all plies through element thickness is declared M.S. (min), and reported as Comp. 3.

Output Set 100,000 : POST (Element post-processing, columns 8 to 47)

- By default all 40 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be written, and output file size reduced.

- By default file contains output data for all elements. It can be altered by specifying SET {Num2} option on the command line. Only elements that are listed on SET Bulk Data entry will be included in the file.

- Postprocesing data are based on CROD, CBAR and CBEAM output. UNA searches for all 2-D elements sharing the same line between two grids, and then collects grid point forces from line and all adjacent 2-D elements. For detailed explanation see POST Executive Data Entry.

- The following page explains components data. Table symbols are as follows:

ΔQ	Shear flow balancing axial forces at ends A and B.
Qmax	Maximal shear flow on any of 2-D adjacent elements.
σ_A, σ, σ_B	Axial stresses : end-A, midspan, end-B.
P_A, P, P_B	Axial forces : end-A, midspan, end-B.
M_{1A}, M_1, M_{1B}	Plane 1 bending moment : end-A, midspan, end-B.
M_{2A}, M_2, M_{2B}	Plane 2 bending moment : end-A, midspan, end-B.
V_{1A}, V_1, V_{1B}	Plane 1 shear force : end-A, midspan, end-B.
V_{2A}, V_2, V_{2B}	Plane 2 shear force : end-A, midspan, end-B.
Mt	Torsion moment.
Q21-Q41	Shear flows for edges 1 to 4.
P21-P41	Shear forces for edges 1 to 4.
A, L, Le	Cross section area, length between grids, length between ends.
T, L21-L41	Thickness, length for edges 1 to 4.

- Output is provided for two locations : grid points and element ends (may not be the coincident positions due to CBAR offset). For definition of the output coordinate systems see PARAM POST Executive Data Entry.

Output Set 100,000 : POST (Element post-processing, columns 8 to 47)

Comp.	CROD CBAR CBEAM		CMEMB CSHEAR CQUAD4 CSHELL4
	Output	Loc.	
1	A		T
2	L		L21
3	Le		L23
4	ΔQ	grid	L43
5	Qmax	grid	L41
6	σ_A	element	Q21
7	σ	element	Q23
8	σ_B	element	Q43
9	P_A	grid	Q41
10	P	grid	P21
11	P_B	grid	P23
12	M_{1A}	grid	P43
13	M_1	grid	P41
14	M_{1B}	grid	
15	V_{1A}	grid	
16	V_1	grid	
17	V_{1B}	grid	
18	M_{2A}	grid	
19	M_2	grid	
20	M_{2B}	grid	

Comp.	CROD CBAR CBEAM		CMEMB CSHEAR CQUAD4 CSHELL4
	Output	Loc.	
21	V_{2A}	grid	
22	V_2	grid	
23	V_{2B}	grid	
24	Mt	grid	
25	P_A	element	
26	P	element	
27	P_B	element	
28	M_{1A}	element	
29	M_1	element	
30	M_{1B}	element	
31	V_{1A}	element	
32	V_1	element	
33	V_{1B}	element	
34	M_{2A}	element	
35	M_2	element	
36	M_{2B}	element	
37	V_{2A}	element	
38	V_2	element	
39	V_{2B}	element	
40	Mt	element	

OUTPUT

Global stiffness and mass matrix output.

Format:

OUTPUT	GSTIFF GMASS	=	ASCI BINARY NONE

Examples:

OUTPUT GSTIFF = ASCI
OUTPUT GMASS = BINARY

Describers	**Meaning**
GSTIFF	Global stiffness matrix
GMASS	Global mass matrix
ASCI	ASCI output
BINARY	Binary output
NONE	No output

Remarks:

Global stiffness and mass matrices are exported in ASCI or BINARY format. Upper triangle of each matrix is written in Skyline format as a vector with matrix data stored column-wise from the first non-zero term in the column down to the element on matrix diagonal.

Example: 6X6 matrix upper triangle with elements Sij

```
S11  S12  S13    0    0    0
     S22  S23  S24    0  S26
          S33  S34    0  S36
               S44  S45  S46
                    S55  S56
                         S66
```

Matrix upper triangle stored in skyline vector A

```
 A1   A2   A4
      A3   A5   A7        A12
           A6   A8        A13
                 A9  A10  A14
                     A11  A15
                          A16
```

Vector JD defines locations of diagonal elements in vector A

```
JD(1)=1
JD(2)=3
JD(3)=6
JD(4)=9
JD(5)=11
JD(6)=16
```

Stiffness is written to external file GSTIFF.DAT or GSTIFF.BIN and mass is written in GMASS.DAT or GMASS.BIN.

The following is FORTRAN template example for stiffness ASCI data extraction:

```
   IMPLICIT INTEGER*4 (I-N)
   IMPLICIT REAL*8    (A-H,O-Z)
   DIMENSION          JD(*),ASTIF(*)
C
   OPEN (UNIT=1,FILE='GSTIFF.DAT',FORM='FORMATTED')
   READ (1,10) NEQ,NBLOCK
   READ (1,10) (JD(N),N=1,NEQ)
   DO NB=1,NBLOCK
     READ (1,20) N1,N2
     READ (1,20) (ASTIF(N),N=N1,N2)
   ENDDO
10 FORMAT (8I15)
20 FORMAT (8(1PE15.7))
```

The following is FORTRAN template example for mass BINARY data extraction:

```
   IMPLICIT INTEGER*4 (I-N)
   IMPLICIT REAL*8    (A-H,O-Z)
   DIMENSION          JD(*),AMASS(*)
C
   OPEN (UNIT=2,FILE='GMASS.BIN',FORM='UNFORMATTED')
   READ (2) NEQ,NBLOCK
   READ (2) (JD(N),N=1,NEQ)
   DO NB=1,NBLOCK
     READ (2) N1,N2
     READ (2) (AMASS(N),N=N1,N2)
   ENDDO
```

where:	NEQ	number of equations (matrix size)
	JD(NEQ)	positions of matrix diagonal elements

Requests page numbering for the output file.

Format:

PAGES = | n |

Examples:

PAGES = 0
PAGES = 10

Describers	Meaning	
n = 0	Disables page numeration.	
= Num	Enables page numeration starting from Num	(default = 1)

Remarks:

1. Disabling page numeration by

   PAGES = 0

   page headers are disabled as well . Only first header for each output type appears in the output file (for displacements, stresses etc.). This allows that output data file can be easily imported to the spreadsheet program for further processing.

PARAM

Controls element internal load and stress output type, location, and coordinate system . It controls element type and usage of incompatible modes for selected elements as well.

Format:

PARAM	BAR BEAM STRIP MEMB SOLID SHEAR PLATE SUPEL POST		Num1	=	Num2

Examples:

PARAM BAR {1} = 2
PARAM PLATE {2} = 9

Describers	**Meaning**
BAR	Bar parameters.
BEAM	Beam parameters.
STRIP	Strip parameters.
MEMB	Membrane parameters.
SOLID	Solid element parameters.
SHEAR	Shear panel parameters.
PLATE	Plate element parameters.
SUPEL	Superelement parameters.
POST	Post-processing parameters.
Num1	Parameter type (Integer).
Num2	Option (Integer)

Remarks:

This Executive Data Entry controls element internal load and stress output type, location, coordinate system etc. It is used to control element type and usage of the incompatible modes for selected elements as well (ex: CSHELL4). Parameter type (Num1) and option (Num2) are element depended. The following pages provide full explanation for each element type.

PARAM BAR - Controls bar internal load output

- Option for output type

PARAM BAR 1	=	1	Internal load, beam convention, full output (default)
		2	Internal load, beam convention, X-Y plane
		3	Internal load, beam convention, X-Z plane
		4	Internal load, element convention
		5	Stresses at defined points

- Option for internal load location and output coordinate system

PARAM BAR 2	=	1	Grid points (G)
		2	Element ends (E) (default)

If grid points location is required load components are outputted in the coordinate system defined as follows:

X-axis : vector from grid point GA to grid point GB.
X-Y plane : defined with X-axis and element orientation vector V.
Z-axis : perpendicular to X-Y plane, right hand system.

If element end points location is required, (may be offset from grid points), element local coordinate system is used for outputting the load components. For definition of bar element local coordinate system, end points offset, grid points, and orientation vector V, see CBAR Bulk Data Entry.

PARAM BEAM - Controls beam internal load output

- Option for output type

PARAM	BEAM	=	1	Internal load, element convention	(default)
			2	Stresses at defined points	

PARAM STRIP - Controls strip element nonstationary correction

- Option for Theodorsen nonstationary aerodynamic correction C(k)

PARAM	STRIP	1	=	1	C(k) = 1.0 (constant)	
				2	C(k) = F(ωc/2U)	(default)

This option controls the use of quasi-steady C(k)=1.0 aerodynamic, or nonstationary Theodorsen correction that is function of reduced frequency.

PARAM MEMB - Controls membrane element output and element options

- Option for stress output type

PARAM MEMB 1	=	1	σx, σy, τxy, σvmis/Tsai	(default)
		2	σmax, σmin, τmax, σvmis/Tsai	
		3	εx, εy, γxy, M.S.	
		4	εmax, εmin, γxymax, M.S.	
		5	Nx, Ny, Nxy, σvmis/Tsai	
		6	Nmax, Nmin, Nxymax, σvmis/Tsai	

where are:

	Isotropic material	Orthotropic (Multilayered) material
σx, σy, τxy	Stresses in element local system	Stresses in material 1-2 system
σvmis/Tsai	Equivalent Von Misses stress	Tsai-Hill Criterion value
εx, εy, γxy	Strains in element local system	Strains in material 1-2 system
M.S.	Strain based Margin of Safety	Strain based Margin of Safety
Nx, Ny, Nxy	Unit forces	Unit forces

- Option for selecting coordinate system for unit forces (Nx,Ny,Nxy)

PARAM MEMB 2	=	1	Local element X'Y' system		(default)
		2	Local element X'Y' system	(isotropic)	
			Material 1-2 system	(orthotropic)	
			System 0-90	(multilayered)	

- Option for stress output location

PARAM MEMB 3	=	1	At element centre (C.G.)	(default)
		2	At midpoints of the edges	
		3	At centre + midpoints of the edges	
		4	At nodes	

- Option for use of incompatible modes

PARAM MEMB 4	=	1	Yes	(default)
		2	No	

PARAM SOLID - Controls solid element output and element options

- Option for stress output location

PARAM SOLID 1 =	1	At element centre (C.G.)	(default)
	2	At midpoints of the sides	
	3	At centre + midpoints of the sides	
	4	At nodes	

- Option for stress output type

PARAM SOLID 2 =	1	σx, σy, σz, σvmis	(default)
	2	τxy, τyz, τzx, σvmis	
	3	σmax, σmed, σmin, σvmis	
	4	τmax, τmed, τmin, σvmis	
	5	σx, σy, σz, τxy, τyz, τzx, σvmis	
	6	σmax,σmed,σmin,τmax,τmed, τmin, σvmis	

- Option for stress output coordinate system

		Isotropic mat	Orthotropic mat.	
PARAM SOLID 3 =	1	basic XYZ	material 123	(default)
	2	local X'Y'Z'	material 123	
	3	basic XYZ	basic XYZ	
	4	local X'Y'Z'	basic XYZ	
	5	basic XYZ	local X'Y'Z'	
	6	local X'Y'Z'	local X'Y'Z'	

- Option for use of incompatible modes

PARAM SOLID 4 =	1	Yes	(default)
	2	No	

<u>PARAM SHEAR - Controls shear panel output and element options</u>

- Option for generating the rods on element edges 1 and 3

PARAM SHEAR 1	=	0	Do not generate rod elements	(default)
		1	Generate rod elements	

- Option for generating the rods on element edges 2 and 4

PARAM SHEAR 2	=	0	Do not generate rod elements	(default)
		1	Generate rod elements	

- Option for generated rods output type

PARAM SHEAR 3	=	0	No output	(default)
		1	Force	
		2	Stress	
		3	Area	
		4	Force, Stress and Area	

PARAM PLATE - Controls plate elements output and element options

- Option for multilayered composite element type

PARAM	PLATE	1	=	1	Shell (in-plane + bending stiffness)	(default)
				2	Membrane (in-plane stiffness only)	
				3	Plate (bending stiffness only)	

- Option for output type

PARAM	PLATE	2	=	1	σx, σy, τxy, σvonmis	(default)
				2	εx, εy, γxy, εvonmis	
				3	Nx, Ny, Nxy, Mx, My, Mxy, Qx, Qy	
				4	σmax, σmin, τmax, Angle	
				5	εmax, εmin, γmax, Angle	
				6	Nmax, Nmin, Nxymax, Angle	
				7	Mmax, Mmin, Mxymax, Angle	
				8	σ1, σ2, τ12, M.S. (composite plies)	
				9	ε1, ε2, γ12, M.S. (composite plies)	

where are:

σx, σy, τxy, σvonmis	Stresses in element local system. Equivalent Von Misses stress.
σ1, σ2, τ12, M.S.	Stresses in material 1-2system. Ply Margin of Safety.
εx, εy, γxy, εvonmis	Strains in element local system. Equivalent Von Misses strain.
ε1, ε2, γ12, M.S.	Strains in material 1-2system. Ply Margin of Safety.
Nx, Ny, Nxy, Mx, My, Mxy, Qx,Qy	Unit force, moment and transverse shear (load per unit width).
max, min	Principal values.
Angle	Angle between element x-axis and max principal direction.

- Option for stress and strain surface location

PARAM	PLATE	3	=	1	Middle	
				2	Bottom, Top	(default)
				3	Bottom, Middle, Top	

PARAM PLATE - Controls plate elements output and element options

- Option for in-plane output location

PARAM	PLATE	4	=	1	At element centre (C.G.)	(default)
				2	At midpoints of the edges	
				3	At centre + midpoints of the edges	
				4	At nodes	

- Option for selection coordinate system for unit loads (Nx,Ny,..,Qy)

PARAM	PLATE	5	=	1	Local element XY system		(default)
				2	Local element XY system	(isotropic)	
					Material 1-2 system	(orthotropic)	
					Laminate 0-90 system	(multilayered)	

- Option for use of incompatible modes (CSHELL4 quadrilateral element)

PARAM	PLATE	6	=	1	Yes	(default)
				2	No	

PARAM SUPEL - Controls superelement output and element options

- Option for stress output type

PARAM	SUPEL	1	=	1	Internal forces acting from main structure to superelement	(default)
				2	Internal moments acting from main structure to superelement	
				3	Internal forces and moments acting from main structure to superelement	
				4	Internal forces acting from superelement to main structure	
				5	Internal moments acting from superelement to main structure	
				6	Internal forces and moments acting from superelement to main structure	

- Option for creating UNA.SUP file containing **forces** acting from main structure to superelement

PARAM	SUPEL	2	=	0	Do not create file	(default)
				1	Create file	

- Option for creating UNA.SUP file containing **moments** acting from main structure to superelement

PARAM	SUPEL	3	=	0	Do not create file	(default)
				1	Create file	

- Option for creating UNA.SUP file containing **forces** acting from superelement to main structure

PARAM	SUPEL	4	=	0	Do not create file	(default)
				1	Create file	

- Option for creating UNA.SUP file containing **moments** acting from superelement to main structure

PARAM	SUPEL	5	=	0	Do not create file	(default)
				1	Create file	

PARAM SUPEL - Controls superelement output and element options (con't)

- Option for creating GENEL.SUP file containing superelement stiffness matrix and applied load reduced to the superelement interface nodes (see SOL = 4 solution type) . Stiffness matrix is written in GENEL format (NASTRAN general element Bulk Data entry).

PARAM	SUPEL	6	=	0	Do not create file	(default)
				1	Create file	

The following are definitions for internal load output coordinate systems:

1. Internal forces and moments acting from main structure to superelement are output in superelement BASIC coordinate system.

2. Internal forces and moments acting from superelement to main structure are output in main structure DISPLACEMENT coordinate systems (see IDIS on the GRID Bulk Data entry)

Output data file UNA.SUP contains nodal forces and moments in the UNA format. Forces and moments are printed as Bulk Data entries FORCE and MOMENT. Interaction between superelements and main structure (and vice versa), can be substituted by forces and moments as defined in UNA.SUP data file.

PARAM POST - Controls post-processing output

- Option for critical search type

PARAM	POST 1	=	1	All load cases	(default)
			2	Max/Min	
			3	Max/Min, load case	

- Option for 2-D elements post-processing output

PARAM	POST 2	=	1	Shear flows at all edges	(default)
			2	Shear forces at all edges	

- Option for bar and beam full or reduced post-processing output (beam convention)

PARAM	POST 3	=	1	Internal load, full output	(default)
			2	Internal load, X-Y plane	
			3	Internal load, X-Z plane	

- Option for bar post-processing output location and coordinate system

PARAM	POST 4	=	1	Grid points (G)	(default)
			2	Element ends (E)	

If grid points location is required, load components are output in the coordinate system defined as follows:

X-axis : vector from grid point GA to grid point GB.
X-Y plane : defined with X-axis and element orientation vector V.
Z-axis : perpendicular to X-Y plane, right hand system.

If element end points location is required, (may be offset from grid points), element local coordinate system is used for outputting the load components. For definition of bar element local coordinate system, end points offset, grid points, and orientation vector V, see CBAR Bulk Data Entry.

Requests 1-D and 2-D elements output post-processing and printout.

Format:

POST =	ALL NONE RANGE SET	Num1 Num2	CRITICAL	Real OFF

Examples:

POST = SET 101 CRITICAL 1.

Describers	Meaning
ALL	Selects all elements
NONE	Deselect all elements.
RANGE	Selects elements range.
SET	Selects elements as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Elements range if coming after RANGE.
CRITICAL	Enables round-off of the output data.
Real	Round-off value.
OFF	Disables output round-off.

Remarks:

1. In the case of four-node 2-D elements (CMEMB, CSHEAR, CQUAD4 and CSHELL4), program calculates and prints shear flows or shear forces at element edges.

2. In the case 1-D elements (CROD, CBAR and CBEAM), program searches for all 2-D elements sharing the same line between two grids, and then collects grid point forces from line element and all adjacent 2-D elements.

3. Round-off procedure has effect only on the data written to FEMAP files. The following is example of the round-off procedure:

   CRITICAL = 1., R = 34567.9376 $\Rightarrow$ R = 34568.

4. For controlling the post-processing output, see PARAM POST Executive Data Entry.

PATRAN

Requests and controls output in PATRAN file format.

Format:

PATRAN	DISP FORCE FREEBODY MODES REACT STRESS	=	ON OFF	COMPONENT	Num1	SET	Num2

Examples:

PATRAN = ON
PATRAN FORCE = ON SET 101

Describers	Meaning
DISP	Displacement output file.
FORCE	Element output file.
FREEBODY	Free body load file.
MODES	Modal displacements file.
REACT	SPC reactions file.
STRESS	Nodal output file.
ON / OFF	Activate / deactivate output.
COMPONENT	Requests specific number of components for FORCE and STRESS output files.
Num1	Number of components that are required.
SET	Select only those elements listed on SET Bulk Data entry to be included in element output, or to be used during nodal data averaging for nodal output
Num2	Set number for SET Bulk Data entry.

Remarks:

This entry directs UNA to provide additional files containing analysis output in the PATRAN format. These files are intended to be used for postprocesing, i.e. deformed, contour and criteria plots, SPC reactions and Free Body load visualization with the PATRAN, FEMAP or another postprocesing software capable to read and process PATRAN type files.

UNA creates the following files with analysis output in the PATRAN file format:

File Name	Output type	PATRAN file type
UNA_RXN.PAT	SPC reactions	Model data - force
UNA_FBDY.PAT	Free body load	Model data - force
UNA_SUB*.DIS	Nodal displacements	Displacement
UNA_MOD*.DIS	Modal displacements	Displacement
UNA_SUB*.NOD	Nodal stresses	Nodal
UNA_SUB*.ELS	Element internal load and stresses	Element

where * is standing for load case or mode number.

REACT : SPC reactions file (UNA-RXN.PAT)

File has to be imported into postprocessor as a model data file. SPC reactions are output for all load cases in the UNA analysis run. Nodal forces and moments are output in structure BASIC coordinate system as a new load case with a label offset of 100,000, i.e. if the applied load case was 101, reactions are returned as a load case 100,101. Selective output is controlled with REACT = CRITICAL {Real}, see REACT Executive Data entry.

FREEBODY : Free body load file (UNA-FBDY.PAT)

File has to be imported into postprocessor as a model data file. Free body load is output for all load cases in the UNA analysis run. Only the first free body, with lowest FID, is considered (if more than one is specified in the input file). Nodal forces and moments are output in structure BASIC coordinate system as a new load case with a label offset of 200,000, i.e. if the applied load case was 101, free body load is returned as a load case 200,101. Selective output is controlled by Executive Data Entry FREEBODY = CRITICAL {Real}.

DISP : Displacement file (UNA-SUB*.DIS)

There is one displacement file for each load case in the UNA analysis run. Displacements are output in structure BASIC coordinate system.

MODES : Modal displacement file (UNA-MOD*.DIS)

There is one modal displacement file for each mode in the UNA analysis run. Modal displacements are output in structure BASIC coordinate system.

STRESS : Nodal output file (UNA-SUB*.NOD)

- There is one nodal output file for each load case in the UNA analysis run.

- By default all 41 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be output in the file. This option allows file size reduction if only a first few components are of the interest.

- In order to find the nodal stresses UNA takes output from plane (CMEMB, CTRIA3, CQUAD4, SHELL3, CSHELL4) and solid (CSOLID) elements, other element types are ignored. For particular node contributions from **all** adjacent elements mentioned above are averaged by default. This can be altered by specifying SET {Num2} option on the command line. Only output data from elements that are listed on that SET Bulk Data entry will be taken into account during averaging procedure.

- The following page explains component data. Table symbols are as follows:

σvmis	Equivalent Von-Misses stress.
σmax, σmin, τmax	Plane element principal normal and shear stresses.
σmax, σmed, σmin τmax, τmed, τmin	Solid element principal normal and shear stresses.
σx, σy, σz, τxy, τyz, τzx	Components of complete stress tensor. The components are output in BASIC coordinate system.
Tsai-Hill	Tsai-Hill criterion, max. value found through thickness at node.
M.S.	Margin of Safety for composite material. Min. value found through plane element thickness at node.
top, middle, bottom	Plate element top, middle and bottom plane positions.

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared σvmis (max), and reported as Comp. 1.

- For multilayered composite material UNA calculates margins of safety for each ply, see MAT2 Bulk Data entry. Minimal value from all calculated margins for all plies through element thickness is declared M.S. (min), and reported as Comp. 3.

STRESS : Nodal output file (UNA-SUB*.NOD)

Comp.	Meaning	Position
1	σvmis	Max
2	Tsai-Hill	Max
3	M.S.	Min
4		
5		
6	σvmis	top
7	σmax	top
8	σmin	top
9	τmax	top
10	σvmis	middle
11	σmax	middle
12	σmin	middle
13	τmax	middle
14	σvmis	bottom
15	σmax	bottom
16	σmin	bottom
17	τmax	bottom
18	σmax	solid
19	σmed	solid
20	σmin	solid

Comp.	Meaning	Position
21	τmax	solid
22	τmed	solid
23	τmin	solid
24	σx	top
25	σy	top
26	σz	top
27	τxy	top
28	τyz	top
29	τzx	top
30	σx	middle
31	σy	middle
32	σz	middle
33	τxy	middle
34	τyz	middle
35	τzx	middle
36	σx	bottom
37	σy	bottom
38	σz	bottom
39	τxy	bottom
40	τyz	bottom
41	τzx	bottom

FORCE : Element output file (UNA-SUB*.ELS)

- There is one element output file for each load case in the UNA analysis run.
- By default all 24 components (columns) with output data are written into the output file. It can be altered by specifying COMPONENT {Num1} option on the command line. Only components up to **Num1** number will be written, and output file size reduced.
- By default file contains output data for all elements. It can be altered by specifying SET {Num2} option on the command line. Only elements listed at SET will be included in the file.
- The following page explains component data. Table symbols are as follows:

σa	Axial stress for CROD, CBAR and CBEAM.
$\sigma vmis$	Equivalent Von-Misses stress.
σx, σy, τxy	In-plane stresses, plane element local coordinate system.
σmax, σmin,τmax	In-plane principal stresses, plane element middle plane.
σx, σy, σz, τxy,τyz, τzx	Solid element stresses. SYS is equal to PARAM SOLID 3 = {Num} option that specifies coordinate system for stress components.
Tsai-Hill	Tsai-Hill criterion, max. value found through element thickness.
M.S.	Margin of Safety for composite material. Min. value found through plane element thickness at node.
Pa, Mt	Axial force and torsion moment.
M_{1A}, M_{1B}, V_{1A}, V_{1B}	CBAR plane 1 bending moment and shear force : end-A, end-B.
M_{2A}, M_{2B}, V_{2A}, V_{2B}	CBAR plane 2 bending moment and shear force : end-A, end-B.
Q21-Q41	CSHEAR panel shear flows for edges 1 to 4.
Nx, Ny, Nxy, Qx, Qy Mx, My, Mxy	Plane element unit loads in element local system. (force/moment/transverse shear per unit width).
A, K, B, T	Area, spring stiffness, damping factor, thickness.
Pload	Element distributed load (pressure, force/unit length).
X/C, C, B, A	CSTRIP element control point chorwise location, chord length, panel width, panel area.
Py, My, Z-press	CSTRIP element lift force, pitch moment at control point, element local coordinate system. Z-press is average pressure.
Cla, Cma, Clo, Cmo	CSTRIP element coefficients of lift and pitching moment.
top, middle, bottom	Plate element top, middle and bottom plane positions.

FORCE : Element output file (UNA-SUB*.ELS)

Comp.	**CELAS2 CDAMP2**	**CROD CVISC**	**CBAR CBEAM**	**CSOLID**	**CSHEAR**	**2-D plane (.. CTRIA3…. .. CQUAD4 ..)**	**CSTRIP**
1		σa	σa	σvmis	σvmis	σvmis (max)	X/C
2						Tsai-Hill (max)	C
3						M.S. (min)	B
4	K (B)	A (B)	A	SYS	T	T	A
5			Pload	Pload	Pload	Pload	
6	Pa	Pa	Pa			Nx	Pz
7	Pa	Pa	Pa			Ny	My
8	Pa	Pa	Pa		Nxy	Nxy	Z-press
9	Mt	Mt	Mt			Mx	
10	Mt	Mt	M_{1A}			My	
11	Mt	Mt	M_{1B}			Mxy	
12		σa	V_{1A}		Q21	Qx	Cla
13		σa	V_{1B}		Q23	Qy	Cma
14		σa	M_{2A}		Q43	σmax (midd)	Clo
15			M_{2B}		Q41	σmin (midd)	Cmo
16			V_{2A}	σx		σx (top)	
17			V_{2B}	σy		σy (top)	
18				σz	τxy	τxy (top)	
19		σa	σa	τxy		σx (middle)	
20		σa	σa	τyz		σy (middle)	
21		σa	σa	τzx	τxy	τxy (middle)	
22						σx (bottom)	
23						σy (bottom)	
24					τxy	τxy (bottom)	

- For isotropic material UNA calculates equivalent Von Misses stress for top, middle and bottom plate element plane. Maximal value is declared σvmis (max), and reported as Comp. 1.

- For multilayered composite material UNA calculates margins of safety for each ply, see MAT2 Bulk Data entry. Minimal value from all calculated margins for all plies through element thickness is declared M.S. (min), and reported as Comp. 3.

PSDRESPONSE

Requests and controls auto - psd response output.

Format:

PSDRESP	DISP VELO ACCE ROD BAR BEAM MEMB SOLID SHEAR PLATE CELAS2 DAMP2 VISC STRIP	=	ALL NONE RANGE SET	Num1 Num2	COMP	Num1 Num2	SUMM FULL

Examples:

PSD BAR = SET 101 COMP 10 12
PSD STRIP = ALL FULL

Describers	**Meaning**
DISP	Nodal displacements.
VELO	Nodal velocities.
ACCE	Nodal accelerations.
ROD - STRIP	Element type selection. If blank all types are selected.
ALL	Selects all entities.
NONE	Deselect all entities.
RANGE	Selects entity range.
SET	Selects entities as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Node range if coming after RANGE.
COMPONENT	Select entity components
Num1 - Num2	Component range if coming after COMP.
SUMM	Summary (root-mean-square) output only.
FULL	Full auto-psd and summary output required.

Remarks:

1. The nodal displacements, velocities and accelerations are output for all forcing frequencies as specified at selected FREQ, FREQ1 and FREQ2 entries. The components 1 - 3 are translations and components 4 - 6 are rotations in the grid displacement system.

2. Element component data are explained in the table on the following page. The following is description of the table symbols:

σa	Axial stress for CROD, CBAR and CBEAM.
σx, σy, τxy	In-plane stresses, plane element local coordinate system.
σx, σy, σz, τxy, τyz, τzx	Solid element stresses. Components are output in coordinate system specified by PARAM SOLID 3 = {Num}.
Pa, Mt	Axial force and torsion moment.
M_{1A}, M_{1B}, V_{1A}, V_{1B}	CBAR plane 1 bending moment and shear force : end-A, end-B.
M_{2A}, M_{2B}, V_{2A}, V_{2B}	CBAR plane 2 bending moment and shear force : end-A, end-B.
Q21-Q41	CSHEAR panel shear flows for edges 1 to 4.
Nx, Ny, Nxy, Qx, Qy Mx, My, Mxy	Plane element unit loads in element local system. (force/moment/transverse shear per unit width).
Py, My, Z-press	CSTRIP element lift force, pitch moment at control point, element local coordinate system. Z-press is average pressure.
top, middle, bottom	Plate element top, middle and bottom plane positions.

Comp.	CELAS2 CDAMP2	CROD CVISC	CBAR CBEAM	CSOLID	CSHEAR	2-D plane (.. CTRIA3…. .. CQUAD4 ..)	CSTRIP
1							
2							
3							
4							
5							
6	Pa	Pa	Pa			Nx	Pz
7	Pa	Pa	Pa			Ny	My
8	Pa	Pa	Pa		Nxy	Nxy	Z-press
9	Mt	Mt	Mt			Mx	
10	Mt	Mt	M_{1A}			My	
11	Mt	Mt	M_{1B}			Mxy	
12		σ_a	V_{1A}		Q21	Qx	
13		σ_a	V_{1B}		Q23	Qy	
14		σ_a	M_{2A}		Q43		
15			M_{2B}		Q41		
16			V_{2A}	σ_x		σ_x (top)	
17			V_{2B}	σ_y		σ_y (top)	
18				σ_z	τ_{xy}	τ_{xy} (top)	
19		σ_a	σ_a	τ_{xy}		σ_x (middle)	
20		σ_a	σ_a	τ_{yz}		σ_y (middle)	
21		σ_a	σ_a	τ_{zx}	τ_{xy}	τ_{xy} (middle)	
22						σ_x (bottom)	
23						σ_y (bottom)	
24					τ_{xy}	τ_{xy} (bottom)	

Selects the RANDPS Bulk Data Entry to be used in random vibration analysis.

Format:

RANDOM = | n |

Examples:

RANDOM = 101

Describers	Meaning
n	Set identification of a RANDPS Bulk Data Entry. (Integer > 0).

Remarks:

1. A selected RANDPS power spectral density set is applied during the random vibrations analysis (SOL = 8,9).

REACTIONS

Requests SPC reactions printout in the output file.

Format:

REACTIONS	FORCE MOMENT	=	ALL NONE RANGE SET	Num1 Num2	SPC FULL	CRITICAL	Real OFF

Examples:

REACT = ALL
REACT MOMENT = SET 101 FULL

Describers	**Meaning**
FORCE	Force selected. If blank both force and moment are selected.
MOMENT	Moment selected. If blank both force and moment are selected.
ALL	Selects all nodes.
NONE	Deselect all nodes.
RANGE	Selects node range.
SET	Selects nodes as specified at SET Bulk Data entry.
Num1	Set number if coming after SET.
Num1 - Num2	Node range if coming after RANGE.
SPC	SPC reactions requested.
FULL	SPC reactions and MPC, COUPG, RBE2 reactions requested.
CRITICAL	Alter default tolerance for selective printout.
Real	Minimum force (moment) value if selective output is enabled. Grids with force components larger than defined value will be printed in output file.
OFF	Disables selective output.

Remarks:

1. When command does not include any of available options FORCE or MOMENT, i.e.

   REACT = ALL

   then it is related to both forces and moments.

2. Reaction results are output in DISPLACEMENT coordinate system (see IDIS on the GRID Bulk Data entry).

Specify time span in transient analysis.

Format:

RESPONSE = | Real1 Real2 |

Examples:

RESP = 0.8 15.
RESP = 0. 200.

Describers	Meaning	
Real1	Starting time	(default = 0.0)
Real2	Ending time	(default = 10.0)

Remarks:

1. This entry specifies first and last value for span definition. For the transient response solution type, Real1 represents starting analysis time, and Real2 represents solution ending time. Defined interval is divided into INC equal parts (see INCREMENTS).

SDAMPING

Defines modal damping in frequency response, random vibrations and dynamic gust analysis.

Format:

SDAMPING	G Q CRIT	=	Real TABLE	Num	STRUCTURAL	ON OFF

Examples:

SDAMPING G = 0.02 STRUCTURAL ON
SDAMPING Q = TABLE 12

Describers	**Meaning**
G	Two times critical damping ratio G = 2∗C/Co
Q	Amplification quality factor Q = 1/G
CRIT	Critical damping ratio CRIT = C/Co = G/2
Real	Damping value. (Real > 0.).
TABLE	TABLE Bulk Data Entry is used do define damping over frequency range.
STRUCTURAL	Complex structural damping.
ON	Complex structural damping enabled.
OFF	Complex structural damping disabled.

Remarks:

1. By default modal damping is incorporated in the viscous damping form i.e.

$$i \cdot \Omega \cdot S_{2hh} = i \cdot \Omega \cdot \frac{g_h}{\omega_h} k_h = i \cdot \Omega \cdot g_h \omega_h m_h$$

   where Ω is excitation frequency and ω_h undamped vibration frequency of the h-th mode in [rad/sec]. STRUCTURAL ON command incorporates damping in the form of complex structural damping, i.e.

$$i \cdot S_{3hh} = i \cdot g_h k_h = i \cdot g_h \omega_h^2 m_h$$

2. TABLE Bulk Data entry is used to interpolate damping over the frequency range. If TABLE is skipped, the specified damping value is applied over the whole frequency range.

Requests stiffness matrix shifting by specifying shifting frequency in [Hz] (SOL=2,5,6,8,9), or by specifying eigenvalue shift (SOL=3).

Format:

SHIFT = | Real |

Examples:

SHIFT = -3.14
SHIFT = 100.

Describers	**Meaning**	
Real	Shifting value.	(default=0.0)

Remarks:

1. In free vibration based solution mass matrix is multiplied by shifting frequency in [rad/sec] and added to stiffness matrix

   $K' = K + \omega M$

   where $\omega=2\pi f$. f represents frequency in [Hz] defined by SHIFT command. Shifting frequency can be positive or negative. Default value is SHIFT = 0. (no shifting).

2. In elastic buckling analysis geometric stiffness matrix is multiplied by shifting eigenvalue and added to stiffness matrix

   $K' = K + \lambda K_\sigma$

   where λ represents value defined by SHIFT command. Shifting eigenvalue can be positive or negative. Default value is SHIFT = 0.0 (no shifting).

SOFTEXIT

Analysis run soft exit.

Format:

SOFTEXIT = | n |

Examples:

SOFTEXIT = 0
SOFTEXIT = 2

Describers		**Meaning**	
n	= 0	No soft exit	(default)
	= 1	Exit after individual elements processing	
	= 2	Exit after global stiffness matrix assembly	
	= 3	Exit after global mass matrix assembly	

Remarks:

1. This card defines soft exit options for aborting run.

Specify a Finite Element Analysis type.

Format:

SOLUTION = | n |

Examples:

SOL = 0
SOL = 2

Describers		Meaning	
n	= 0	Input data check	(default)
	= 1	Linear static analysis	
	= 2	Modal analysis	
	= 3	Buckling analysis	
	= 4	Superelement generation	
	= 5	Transient response	
	= 6	Frequency response	
	= 8	Random vibrations	
	= 9	Aeroelastic response	

Remarks:

2. This card defines structural analysis solution type. If omitted, program performs default procedure (input data checking).

SORT

Ordering of symmetric sparse matrix for small profile and waterfront or for small bandwidth.

Format:

SORT	=	PROFILE GBS BAND SEQGP X1+ X2- X3+ OFF	Num

Examples:

SORT = PROFILE
SORT = SEQGP
SORT = X2+ X1+ X3+ {101}

Describers	Meaning
PROFILE	Profile reduction by Sloan algorithm (default).
GBS	Profile reduction by Gibbs-Poole-Stockmeyer algorithm.
BAND	Bandwidth reduction by Gibbs-King algorithm.
SEQGP	Re-order grids via SEQGP Bulk Data Entries.
X1+ X2+ X3+ X1- X2- X3-	Re-order grids by coordinate axes method. Defines priority and direction of sorting.
OFF	Disable sorting
Num	Specify sorting coordinate system. If Num = 0, or does not exist, grid sorting is performed in BASIC coordinate system.

Remarks:

1. Grid points sorting helps to reduce stiffness matrix profile and to increase solution speed. The default (PROFILE) produces the best profile reduction in most cases.

2. If axes method is used, then it is recommended to select sorting axes in a fashion that longest structure dimension is listed first, and shortest last. For example if structure Z length is the longest and Y is the shortest, the following is the recommended sorting:

   SORT = X3+ X1+ X2+

   where X1, X2, X3 stand for X, Y, Z , and + sign specify ascending order.

Selects a single-point constraint set to be applied.

Format:

SPC = | n |

Examples:

SPC = 10

Describers	**Meaning**
n	Set identification number of a single-point constraint that appears on a SPC or SPC1 Bulk Data entry (Integer > 0)

SUBCASE

Selects a static subcase in static or buckling analysis.

Format:

SUBCASE =	n

Examples:

SUBCASE = 101

Describers	Meaning
n	Subcase identification number. (Integer > 0).

Remarks:

1. Selects a static subcase by referring to SID number as appears on FORCE*, MOMENT*, PLOAD*, QLOAD*, QPRESS, AVECTOR, ROTATIO, TEMP* and AESTAT Bulk Data Entries.

Selects a file name for generated superelement

Format:

SUPNAME = "Supname"

Examples:

SUPNAME = WING.SUP
SUPNAME = "SECT_44.SUP"

Remarks:

1. This entry specifies a file name assigned to generated superelement. At the end of solution type

   SOLUTION = 4

   which represents superelement generation, UNA makes a binary file under the specified name. Binary file contains all necessary data for generated substructure (reduced stiffness matrix and reduced load vector). In the case when SUPNAME Data Entry does not exist, UNA creates file with superelement data under the default name:

   SUPNAME = "SUPELEM.BIN"

SYSTEM

Service printing.

Format:

SYSTEM	M1HH A2HH B2HH S2HH B3HH S3HH A4HH K4HH DIAG	=	ROW	Num1 Num2	COLUMN	Num1 Num2

Examples:

SYSTEM K4HH = ROW 1 6 COLUMN 3 6
SYSTEM DIAG = ROW 1 3

Describers	**Meaning**
M1HH	Modal mass matrix.
A2HH	Modal aero damping matrix.
B2HH	Modal viscous damping matrix, direct modal definition.
S2HH	Modal viscous damping matrix, sum of element viscous damping.
B3HH	Modal complex stiffness damping matrix, direct modal definition.
S3HH	Modal complex stiffness damping matrix, sum of element structural damping.
A4HH	Modal aero stiffness matrix.
K4HH	Modal structural stiffness matrix.
DIAG	Modal load and solution vectors.
ROW	Matrix row.
COLUMN	Matrix column.
Num1 - Num2	Rows or columns range to be printed.

Remarks:

1. This entry requests and controls output for the generalized modal system matrices during frequency response analysis, random vibration analysis and dynamic gust analysis (SOL = 6,8,9). Modal load and solution vectors are printed for each frequency step. The generalized system of equations is defined as

$$\left[-\Omega^2 M_{1HH} + i\cdot\Omega\cdot\left(\frac{Q}{V}A_{2HH} + B_{2HH} + S_{2HH}\right) + i\cdot\left(B_{3HH} + S_{3HH}\right) + Q.A_{4HH} + K_{4HH}\right]\cdot\bar{X}_H(\Omega) =$$

$$= \bar{P}_H(\Omega) + Q\cdot\bar{P}_{wH}(\Omega)$$

where

$\left[M_{1HH}\right] = [\phi]^T\left[\Sigma m_e\right][\phi]$: modal mass, all elements mass.

$\left[A_{2HH}\right] = [\phi]^T\left[\Sigma a_{2e}\right][\phi]$: modal aerodynamic damping, all aero elements.

$\left[B_{2HH}\right] = [\phi]^T\left[\Sigma b_{2e}\right][\phi]$: modal viscous damping, all elements.

$\left[S_{2HH}\right] = \left[\Sigma g_h\omega_h m_h\right]$: modal viscous damping, direct modal definition

$\left[B_{3HH}\right] = [\phi]^T\left[\Sigma b_{3e}\right][\phi]$: modal complex stiffness damping, all elements.

$\left[S_{3HH}\right] = \left[\Sigma g_h\omega_h^2 m_h\right]$: modal complex stiffness damping, direct modal definition.

$\left[A_{4HH}\right] = [\phi]^T\left[\Sigma a_{4e}\right][\phi]$: modal aerodynamic stiffness, all aero elements.

$\left[K_{4HH}\right] = [\phi]^T\left[\Sigma k_{4e}\right][\phi]$: modal structural stiffness, all structural elements.

$\left\{\bar{P}_H(\Omega)\right\} = [\phi]^T\cdot\left\{\bar{P}(\Omega)\right\}$: modal mechanical load vector.

$\left\{\bar{P}_{wH}(\Omega)\right\} = [\phi]^T\cdot\left\{\bar{P}_w(\Omega)\right\}$: modal gust load vector.

$\left\{\bar{X}_H(\Omega)\right\} = [\phi]^T\cdot\left\{\bar{X}(\Omega)\right\}$: modal solution vector.

Ω : Excitation frequency [rad/sec]
Q : Dynamic pressure
V : Velocity

For definition of S_{2HH} and S_{3HH} matrices see SDAMPING Executive Control entry.

TITLE

Defines a character string that will appear on the first heading line at each page of UNA output.

Format:

TITLE = Any Text 65 characters long

Examples:

TITLE = "Any text you want to print"
TITLE = "Test problem No. 3"

Remarks:

1. Text defined by this card will be printed on each page in the output file.

Requests output in I_DEAS "UNIVERSAL" file format.

Format:

UNIVERSAL	GEOM DISPL REACT GPFORCE VONMISS TSAIHILL STRESS STRAIN FORCE ENERGY MODES LAMTSAI LAYTSAI LAYSTRESS	=	ON OFF

Examples:

UNIV = ON
UNIV VONMIS = ON

Describers	**Meaning**
GEOM	Finite element model geometry and external loads.
DISP	Nodal displacements.
REACT	SPC reactions.
GPFORCE	Element grid point forces.
VONMIS	Element Von Misses stresses.
TSAIHILL	Element Tsai-Hill criterion values.
STRESS	Element nodal point stresses.
STRAIN	Element nodal point strains.
FORCE	Element internal forces and moments Nx, Ny, Nxy, Mx, My, Mxy.
ENERGY	Element specific deformation energy.
MODES	Eigenvectors (modal displacements).
LAMTSAI	Max. Tsai-Hill value found through element thickness.
LAYTSAI	Tsai-Hill value for each ply through element thickness.
LAYSTRESS	Stresses for each ply through element thickness.
ON	Create output.
OFF	Do not create output.

Remarks:

- This entry directs UNA to provide an additional files containing analysis output in the "UNIVERSAL" format. These files are intended to be used for postprocesing. This includes deformed, contour and criteria plots, SPC reactions visualization with the SDRC I_DEAS, ESP FEMAP or another postprocesing software capable to read and process UNIVERSAL type files.

- UNA creates the following files with analysis output in the UNIVERSAL file format:

File Name	**Output type**	**Output request**
UNA.UF1	Finite element model geometry, boundary conditions and external load	UNIV GEOM = ON
UNA.UF2	Analysis run output (except modes)	UNIV (DISP, REACT, ..., LAYSTRESS) = ON
UNA.UF3	Eigenvectors (modal displacements)	UNIV MODES = ON

- The following table lists recommended output to be used for postprocessing for each element type

Type	Material	Vmis	Tsai	Stress	Strain	Force	Lam-Tsai	Lay-Tsai	Lay-Stress
MEMB	Isotropic	+		*	*	*			
MEMB	Orthotropic	+		*	*	*			
MEMB	Multilayered	*			*	*	*	*	*
SOLID	Isotropic	*		*					
SOLID	Orthotropic	*		*					
SHEAR	Isotropic	*		*		*			
PLATE	Isotropic	+		*	*	*			
PLATE	Orthotropic	+		*	*	*			
PLATE	Multilayered	*			*	*	*	*	*

* - recommended
\+ - can be used, but not recommended

- Data set named VONMISS contains element grid points Von-Misses stresses. In-plane stresses are calculated on the following way

$$\cdot\sigma_X = \frac{N_X}{t} \qquad \cdot\sigma_Y = \frac{N_Y}{t} \qquad \tau_{XY} = \frac{N_{XY}}{t}$$

  so Von-Misses stress represents average value through element thickness. For MEMBRANE, SOLID and SHEAR panel calculated values are exact. For shell elements, with element membrane and bending stiffness, exact values on element top and bottom planes cannot be represented. In that case for isotropic and orthotropic element materials data set of STRESS type is recommended. IDEAS calculates and represent exact top, mid-plane and bottom plane Von-Misses values using STRESS data.

- Data set of STRESS type contains element grid point stresses. This data set is generally recommended for all element types with isotropic or orthotropic materials. IDEAS calculates min., max. and Von-Misses stresses from this data set.

- Data set of STRAIN type contains element grid point strains. It allows min and max strain visualization. It can be used to visualize of the through the thickness deformations and stresses of multilayered composite materials. IDEAS moduo LAMINA performs such visualization using data from this data set (multilayered material properties are sent to IDEAS via UNA.UF1 file).

- Data set of LAMTSAI type contains through the thickness max. Tsai-Hill values.

- Data set of LAYTSAI type contains Tsai-Hill values for each ply for all elements. It is possible to visualize each lamina values using this data set..

- Data set of LAYTSTRESS type contains each ply stresses for all elements. It enables visualization of each lamina max., min. and Von-Misses stresses.

VERSION

Specify input file version.

Format:

VERSION = | n |

Examples:

VERSION = 5.24

Describers	**Meaning**
n	UNA version standard that file comply with.

Remarks:

1. VERSION defines the UNA standard that input file is complying with. It allows program to take into the account changes in the input data format that take place between UNA versions. For example, in order to run UNA version 7.2 program with input file that is made for UNA version 5.24, specify in the input file

   VER = 5.24

   and run file with UNA72 program.

2. UNA will not run without this Executive Data Entry.

Selects aerodynamic correction factors.

Format:

WTFACT = | n |

Examples:

WTFACT = 1401

Describers	Meaning
n	Set identification of a WTFACT Bulk Data Entry.

Remarks:

1. A selected WTFACT set of aerodynamic correction factors is applied to aerodynamic elements.

Section 2

Bulk Data Section

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

$ Followed by any characters

Example:

```
$
$  Material al 7475-T7351 plate
$
```

Remarks:

1. Comments are ignored by the program.

AERO

Aerodynamic parameters.

Format:

1	2	3	4	5	6	7	8	9	10
AERO	ASID	RSID	REFC	REFB	REFS	Q	VELO	MACH	

Example:

AERO	99	88	234.	17500.	65000.	1.013E-3	13051.	0.381	

Field	Contents
ASID	Aerodynamic coordinate system identification number. (Integer > 0).
RSID	Reference coordinate system for rigid body motions and aerodynamic derivatives. (Integer > 0).
REFC	Reference chord length. (Real > 0.).
REFB	Reference span. (Real > 0.).
REFS	Reference wing area. ((Real > 0.).
Q	Dynamic pressure. (Real > 0.).
VELO	Aircraft forward velocity. (Real > 0.).
MACH	Mach number. (Real > 0.).

Remarks:

1. Only one AERO entry is allowed.
2. ASID is rectangular system. Flow is in positive x-direction.
3. RSID is rectangular system. All AESTAT degrees of freedom defining trim variables will be defined in this coordinate system. Aircraft aerodynamic derivatives (i.e. C_{L0}, $C_{L\alpha}$, C_{M0}, $C_{M\alpha}$ ..) that are based on REFS wing area will be calculated in this coordinate system as well.
4. Dynamic pressure Q is defined as follows

$$Q = \frac{1}{2}\rho V^2$$

Specifies rigid body motions to be used as a trim variables in static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AESTAT	SID	ANGLEA	SIDES						

Example:

AESTAT	101	0.052	0.0						

Field	**Contents**
SID	Load set identification number. (Integer > 0).
ANGLEA	Angle of attack in radians. (Real).
SIDES	Angle of sideslip in radians. (Real).

Remarks:

1. The degrees of freedom defined with this entry represents rigid body motion in the reference coordinate system RSID defined on the AERO entry. The ANGLEA is angle of attack defined as aircraft rotation around RSID y-axis, and SIDES is sideslip angle defined as aircraft rotation around RSID z-axis.

2. The degrees of freedom defined with this entry are variables that produce aerodynamic loads in the static solution run (SOL=1).

3. The aerodynamic load defined with this entry is considered to be of element static load type, so scale factors that are defined with LOADQ data entry will be applied to it.

AVECTOR

Defines acceleration vectors for gravity or other acceleration loading.

Format:

1	2	3	4	5	6	7	8	9	10
AVECTOR	SID	CID	G	N1	N2	N3			

Example:

AVECTOR	2	3	9.81	-1.0	0.0	1.0			

Field	Contents
SID	Load set identification number. (Integer > 0).
CID	Coordinate system identification number. (Integer ≥ 0; Default=0).
G	Acceleration vector scale factor. (Real).
N1, N2, N3	Acceleration vector components measured in coordinate system CID. (Real).

Remarks:

1. UNA defines structural inertial load upon the following expression:

   $$\vec{Fin} = M\,\vec{A} = M\,G\,(N1\,\vec{i} + N2\,\vec{j} + N3\,\vec{k})$$

   where are:

   $\vec{Fin}$ - inertial load, acting in direction of acceleration vector

   M - structural mass

   $\vec{A}$ - acceleration vector

2. A CID of zero references the BASIC coordinate system.

CBAR

Defines a beam element.

Format:

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

Example:

CBAR	2	39	7	3	0.6	18.	26.		
		513							

Alternate Format and Example:

CBAR	EID	PID	GA	GB	G0			QID	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

CBAR	2	39	7	6	105				
		513							

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PBAR entry. (Integer > 0).
GA, GB	Grid point identification numbers of connection points. (Integer > 0, GA ≠ GB).
X1, X2, X3	Components of orientation vector V, from GA, in the displacement coordinate system at GA. (Real).
G0	Alternate method to supply the orientation vector V using grid point G0. Direction of V is from GA to G0. (Integer > 0).
QID	QLOAD1 or QLOAD2 load data entry pointer. (Integer).

PA, PB	Pin flags for bar ends A and B, respectively. 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 (see Figure 1). The bar must have stiffness associated with the PA and PB degrees of freedom to be released by pin flags. For example, if PA=4 is specified, the PBAR entry must have a value for J, the torsion stiffness. Up to 5 unique integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0).
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors Wa and Wb, respectively (see Figure 1) in displacement coordinate systems at points GA and GB, respectively. (Real or blank)

Remarks:

1. Element identification numbers must be unique with respect to all other CBAR elements.
2. Grid point G0 must not lay on element local X-axis.
3. The following figures define bar element geometry

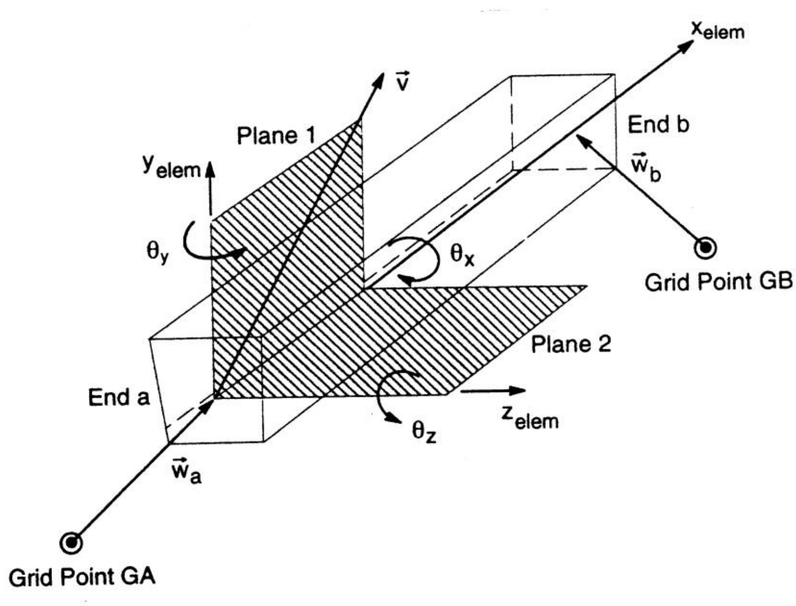


Figure 1. CBAR Element Geometry

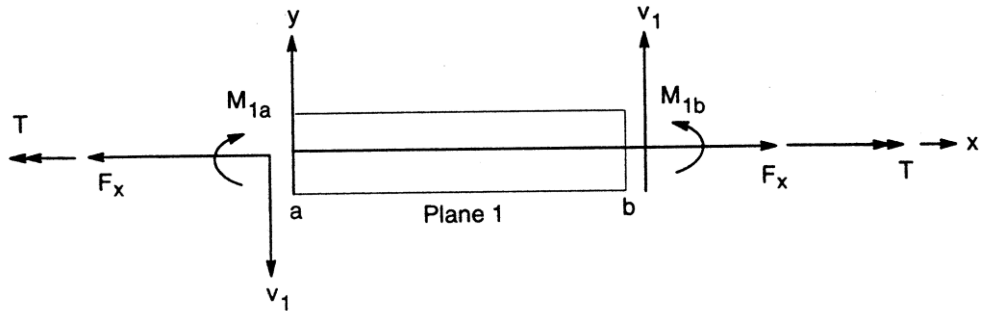


Figure 2. CBAR Element Internal Forces and Moments (x-y Plane)

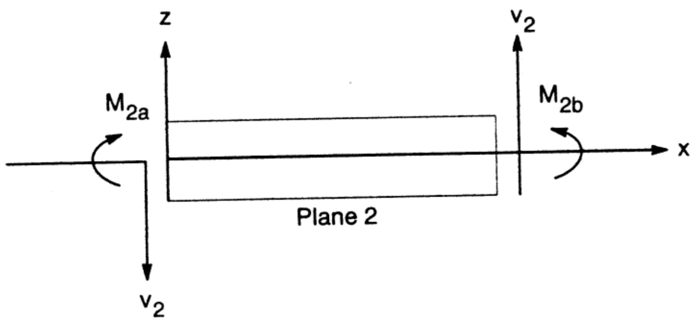


Figure 3. CBAR Element Internal Forces and Moments (x-z Plane)

4. QID identification links element with QLOAD1 or QLOAD2 Bulk Data Entries for applying the BAR element load.

5. The continuation may be omitted if there are no pin flags or offset.

6. CBAR element output is controlled by PARAM BAR Executive Data Entry.

Defines a simple beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	G0			QID	
	PA	PB							

Example:

CBEAM	2	39	16	17	9			3	
	345								

Alternate Format and Example:

CBEAM	EID	PID	GA	GB	X	Y	Z	QID	
	PA	PB							

CBEAM	2	39	16	17	0.5	20.	25.	3	
	345								

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PBEAM entry. (Integer > 0).
GA, GB	Grid point identification numbers of connection points. (Integer > 0, GA ≠ GB).
G0	Direction of orientation vector V is from GA to G0. (Integer > 0).
X, Y, Z	Coordinates of the third point for defining X-Y plane. (Reals).
QID	QLOAD1 or QLOAD2 load data entry pointer. (Integer).
PA, PB	Pin flags for bar ends A and B, respectively. 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 (see Figure 1). The beam must have stiffness associated with the PA and PB degrees of freedom to be released by pin flags. For example, if PA=4 is specified, the PBEAM entry must have a value for J, the torsion stiffness. Up to 5 unique integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0).

Remarks:

1. Element identification numbers must be unique with respect to all other CBEAM elements.

2. Grid point G0 must not lay on element local X-axis.

3. If third grid point identification is equal to zero (G0 = 0), or appropriate field is blank, program uses X, Y and Z coordinates to specify a third point that defines element local X-Y plane. X, Y and Z are given in BASIC coordinate system.

4. QID identification links element with QLOAD1 or QLOAD2 Bulk Data Entry for applying the BEAM element load.

5. The continuation may be omitted if there are no pin flags.

6. The following figure defines beam element geometry :

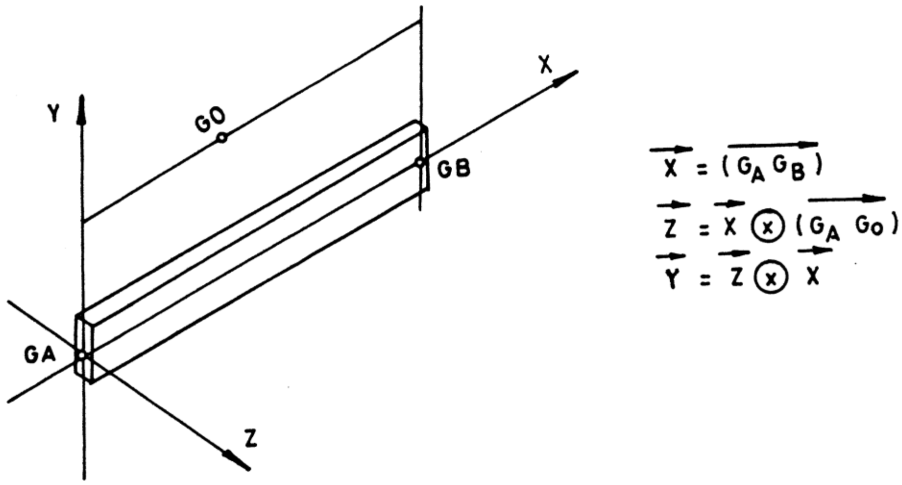


Figure 1. CBEAM Element Geometry

CDAMP2

Defines a scalar damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example:

CDAMP2	28	15.5	32	1	19	1			

Field	Contents
EID	Unique element identification number. (Integer > 0).
B	Value of scalar damper. (Real).
G1, G2	Grid point identification numbers of connection points. (Integer ≥ 0).
C1, C2	Component numbers. ($0 \leq$ Integer ≤ 6) .

Remarks:

1. Element identification numbers must be unique with respect to all other CDAMP2 elements.

2. 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 point whose displacement is constrained to zero.

3. The two connection points (G1,C1) and (G2,C2) must be distinct.

Defines a scalar spring element.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE		

Example:

CELAS2	28	3.1E+3	32	3	19	3			

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
K	Stiffness of the spring. (Real).
G1, G2	Grid point identification numbers of connection points. (Integer ≥ 0).
C1, C2	Component numbers. (0 ≤ Integer ≤ 6) .
GE	Structural element damping coefficient GE=2∗C/Co. (Real or blank).

Remarks:

1. Element identification numbers must be unique with respect to all other CELAS2 elements.
2. 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 point whose displacement is constrained to zero.
3. The two connection points (G1,C1) and (G2,C2) must be distinct.

CMASS

Defines a concentrated mass at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example:

CMASS	2	15	6	49.7					
	16.2		16.2			7.8			

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
G	Grid point identification number. (Integer > 0).
CID	Coordinate system identification number. (Integer ≥ 0; Default is 0). For CID = -1 see X1, X2, X3 below.
M	Mass value. (Real).
X1, X2, X3	Offset distances from the grid point to the centre of gravity of the mass in the coordinate system defined in filed 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the centre of gravity of the mass in the basic coordinate system. (Real).
Iij	Mass moments of inertia measured at mass centre of gravity in the coordinate system defined by filed 4. If CID = -1, the basic coordinate system is implied. (Real ≥ 0.0).

Remarks:

1. Element identification numbers must be unique with respect to all other CMASS elements.

2. Continuation card can be avoided if I22 - I33 are equal to zero.

3. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a non-basic coordinate system, but Iij must be defined in basic coordinate system.

4. The form of the inertia matrix about its centre of gravity is taken as:

M					
	M				
		M			
			I11	-I21	-I31
			-I21	I22	-I32
			-I31	-I32	I33

where

$$M = \int \rho \, dv$$

$$I11 = \int \rho (x_2^2 + x_3^2) \, dv$$

$$I22 = \int \rho (x_1^2 + x_3^2) \, dv$$

$$I33 = \int \rho (x_1^2 + x_2^2) \, dv$$

$$I21 = \int \rho \; x_1 x_2 \, dv$$

$$I31 = \int \rho \; x_1 x_3 \, dv$$

$$I32 = \int \rho \; x_2 x_3 \, dv$$

and X_1, X_2, X_3 are components of distance from the centre of gravity in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied automatically.

Defines an isoparametric membrane element from three to eight nodes.

Format:

1	2	3	4	5	6	7	8	9	10
CMEMB	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T	QID			MCID		

Example:

CMEMB	111	203	31	74	75	32		12	
			2.1				2		

Field	Contents
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of PSHELL or LAYERS entry. (Integer > 0).
G1, G2, ..., G8	Grid point identification numbers of connection points. (Integers ≥ 0, all unique).
T	Element thickness. (Real ≥ 0.0).
QID	QPRESS or DPRESS data entry identification number. Defines normal pressure on element surface. (Integer ≥ 0).
MCID	= Blank; Material orientation angle THETA=0.0 is assumed. = Integer; Material coordinate system MCID identification number. = Real; Material orientation angle THETA value in degrees.

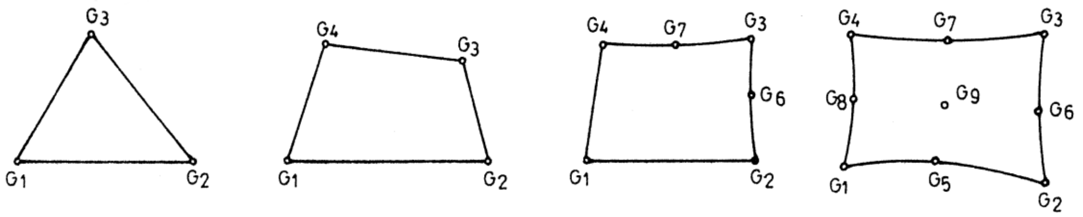


Figure 1. CMEMB elements geometry

Remarks:

1. Element identification numbers must be unique with respect to all other CMEMB elements.

2. Only existing nodes have to be specified. Nodal numbering convention is enclosed on Figure 1. Continuation card may be avoided if element has less than seven nodes.

3. Element local x-y system is defined on the following way: Coordinate origin is at element C.G. In the case of triangular element X-axis is parallel to edge G_1G_2. In the case of quadrilateral element X-axis passes through origin and middle of the edge G_2G_3. Y-axis is normal to X-axis and oriented to form right hand coordinate system.

4. To control the output see PARAM MEMB Executive Control Entry.

5. If thickness field is left is blank, T will be set equal to value T on the PSHELL entry.

6. If a coordinate system is used to define material orientation, then 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element. If MCID is real number, then material property angle θ is given in the MCID field. Angle θ is measured from edge G_1G_2 to material 0-axis.

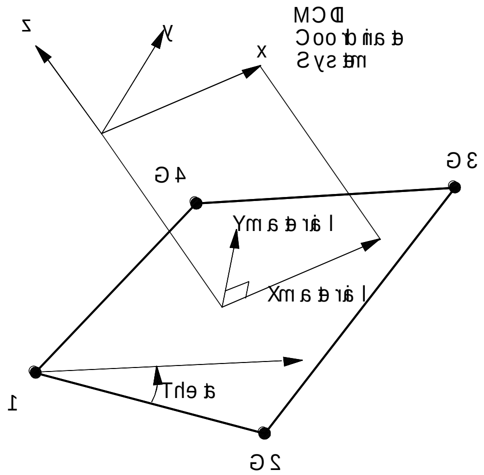


Figure 2. Material Coordinate System Definition.

CORD1C

Defines a cylindrical coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1C	3	16	32	19					

Field	Contents
CID	Coordinate system identification number. (Integer > 0).
GiA, GiB	Grid point identification numbers. (Integer > 0 ; G1A ≠ G2A ≠ G3A; G1B ≠ G2B ≠ G3B).

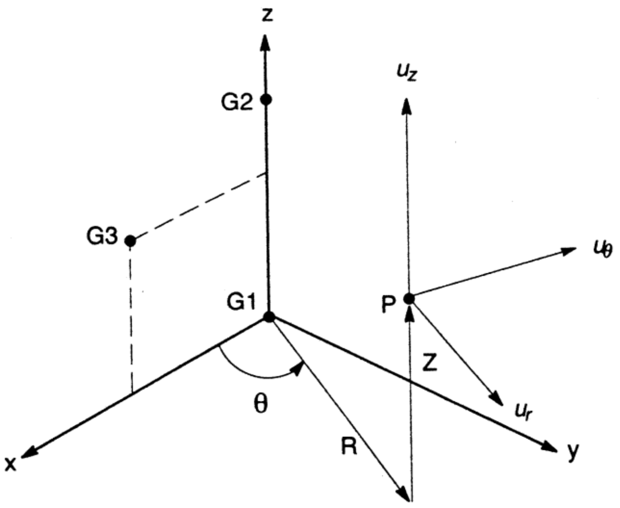


Figure 1. CORD1C Definition

Remarks:

1. Coordinate system identification number CID must be unique with respect to all other coordinate systems of any type (CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S).

2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate system 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. The three grid points GiA (or GiB) must be noncollinear and not coincident.

4. The location of a grid point (P in Figure 1) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.

5. The displacement coordinate directions at P are dependent on the location of P as shown in Figure 1 (U_r , U_θ , U_z).

6. For points on Z axis $U_r = U_x$ and $U_\theta = U_y$.

CORD1R

Defines a rectangular coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1R	3	16	32	19					

Field	Contents
CID	Coordinate system identification number. (Integer > 0).
GiA, GiB	Grid point identification numbers. (Integer > 0 ; G1A ≠ G2A ≠ G3A; G1B ≠ G2B ≠ G3B).

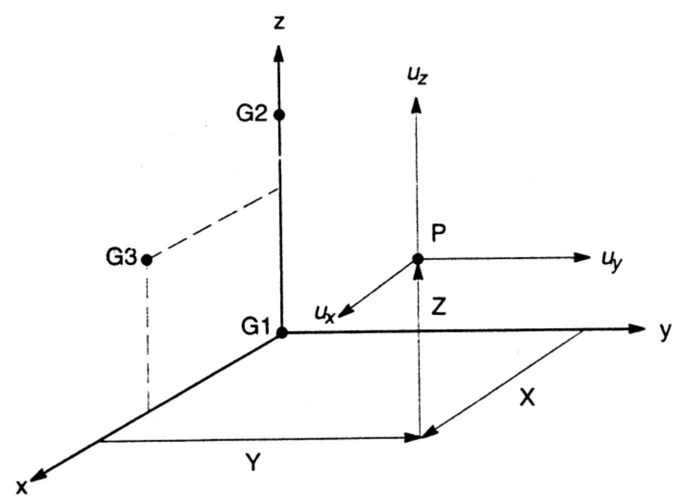


Figure 1. CORD1R Definition

Remarks:

1. Coordinate system identification number CID must be unique with respect to all other coordinate systems of any type (CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S).

2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate system 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 x-z plane. The three grid points GiA (or GiB) must be noncollinear and not coincident.

4. The location of a grid point (P in Figure 1) in this coordinate system is given by (X, Y, Z).

5. The displacement coordinate directions at P are shown by (U_x , U_y , U_z).

Defines a spherical coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1S	3	16	32	19					

Field	**Contents**
CID	Coordinate system identification number. (Integer > 0).
GiA, GiB	Grid point identification numbers. (Integer > 0 ; G1A ≠ G2A ≠ G3A; G1B ≠ G2B ≠ G3B).

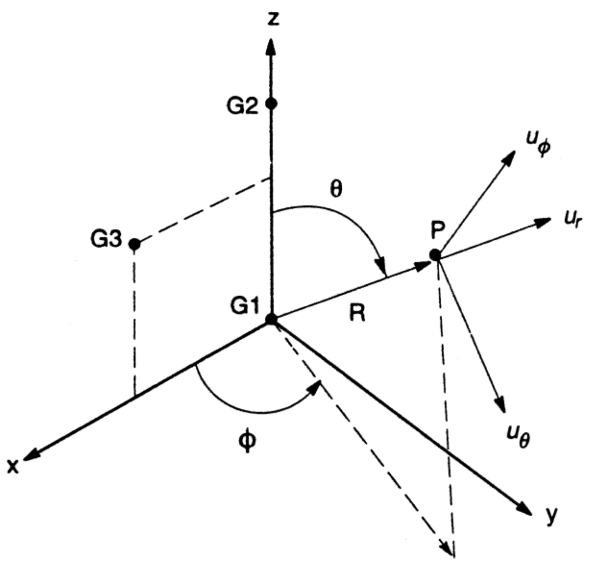


Figure 1. CORD1S Definition

Remarks:

1. Coordinate system identification number CID must be unique with respect to all other coordinate systems of any type (CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S).

2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate system 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. The three grid points GiA (or GiB) must be noncollinear and not coincident.

4. The location of a grid point (P in Figure 1) in this coordinate system is given by (R, θ, ϕ) where θ and ϕ are measured in degrees.

5. The displacement coordinate directions at P are dependent on the location of P as shown in Figure 1 (U_r , U_θ , U_ϕ).

6. For points on Z axis $U_r = U_z$, $U_\theta = U_x$ and $U_\phi = U_y$.

CORD2C

Defines a cylindrical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	**Contents**
CID	Coordinate system identification number. (Integer > 0).
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default (zero) is basic coordinate system.
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real).

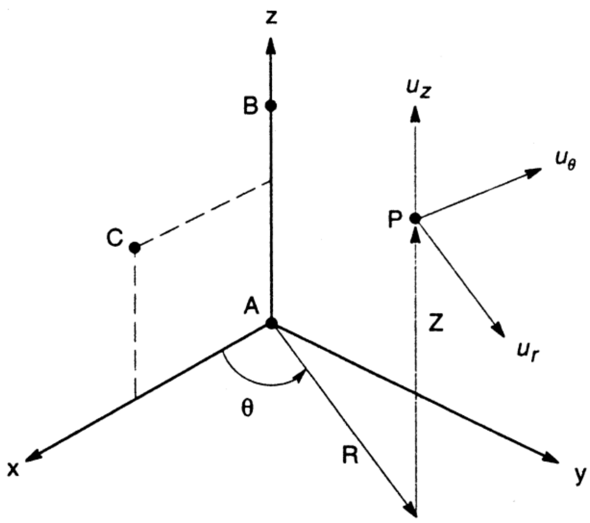


Figure 1. CORD2C Definition

Remarks:

1. Coordinate system identification number CID must be unique with respect to all other coordinate systems of any type (CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S).

2. The three points (A1,A2,A3), (B1,B2,B3), (C1,C2,C3) must be unique and noncollinear. 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.

3. The continuation entry is not required if C1 = C2 = C3 = 0.

4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in Figure 1) in this coordinate system 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 in Figure 1 (U_r , U_θ , U_z).

7. For points on Z axis $U_r = U_x$ and $U_\theta = U_y$.

CORD2R

Defines a rectangular coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number. (Integer > 0).
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default (zero) is basic coordinate system.
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real).

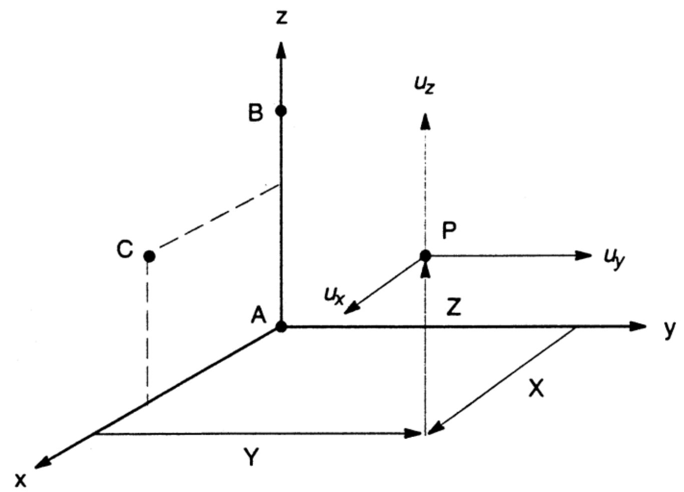


Figure 1. CORD2R Definition

Remarks:

1. Coordinate system identification number CID must be unique with respect to all other coordinate systems of any type (CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S).

2. The three points (A1,A2,A3), (B1,B2,B3), (C1,C2,C3) must be unique and noncollinear. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the x-z plane. The reference coordinate system must be independently defined.

3. The continuation entry is not required if C1 = C2 = C3 = 0.

4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in Figure 1) 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).

Defines a spherical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number. (Integer > 0).
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default (zero) is basic coordinate system.
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real).

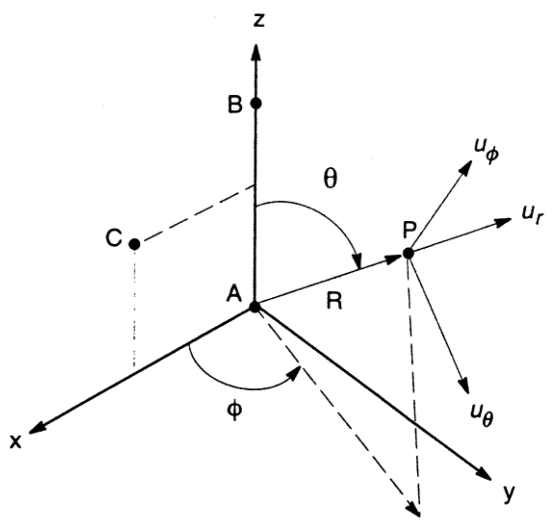


Figure 1. CORD2S Definition

Remarks:

1. Coordinate system identification number CID must be unique with respect to all other coordinate systems of any type (CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, CORD2S).

2. The three points (A1,A2,A3), (B1,B2,B3), (C1,C2,C3) must be unique and noncollinear. 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.

3. The continuation entry is not required if C1 = C2 = C3 = 0.

4. If RID is zero or blank, the basic coordinate system is used.

5. The location of a grid point (P in Figure 1) 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 dependent on the location of P as shown in Figure 1 (U_r , U_θ , U_ϕ).

7. For points on Z axis $U_r = U_z$, $U_\theta = U_x$ and $U_\phi = U_y$.

Defines a set of coupled degrees of freedom between two grids.

Format:

1	2	3	4	5	6	7	8	9	10
COUPG	SID	GM	GS	C					

Example:

COUPG	100	13	24	3456					

Field	**Contents**
SID	Identification number of grid coupling set. (Integer > 0).
GM	Master (independent) grid identification number. (Integer > 0).
GS	Slave (dependent) grid identification number. (Integer > 0).
C	Component numbers. (Any unique combination of integers 1 through 6 with no embedded blanks).

Remarks:

1. Coupling set must be selected with the Executive Control Entry MPC = SID.

2. Continuation entry is not allowed.

3. Component numbers C specify degrees of freedom which are coupled between master and slave grid points. The deflections of the coupled DOF's will be the same inside each load case.

4. The following are conditions which COUPG has to satisfy:
   - No SPC or SPC1 constraints are allowed at dependent degrees of freedom.
   - No dependent degrees of freedom are allowed at superelement boundary (SOL=4).
   - Dependent degrees of freedom cannot be dependent or independent in another MPC entry, COUPG entry or RBE2 rigid element.

Defines an isoparametric membrane-bending quadrilateral element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	MCID	ZOFFS	
	QID		T						

Example:

CQUAD4	111	203	31	74	75	22	+45.	0.5	

Field	Contents
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PSHELL or LAYERS entry. (Integer > 0).
Gi	Grid point identification numbers of connection points. (Integers > 0).
MCID	= Blank; Material orientation angle THETA=0.0 is assumed. = Integer; Material coordinate system MCID identification number. = Real; Material orientation angle THETA value in degrees.
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real).
QID	QPRESS or DPRESS data entry identification number. (Integer ≥ 0).
T	Element thickness. (Real ≥ 0.0).

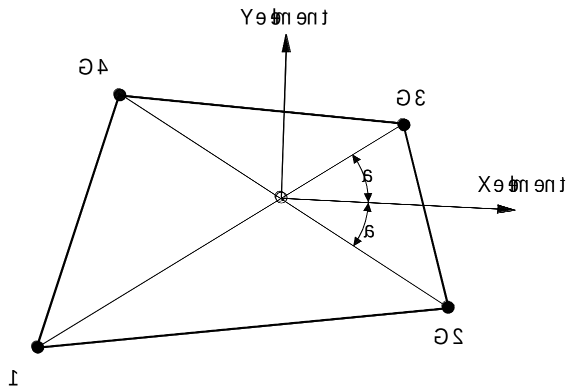


Figure 1. CQUAD4 Element Connection and Coordinate System.

Remarks:

1. Element membrane part is an isoparametric quadrilateral with reduced integration for shear. Bending part is based on Mindlin-Reissner theory that includes transverse shear flexibility.

2. Element identification numbers must be unique with respect to all other CQUAD4 elements.

3. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element. Element local x-y system is defined on Figure 1. Coordinate origin is at element C.G. X-axis represents a vector defined by intersection of diagonals, and point that splits angle between diagonals on two equal parts. Y-axis is normal to X-axis and oriented to form right hand coordinate system.

4. Element may be offset from the connection points by means of the ZOFFS field. Element reference plane is defined as a middle plane in the case of isotropic material, or as specified at LAYERS Bulk Data Entry in the case of composite material. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system.
   The use of ZOFFS will cause incorrect results in buckling analysis. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

5. If a coordinate system is used to define material orientation, then 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element. If MCID is real number, then material property angle θ is given in the MCID field. Angle θ is measured from edge G_1G_2 to material 0-axis.

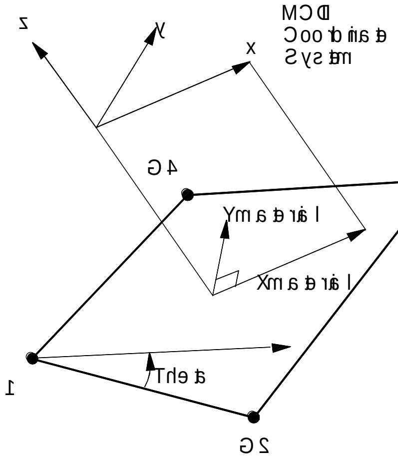


Figure 2. Material Coordinate System Definition.

Defines a tension-compression-torsion element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2	MID	A	J	C	
	NSM								

Example:

CROD	2	13	16	17					

Field	Contents
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PROD entry. (Integer ≥ 0).
G1, G2	Grid point identification numbers of connection points. (Integer > 0, G1 ≠ G2)
MID	Material identification number.(Integer ≥ 0).
A	Area of the rod. (Real).
J	Torsional constant. (Real).
C	Coefficient for torsional stress determination. (Real).
NSM	Non-structural mass per unit length. (Real).

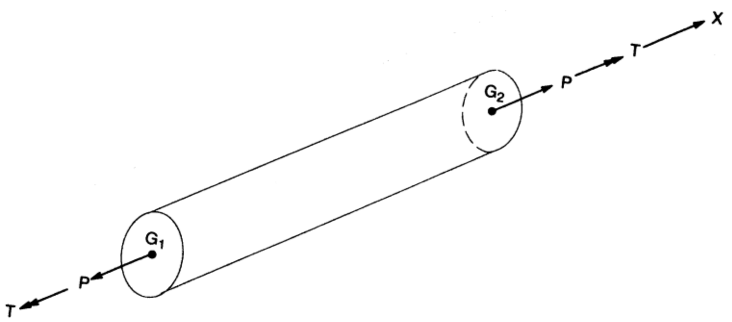


Figure 1. CROD Element Internal Forces and Moments

Remarks:

1. Element identification numbers must be unique with respect to all other CROD elements.

2. Values for blank (or with zero values) element data (MID, A, J, C, NSM) are taken from PROD data entry.

3. If PID = 0 material identification number must be defined.

4. MID identification number may only reference MAT1 material entries.

5. Coefficient C is used to calculate torsional shear stress by the following expression :

$$.\tau = \frac{CM_\theta}{J}$$

where M_θ represents torque, and J torsional constant.

CSHEAR

Defines a shear panel element.

Format:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4	T	MID	

Example:

CSHEAR	3	6	1	5	3	7			

Field	Contents
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PSHEAR or PSHELL entry. (Integer ≥ 0).
Gi	Grid point identification numbers of connection points. (Integer > 0; $G1 \neq G2 \neq G3 \neq G4$).
T	Element thickness. (Real).
MID	Material identification number. (Integer ≥ 0).

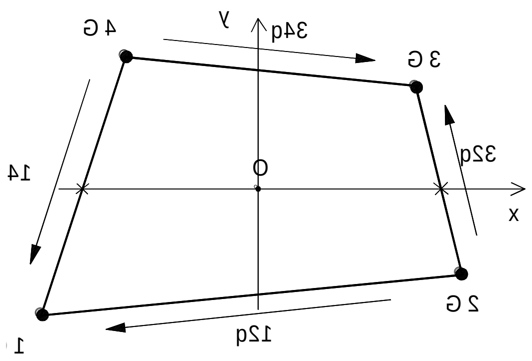


Figure 1. CSHEAR Element Connection and Coordinate System.

Remarks:

1. Element identification numbers must be unique with respect to all other CSHEAR elements.

2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.

3. Property identification number PID can be avoided if material and thickness are defined.

4. Values for blank element data (MID, T) are taken from PSHEAR data entry, and if PSHEAR with PID number does not exist, from PSHELL data entry.

5. All interior angles must be less than 180°.

6. Temperature loads are not generated for shear elements.

7. Element local x-y system is defined on Figure 1. Coordinate origin is at element C.G. X-axis passes through origin and middle of the edge G_2G_3. Y-axis is normal to x-axis and oriented to form right hand coordinate system.

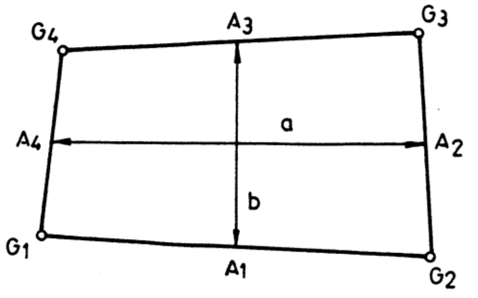
8. The convention defining positive shear flows is shown on Figure 1.

9. Rod elements are generated on the shear panel edges by Executive Data Entry :

   PARAM SHEAR 1 = [Num]
   PARAM SHEAR 2 = [Num]

   The first above stated entry requests automatic rod generation on edges G_1G_2 , G_3G_4 and second on edges G_2G_3 , G_4G_1 . The effective cross section area is defined in a such way that equivalent moment of inertia is equal to element moment of inertia for in-plane bending.

10. Generation of rod elements on element edges can be defined by PSHEAR data entry as well.



$$I_1 = \frac{t b^3}{12}$$

$$I_2 = \frac{t b^3}{12}$$

$$A_1 = A_3 = \frac{tb}{6}$$

$$A_2 = A_4 = \frac{ta}{6}$$

t = panel thickness

$A_1 = A_3$ = rod area

$A_2 = A_4$ = rod area

CSHELL3

Defines an isoparametric membrane-bending DKT triangular element.

Format:

1	2	3	4	5	6	7	8	9	10
CSHELL3	EID	PID	G1	G2	G3		MCID	ZOFFS	
	QID		T						

Example:

CSHELL3	111	203	31	74	75		+45.		

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PSHELL or LAYERS entry. (Integer > 0).
Gi	Grid point identification numbers of connection points. (Integers > 0).
MCID	= Blank; Material orientation angle THETA=0.0 is assumed. = Integer; Material coordinate system MCID identification number. = Real; Material orientation angle THETA value in degrees.
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real).
QID	QPRESS or DPRESS data entry identification number. (Integer ≥ 0).
T	Element thickness. (Real ≥ 0.0).

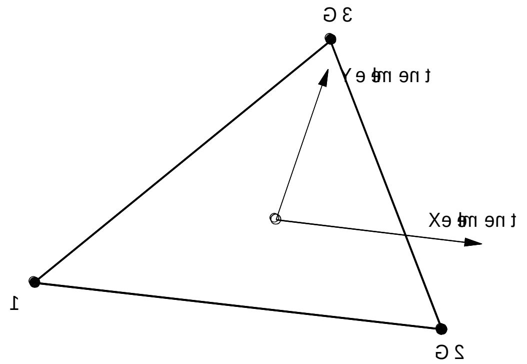


Figure 1. CSHELL3 Element Connection and Coordinate System.

Remarks:

1. Element membrane part is constant strain triangle (CST). Bending part is based on Discrete Kirchhoff Theory (DKT) that has infinite transverse shear stiffness.

2. Element identification numbers must be unique with respect to all other CSHELL3 elements.

3. Grid points G1 through G3 must be ordered consecutively around the perimeter of the element. Element local x-y system is defined on Figure 1. Coordinate origin is at element C.G. X-axis is parallel to edge G_1G_2. Y-axis is normal to X-axis and oriented to form right hand coordinate system.

4. Element may be offset from the connection points by means of the ZOFFS field. Element reference plane is defined as a middle plane in the case of isotropic material, or as specified at LAYERS Bulk Data Entry in the case of composite material. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system.

   The use of ZOFFS will cause incorrect results in buckling analysis. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

5. If a coordinate system is used to define material orientation, then 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element. If MCID is real number, then material property angle θ is given in the MCID field. Angle θ is measured from edge G_1G_2 to material 0-axis.

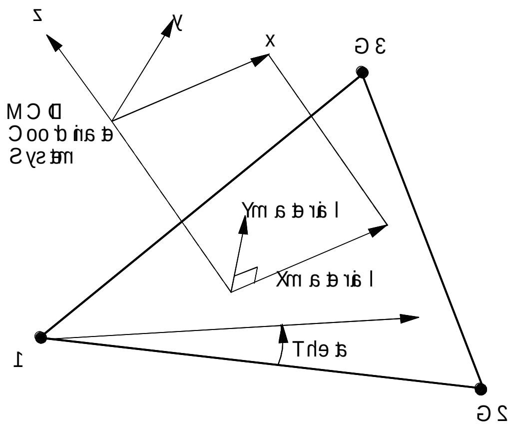


Figure 2. Material Coordinate System Definition.

CSHELL4

Defines an isoparametric membrane-bending DKT quadrilateral element.

Format:

1	2	3	4	5	6	7	8	9	10
CSHELL4	EID	PID	G1	G2	G3	G4	MCID	ZOFFS	
	QID		T						

Example:

CSHELL4	111	203	31	74	75	22	+45.	0.5	

Field	Contents
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PSHELL or LAYERS entry. (Integer > 0).
Gi	Grid point identification numbers of connection points. (Integers > 0).
MCID	= Blank; Material orientation angle THETA=0.0 is assumed. = Integer; Material coordinate system MCID identification number. = Real; Material orientation angle THETA value in degrees.
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real).
QID	QPRESS or DPRESS data entry identification number. (Integer ≥ 0).
T	Element thickness. (Real ≥ 0.0).

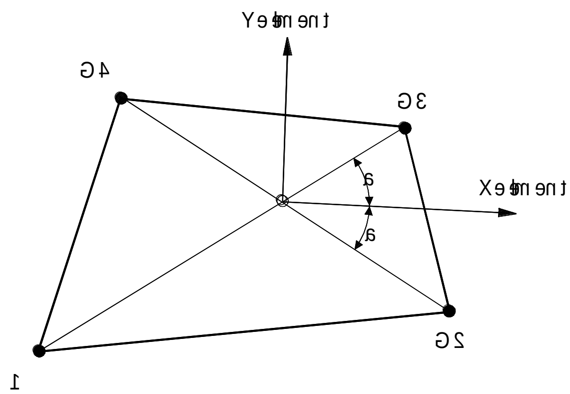


Figure 1. CSHELL4 Element Connection and Coordinate System.

Remarks:

1. Element membrane part is an isoparametric quadrilateral with additional “bubble” displacement functions. Bending part is based on Discrete Kirchhoff Theory (DKT) that has infinite transverse shear stiffness.

2. Element identification numbers must be unique with respect to all other CSHELL4 elements.

3. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element. Element local x-y system is defined on Figure 1. Coordinate origin is at element C.G. X-axis represents a vector defined by intersection of diagonals, and point that splits angle between diagonals on two equal parts. Y-axis is normal to X-axis and oriented to form right hand coordinate system.

4. Element may be offset from the connection points by means of the ZOFFS field. Element reference plane is defined as a middle plane in the case of isotropic material, or as specified at LAYERS Bulk Data Entry in the case of composite material. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system.

   The use of ZOFFS will cause incorrect results in buckling analysis. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

5. If a coordinate system is used to define material orientation, then 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element. If MCID is real number, then material property angle θ is given in the MCID field. Angle θ is measured from edge G_1G_2 to material 0-axis.

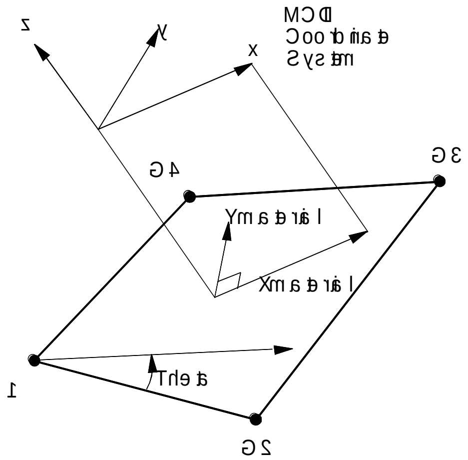


Figure 2. Material Coordinate System Definition.

CSOLID

Defines a tetra, penta or hexa solid element.

Format:

1	2	3	4	5	6	7	8	9	10
CSOLID	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

Example:

CSOLID	111	203	31	74	75	76	12	21	
	25	29							

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PSOLID entry. (Integer > 0).
Gi	Grid point identification numbers of connection points. (Integer ≥ 0 or blank).

Remarks:

1. Element identification numbers must be unique with respect to all other CSOLID elements.

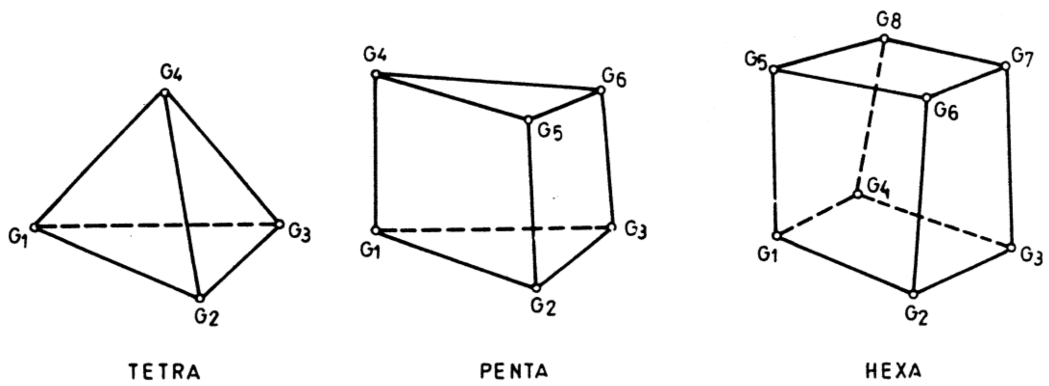


Figure 1. CSOLID Elements Connection

2. Element type is defined with a number of specified grid points. Figure 1 shows for TETRA, PENTA and HEXA elements. In the case of first two element types continuation entry (+) may be avoided.

3. Stress components are output in Basic or Material coordinate systems, depending on the element material (isotropic or orthotropic). Output system is controlled via PARAM SOLID Executive Control Entry.

4. In the case of orthotropic material with some of modulus set to zero, i.e. for modelling sandwich core material, it is necessary to turn-of incompatible modes, see PARAM SOLID Executive Control Entry.

5. Element local x-y-z system is defined on Figure 2. Coordinate origin is at element C.G.

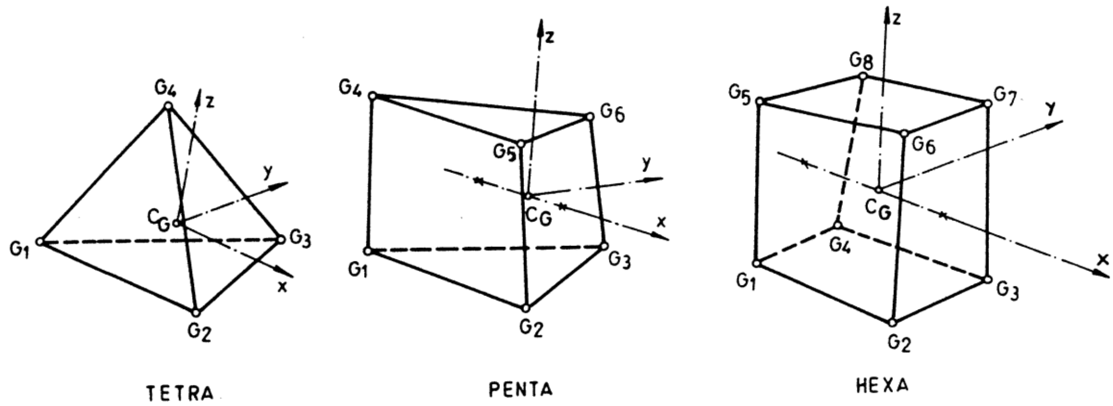


Figure 2. CSOLID Elements Coordinate systems

TETRA	X-axis is parallel to edge G_1G_2 . Y-axis is normal to X and lay in plane parallel to those defined by grids $G_1G_2G_3$. Z-axis is normal to X-Y plane.
PENTA	X-axis joints the centroids of the faces $G_1G_3G_6G_4$ and $G_2G_3G_6G_5$. X-Y plane is defined with X-axis and line which joints centroids of the face $G_1G_2G_5G_4$ and edge G_3G_6 . Z-axis is normal to X-Y plane.
HEXA	X-axis joints the centroids of the faces $G_1G_4G_8G_5$ and $G_2G_3G_7G_6$. X-Y plane is defined with X-axis and line which joints centroids of the faces $G_1G_2G_6G_5$ and $G_4G_3G_7G_8$. Z-axis is normal to X-Y plane.

CSTRIP

Defines an aerodynamic strip element.

Format:

1	2	3	4	5	6	7	8	9	10
CSTRIP	EID	PID	G1	G2	G3	G4	NODE	CK	

Example:

CSTRIP	94105	4105	9101	9102	9104	9013			

Field	**Contents**
EID	Element identification number. (Integer > 0).
PID	Property identification number of a PSTRIP entry. (Integer ≥ 0).
Gi	Grid point identification numbers of connection GRIDA points. (Integer > 0; G1 ≠ G2 ≠ G3 ≠ G4).
NODE	Structural grid point identification number. (Integer ≥ 0).
CK	Theodorsen nonstationary correction = 0 Defined via PARAM STRIP Executive Data Entry. (Default). = 1 *C(k)*=1.0 (constant). = 2 $C(k)=C(\omega c/2U)$ function of reduced frequency.

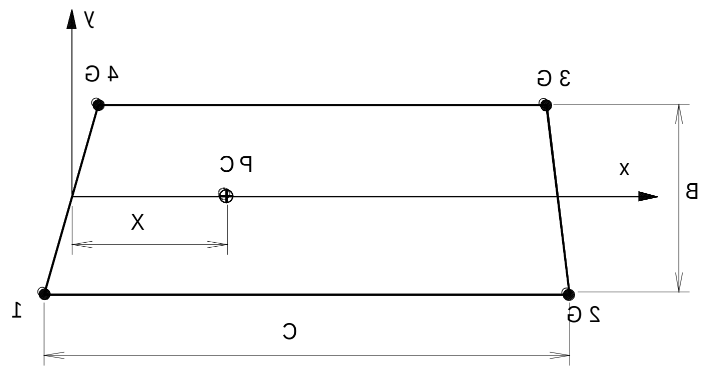


Figure 1. CSTRIP Element Connection and Coordinate System.

Remarks:

1. Element identification numbers must be unique with respect to all other CSTRIP elements.

2. Grid points G1 through G4 must be of the GRIDA type.

3. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.

4. Element local x-y system is defined on Figure 1. Coordinate origin is at middle of the element leading edge G_1G_4. X-axis passes through middle of the edges G_1G_4 and G_2G_3. Y-axis is normal to X-axis and oriented to form right hand coordinate system.

5. Element local X-axis must be co-linear with ASID aerodynamic system X-axis as specified at AERO Bulk Data Entry.

6. Element local cord C and width B are shown on Figure 1. Cord is defined by line connecting middle of the edges G_1G_4 and G_2G_3. Aerodynamic reference area is defined as S_{AREA}=CB.

7. If NODE structural grid is specified, than CSTRIP is directly attached to structure at that grid point via internally generated rigid link. Any SPLINE defined aero - structure linking is ignored in this case.

8. In the nonstationary regime (SOL=9) additional correction *C(k)* will be applied to strip aerodynamic properties. It is based on Teodorsen function and called *lift deficiency factor*

   $$C(k) = \frac{H_1^2(k)}{H_1^2(k)+iH_0^2(k)} \qquad \text{where} \qquad k = \frac{\omega c}{U} \quad \text{is reduced frequency.}$$

   In the case when a constant factor *C(k)*=1.0 is specified for all reduced frequencies *k*, a "quasi-steady" aerodynamic will be used for the panel, see PARAM STRIP Executive Control Entry.

Defines a Superelement.

Format:

1	2	3	4	5	6	7	8	9	10
CSUPEL	EID	CID	GE		"SUPERELEMENT NAME"				
	GS1	GM1	GS2	GM2	GS3	GM3	GS4	GM4	
	GS5	GM5	GS6	GM6	-etc-				

Example:

CSUPEL	110	7			WING.BIN				
	17	1	18	2	91	3	56	9	
	45	4							

Field	Contents
EID	Unique element identification number. (Integer > 0).
CID	Coordinate system identification number. This system is coincident with Superelement Basic coordinate system . (Integer ≥ 0 or blank).
GE	Structural element damping coefficient GE=2∗C/Co. (Real or blank).
SUPERELEMENT NAME	External file name. This file contains all Superelement data (up to 32 characters, fields 5 - 8).
GSi	Grid point identification numbers of Superelement connection points at Superelement to main structure interface. (Integer ≥ 0 or blank).
GMi	Grid point identification numbers of main structure connection points at Superelement to main structure interface. (Integer ≥ 0 or blank).

Remarks:

1. Element identification numbers must be unique with respect to all other CSUPEL elements.

2. Coordinate system CID is coincident with Superelement Basic coordinate system. This provides that Superelement and main structure can be modelled in different coordinate systems. In the case when Superelement and main structure Basic systems are coincident CID = 0 or blank.

3. Superelement name defined in the fields 4-6 represents external binary file with all Superelement data (stiffness, load, grid points etc.). UNA creates that file during Superelement generation procedure (SOL = 4). If the file is not found program will send a message

   ERROR : Unregular Bulk Data Entry at Line : [Num]

4. GSi and GMi are representing grid point numbers on the Superelement and main structure side of the interface, respectively. Grid points has to be coincident, but not necessary with the same number. This approach provides that grid numbers can be defined independently.

5. It is required to enter data for grids with different numbers only. If any of the interface nodes is not specified at CSUPEL Data Entry, it is assumed that grid has the same number in the Superelement and Main structure input data files.

6. If all coincident nodes have the same numbers on the both side of the interface, continuation lines (+) can be avoided.

CTRIA3

Defines an isoparametric membrane-bending triangular element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3		MCID	ZOFFS	
	QID		T						

Example:

CTRIA3	111	203	31	74	75		+45.		

Field	**Contents**
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PSHELL or LAYERS entry. (Integer > 0).
Gi	Grid point identification numbers of connection points. (Integers > 0).
MCID	= Blank; Material orientation angle THETA=0.0 is assumed. = Integer; Material coordinate system MCID identification number. = Real; Material orientation angle THETA value in degrees.
ZOFFS	Offset from the surface of grid points to the element reference plane. (Real).
QID	QPRESS or DPRESS data entry identification number. (Integer ≥ 0).
T	Element thickness. (Real ≥ 0.0).

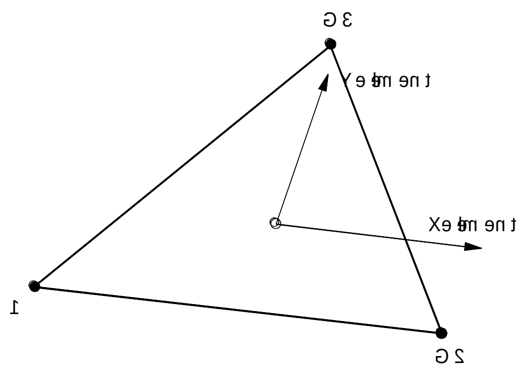


Figure 1. CTRIA3 Element Connection and Coordinate System.

Remarks:

1. Element membrane part is constant strain triangle (CST). Bending part is based on Mindlin-Reissner theory that includes transverse shear flexibility.

2. Element identification numbers must be unique with respect to all other CTRIA3 elements.

3. Grid points G1 through G3 must be ordered consecutively around the perimeter of the element. Element local x-y system is defined on Figure 1. Coordinate origin is at element C.G. X-axis is parallel to edge G_1G_2. Y-axis is normal to X-axis and oriented to form right hand coordinate system.

4. Element may be offset from the connection points by means of the ZOFFS field. Element reference plane is defined as a middle plane in the case of isotropic material, or as specified at LAYERS Bulk Data Entry in the case of composite material. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system.
   The use of ZOFFS will cause incorrect results in buckling analysis. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

5. If a coordinate system is used to define material orientation, then 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element. If MCID is real number, then material property angle θ is given in the MCID field. Angle θ is measured from edge G_1G_2 to material 0-axis.

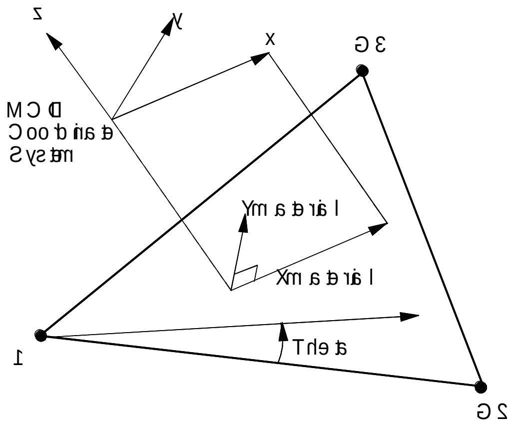


Figure 2. Material Coordinate System Definition.

CVISC

Defines a viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

Example:

CVISC	2	13	16	17					

Field	Contents
EID	Unique element identification number. (Integer > 0).
PID	Property identification number of a PVISC entry. (Integer > 0).
G1, G2	Grid point identification numbers of connection points. (Integer > 0, G1 ≠ G2)

Remarks:

1. Element identification numbers must be unique with respect to all other CVISC elements.

Defines a dynamic concentrated force at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
DFORCE	TID	G	CID	IAX	P0	T0	T1	T2	
	T3								

Example:

DFORCE	7	5	6	3	10.	1.	2.		

Field	**Contents**
TID	Load type identification number. (Integer ≠ 0). > 0 Tabular load case < 0 TABLE Bulk Data Entry identification number
G	Grid point identification number. (Integer > 0).
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0).
IAX	Force direction code. (Integer > 0, 1=x, 2=y, 3=z).
P0	Force amplitude. (Real).
T0, T1, T2, T3	Data used to define force-time function. (Real).

Remarks:

1. A CID of zero or blank (default), references the Basic coordinate system.

2. IAX component specify force direction (x, y or z) in Basic coordinate system. If CID > 0, force direction is defined upon CID system type and IAX component:

CID system	IAX=1	IAX=2	IAX=3
RECTANGULAR	P_X	P_Y	P_Z
CYLINDRICAL	P_R	P_θ	P_Z
SPHERICAL	P_R	P_θ	P_ϕ

Table 1. Force Component Definition.

3. Dynamic force-time function is defined by TID identification. If TID is larger than zero than it represents function identification number as per Table 2. Table specify 18 periodic and non-periodic force-time functions. If smaller than zero, TID represents TABLE Bulk Data Entry identification number with abscise represents time, and ordinate represents force.

4. Force amplitude P0 and values T0, T1, T2, T3 are used for defining force-time functions in Table 2. TO is starting time.

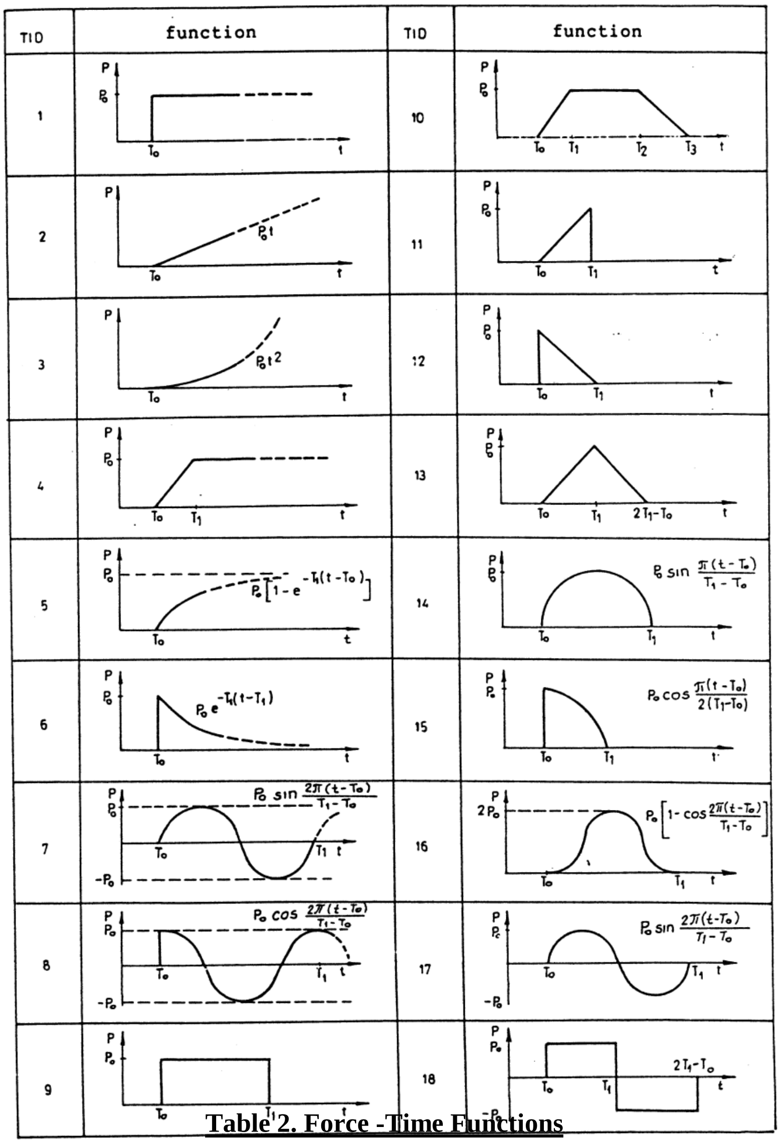


Table 2. Force -Time Functions

Defines a dynamic concentrated moment at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
DMOMENT	TID	G	CID	IAX	P0	T0	T1	T2	
	T3								

Example:

DMOMENT	7	5	6	3	10.	1.	2.		

Field	Contents
TID	Load type identification number. (Integer ≠ 0). > 0 Tabular load case < 0 TABLE Bulk Data Entry identification number
G	Grid point identification number. (Integer > 0).
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0).
IAX	Moment component code. (Integer > 0, 1=x, 2=y, 3=z).
P0	Moment amplitude. (Real).
T0, T1, T2, T3	Data used to define moment-time function. (Real).

Remarks:

1. A CID of zero or blank (default), references the Basic coordinate system.

2. IAX component specify moment axis (x, y or z) in Basic coordinate system. If CID > 0, moment axis is defined upon CID system type and IAX component:

CID system	IAX=1	IAX=2	IAX=3
RECTANGULAR	M_X	M_Y	M_Z
CYLINDRICAL	M_R	M_θ	M_Z
SPHERICAL	M_R	M_θ	M_ϕ

Table 1. Moment Axis Definition.

3. Dynamic moment-time function is defined by TID identification. If TID > 0 than it represents function identification number as per Table 2. Table specify 18 periodic and nonperiodic moment-time functions. If smaller than zero, TID represents TABLE Bulk Data Entry identification number with abscise represents time, and ordinate represents moment.

4. Moment amplitude P0 and values T0, T1, T2, T3 are used for defining moment-time functions in Table 2. TO is starting time.

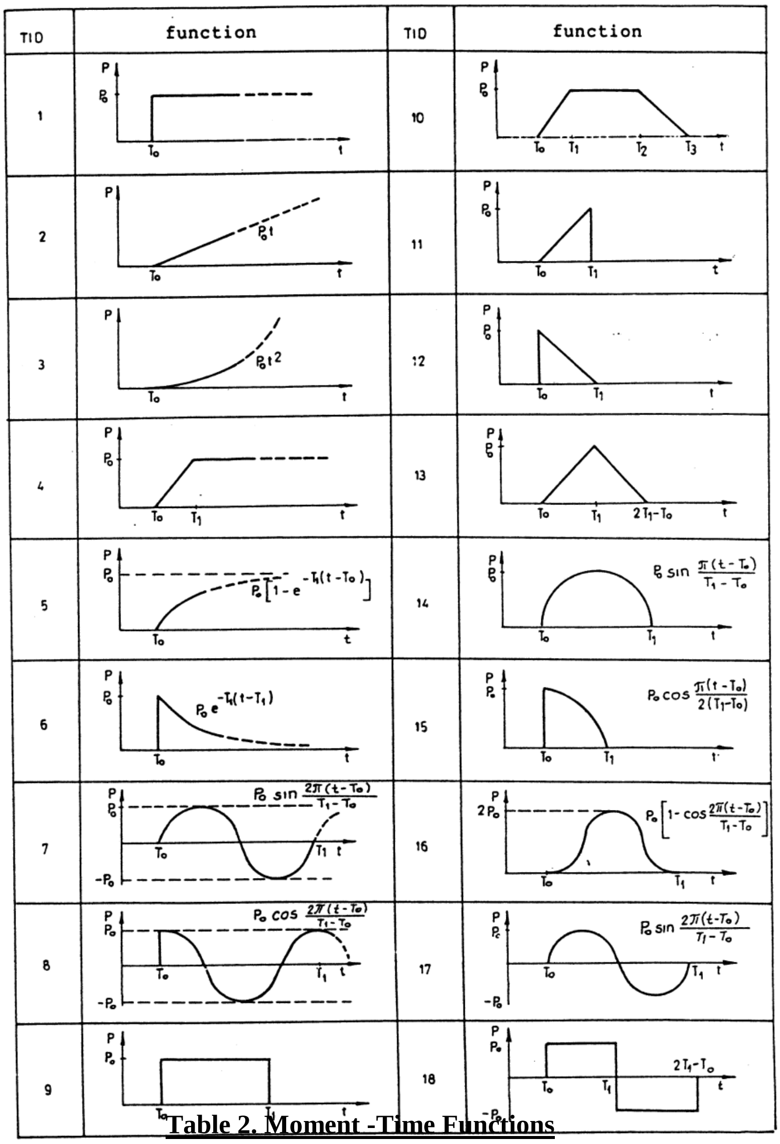


Table 2. Moment -Time Functions

Defines a dynamic pressure.

Format:

1	2	3	4	5	6	7	8	9	10
DPRESS	QID	TID	G	P0	T0	T1	T2	T3	

Example:

DPRESS	22	7	100	0.1	1.	2.			

Field	**Contents**
QID	Dynamic Pressure identification number. (Integer > 0).
TID	Load type identification number. (Integer ≠ 0). > 0 Tabular load case < 0 TABLE Bulk Data Entry identification number
G	Pressure grid point identification number. (Integer ≥ 0).
P0	Pressure amplitude. (Real).
T0, T1, T2, T3	Data used to define pressure-time function. (Real).

Remarks:

1. Dynamic pressure-time function is defined by TID identification. If TID is larger than zero than it represents function identification number as per Table 2. Table specify 18 periodic and non-periodic pressure-time functions. If smaller than zero, TID represents TABLE Bulk Data Entry identification number. The pressure-time function is defined by discrete point method with ab-scise represents time, and ordinate represents pressure.

2. Pressure amplitude P0 and values T0, T1, T2, T3 are used for defining pressure-time functions in Table 1. T0 is starting time.

3. Pressure is applied normal to the element surface.

4. Pressure point is used to define direction of the applied pressure. Positive direction is defined by vector connecting pressure point and element C.G.

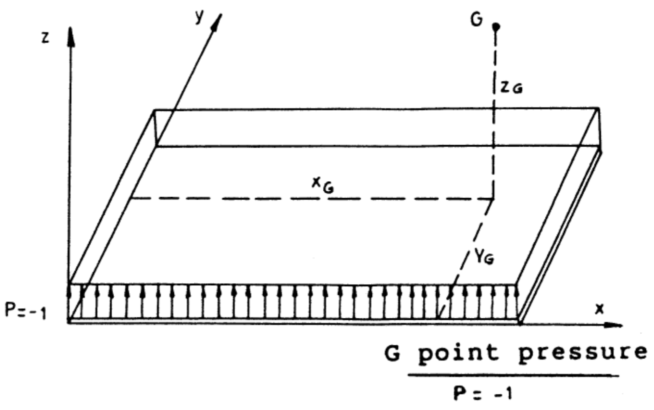


Figure 1. Pressure Direction, Pressure Point Convention

5. In the case when pressure point is not specified, convention for positive pressure direction is as follows:

- For plane elements (CTRIA3, CQUAD4, CSHELL3, CSHELL4) positive pressure value represents element local Z-axis direction.

- For solid elements (CSOLID) positive value specify direction normal to element face and toward the element.

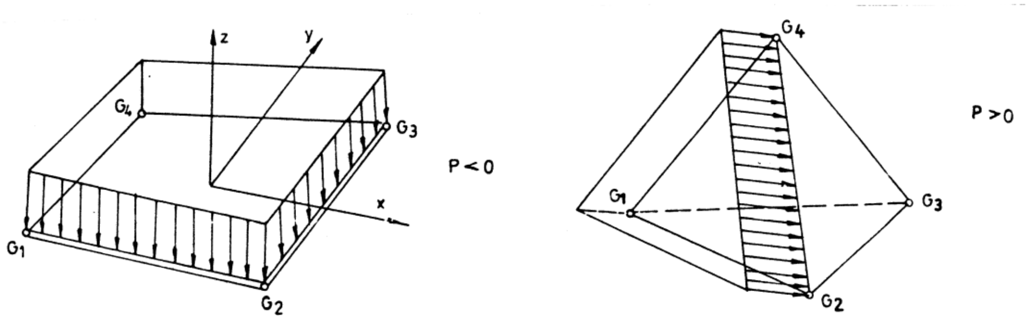


Figure 2. Pressure Direction, Element Convention

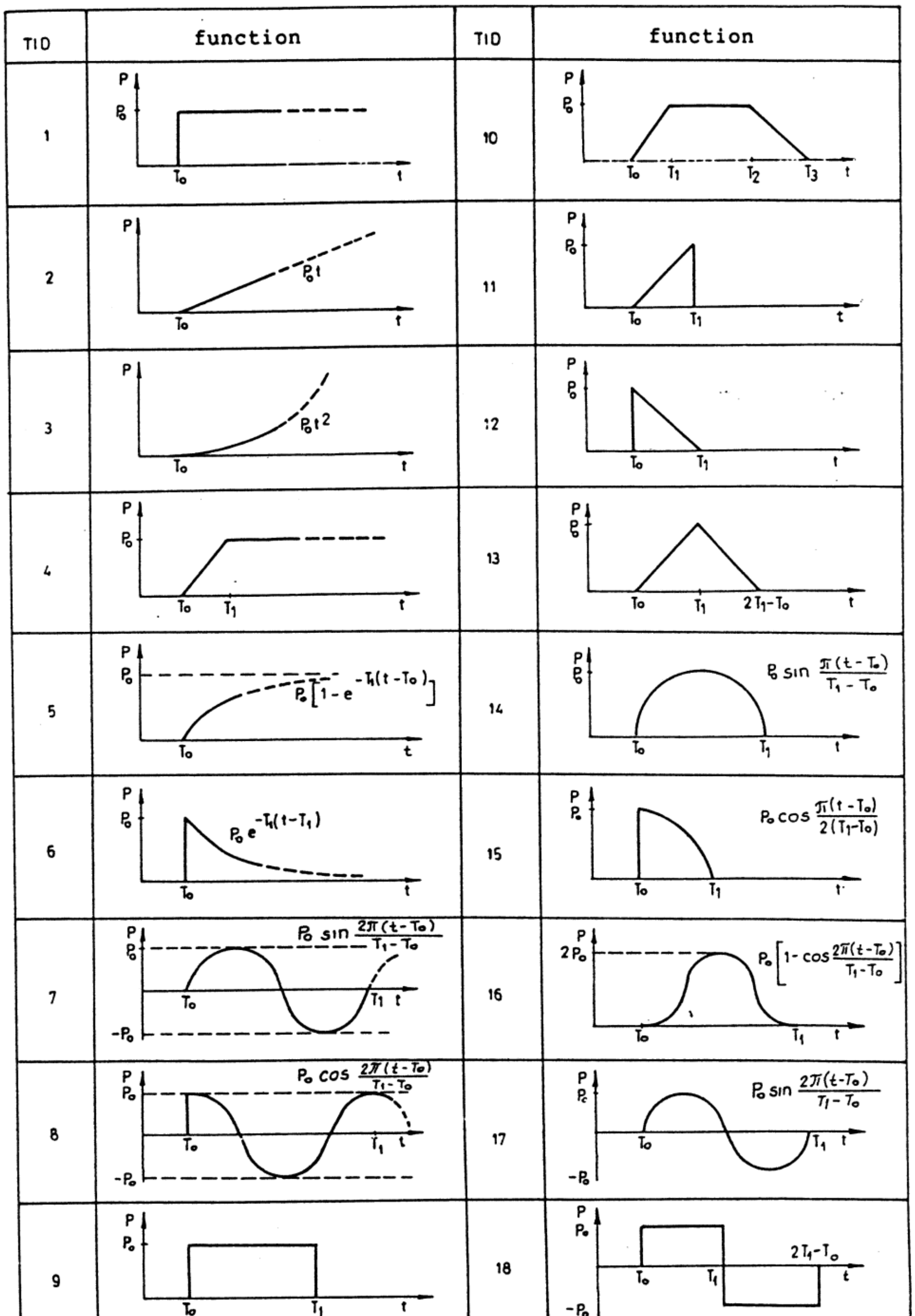


Table 3. Pressure -Time Functions

ENDDATA

Defines the end of Bulk Data Entry block.

Format:

1	2	3	4	5	6	7	8	9	10
ENDDATA									

Example:

ENDDATA									

Remarks:

1. Last entry in the Bulk Data Block. All data in the file positioned after this entry are ignored.

Defines a static concentrated force at grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	P1	P2	P3		

Example:

FORCE	2	5	6	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).
F	Scale factor. (Real).
P1, P2, P3	Components of a force vector measured in coordinate system CID. (Real).

Remarks:

1. The static force applied to grid point G is given by

$$\vec{P} = F\,(P1\,\vec{i} + P2\,\vec{j} + P3\,\vec{k})$$

2. A CID of zero or blank (default) references the Basic coordinate system. If CID > 0, force components depends of CID system type :

CID system	P1	P2	P3
RECTANGULAR	P_X	P_Y	P_Z
CYLINDRICAL	P_R	P_θ	P_Z
SPHERICAL	P_R	P_θ	P_ϕ

Table 1. Force Components Definition.

FORCE1

Defines a static concentrated force by specification of a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example:

FORCE1	2	31	-27.6	17	22				

Field	Contents
SID	Load set identification number. (Integer > 0).
G	Grid point identification number. (Integer > 0).
F	Magnitude of the force. (Real).
G1, G2	Grid point identification numbers. (Integer > 0; G1 and G2 must not be coincident).

Remarks:

1. The static force applied to grid point G is given by

$$\vec{P} = F\,\vec{n}$$

where $\vec{n}$ is a unit vector parallel to a vector from G1 to G2.

Defines group of elements for a free body load calculation.

Format:

1	2	3	4	5	6	7	8	9	10
FREEBODY	FID	CID							
	"ELEM"	E1	E2	E3	E4	E5	E6	E7	
	E8	E9	E10	-etc-					
	"NODE"	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	G11	G12	G13	-etc-		

Example:

FREEBODY	101	5							
	ELEM	17	19	21	89				
	NODE	12	13	25	35	95	11		

Alternate Format and Example:

FREEBODY	FID	CID							
	"ELEM"	E1	"THRU"	E2					
		E3	"THRU"	E4	-etc-				
	"NODE"	G1	"THRU"	G2					
		G3	"THRU"	G4	-etc-				

FREEBODY	101								
	ELEM	5001	THRU	5223					

Field	**Contents**
FID	Free body identification number. (Integer > 0).
CID	Coordinate system identification number. Used to output forces, moments and summary values. (Integer ≥ 0, Default = Blank).
"ELEM"	List of elements starts.
E1, E2, ... , En	Element identification numbers. (Integer ≥ 0).
"NODE"	List of grid points belonging to free body group starts. (Optional).
G1, G2, ... , Gn	Grid point identification numbers. (Integer ≥ 0).

Remarks:

1. Free body group output has to be activated by FREEBODY Executive Control command.

2. Free body group is created from all listed elements. Program calculates free body grid point forces and moments which are acting on a such defined group by summing contributions from all listed elements. By default **all** grid points belonging to free body group are included. If list of grid points is supplied ("NODE"), then only forces, moments and summary values based on listed grids are printed. This procedure allows sectional cuts and checking the sum of forces and moments.

3. By default all forces and moments are output in DISPLACEMENT coordinate system. This can be changed by specifying CID coordinate system. If specified, forces, moments and summary values are output in CID system.

4. FEMAP and PATRAN forces and moments are output in BASIC coordinate system.

5. The input data entry format may have unlimited number of continuation lines (+).

6. Free body load for a spoiler fitting is shown at Figure 1. The four plate elements that create the fitting are listed on the FREEBODY Data Entry. Free body load is shown in the form of grid point forces at hinge and fitting /spoiler joint.

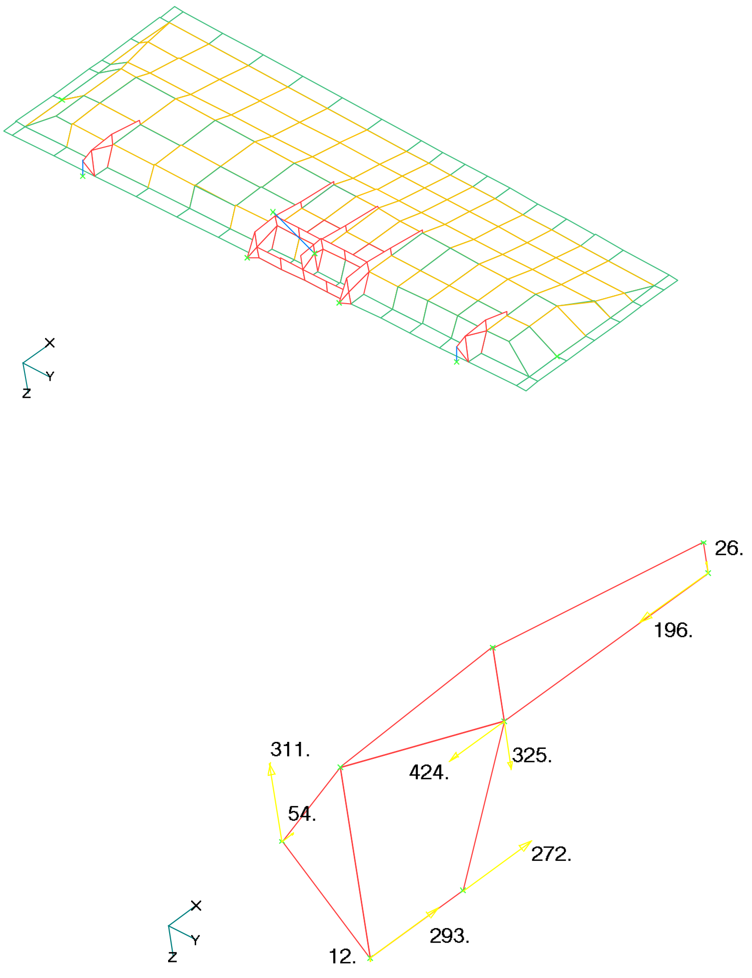


Figure 1. Spoiler Fitting Free Body Load

FREQ

Defines a set of frequencies to be used in the solution of frequency response problems (SOL=6,8,9)

Format:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	F10	-etc-					

Example:

FREQ	99	2.95	3.22	5.7	11.2	15.6	20.4	25.4	
	27.2	28.0							

Field	**Contents**
SID	Set identification number. (Integer > 0).
Fi	Frequency value in Hz. (Real ≥ 0.0).

Remarks:

1. Frequency sets must be selected with Executive Control command FREQUENCY = SID.

2. All FREQ, FREQ1 and FREQ2 entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.

Defines a set of frequencies to be used in the solution of frequency response problems (SOL=6,8,9) by specification of a starting frequency, frequency increment, and number of increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example:

FREQ1	3	0.1	10.0	99					

Field	Contents
SID	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. FREQ1 sets must be selected with Executive Control command FREQUENCY = SID.

2. The units of F1 and DF are cycles per unit time [Hz].

3. The frequencies defined by this entry are given by:

$$f_i = F1 + DF \cdot (i - 1)$$

where i = 1, 2, ... , (NDF+1).

4. All FREQ, FREQ1 and FREQ2 entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.

FREQ2

Defines a set of frequencies to be used in the solution of frequency response problems (SOL=6,8,9) by specification of a starting frequency, frequency increment, and number of logarithmic increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example:

FREQ2	5	0.1	10.0	20					

Field	Contents
SID	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. FREQ2 sets must be selected with Executive Control command FREQUENCY = SID.

2. The units of F1 and DF are cycles per unit time [Hz].

3. The frequencies defined by this entry are given by:

$$f_i = F1 \cdot e^{(i-1)d}$$

where $d = \frac{1}{NF} \ln \frac{F2}{F1}$, and i = 1, 2, ... , (NF+1).

4. All FREQ, FREQ1 and FREQ2 entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.

GRID

Defines the location of a geometric grid point, the directions of its displacements, and its permanent single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	ICOR	X1	X2	X3	IDIS	PS	ISUP	

Example:

GRID	2	3	1.	-2.	1.0		1456		

Field	**Contents**
ID	Grid point identification number. (Integer > 0).
ICOR	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank).
X1, X2, X3	Location of the grid point in coordinate system ICOR. (Real; Default = 0.0).
IDIS	Identification number of coordinate system in which the 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 the grid point. (Any of the integers 1 through 6 or blank).
ISUP	Superelement interface grid point identification. (Integer ≥ 0 or blank).

Remarks:

1. The grid point identification number ID must be unique with respect to all other GRID points.

2. A blank or zero (default) in the ICOR and IDIS fields refers to the basic coordinate system.

3. The meaning of X1, X2, and X3 depends on the type of coordinate system ICOR as follows :

SYSTEM TYPE	X1	X2	X3
RECTANGULAR	X	Y	Z
CYLINDRICAL	R	θ (degrees)	Z
SPHERICAL	R	θ (degrees)	ϕ (degrees)

Table 1. Grid Coordinates

4. ISUP larger than zero defines grid point at superelement interface. In superelement generation procedure (SOL = 4), structure stiffness and load are reduced to interface grid points having ISUP > 0. Generated superelement has reduced stiffness matrix which links those grids only.

GRIDA

Defines the location of an aerodynamic grid point, and directions of its displacements.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDA	ID	ICOR	X1	X2	X3	IDIS			

Example:

GRIDA	2	3	1.	-2.	1.0				

Field	Contents
ID	Grid point identification number. (Integer > 0).
ICOR	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank).
X1, X2, X3	Location of the grid point in coordinate system ICOR. (Real; Default = 0.0).
IDIS	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are defined at the grid point. (Integer ≥ 0 or blank).

Remarks:

1. The grid point identification number ID must be unique with respect to all other GRIDA points.

2. A blank or zero (default) in the ICOR and IDIS fields refers to the basic coordinate system.

3. Aerodynamic mesh (GRIDA points) is used to define geometry for aerodynamic elements (CSTRIP).

4. Aerodynamic mesh (GRIDA points) has to be linked to structural mesh (GRID points) via SPLINE Bulk Data Entry. If GRIDA is not linked to structural mesh, point is grounded with all deflections equal to zero.

5. GRIDA points do not contribute to total number of equations. Degrees of freedom are made dependent from structural deflections via SPLINE linking.

3. The meaning of X1, X2, and X3 depends on the type of coordinate system ICOR as follows :

SYSTEM TYPE	X1	X2	X3
RECTANGULAR	X	Y	Z
CYLINDRICAL	R	θ (degrees)	Z
SPHERICAL	R	θ (degrees)	ϕ (degrees)

Table 1. Grid Coordinates

GUST

Defines the stationary gust for use in aeroelastic response analysis (SOL=9).

Format:

1	2	3	4	5	6	7	8	9	10
GUST	SID	WY	WZ	TID					

Example:

GUST	11	0.0	2.335E-3	98					

Field	Contents
SID	Gust set identification number. (Integer > 0).
WY	Side gust scale factor (gust side velocity/forward velocity). (Real).
WZ	Vertical gust scale factor (gust vertical velocity/forward velocity). (Real).
TID	Set identification number of TABLE entry that gives T(f) versus frequency in [Hz]. If blank, than constant T(f) = 1.0 is assumed. (Integer ≥ 0 or blank).

Remarks:

1. The GUST entry must be selected with the Executive Control command GUST = SID.

2. The gust side (Y) and vertical (Z) angle components are given in Aerodynamic coordinate system as specified at AERO Bulk Data Entry. The components are given as

$$W_Y(f) = WY \cdot T(f) \cdot e^{i\Omega \cdot (t - \frac{X - X_o}{V})} \qquad W_Z(f) = WZ \cdot T(f) \cdot e^{i\Omega \cdot (t - \frac{X - X_o}{V})}$$

where T(f) : frequency dependent scaling function.
X : X-location of aero element C.P. point downstream.
X_O : X-location of aero system origin.
V : aircraft forward speed.
$\Omega = 2\pi f$: excitation frequency.

3. The unit gust velocity (WY=1/V or WZ=1/V) is suggested. The actual r.m.s. value is entered on the TABRNDG entry.

Defines initial displacements for transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
IDISP	G	U1	U2	U3	FI1	FI2	FI3		

Example:

IDISP	32	1.		0.5					

Field	Contents
G	Grid point identification number. (Integer > 0).
U1, U2, U3	Grid point initial translations. (Real; Default = 0.0).
FI1, FI2, FI3	Grid point initial rotations in radians. (Real; Default = 0.0).

Remarks:

1. Defines initial displacements for transient response analysis.
2. Translation and rotation components are specified in grid DISPLACEMENT coordinate system.

INCLUDE

Inserts an external file to the input file. INCLUDE statement may appear anywhere within the Bulk data section of the input data file.

Format:

1	2	3	4	5	6	7	8	9	10
INCLUDE	"FILENAME"								

Field	Contents
FILENAME	Physical filename of the external file to be inserted.

Example:

```
TITLE = "TV005 - Spring-mass system force response"
SOL = 5
LAMBDA = 2
INCREM = 12
RESPONSE  0.  3.36
BEGIN BULK
$
INCLUDE  "SPRING.DAT"
$
ENDDATA
```

Remarks:

1. INCLUDE statement cannot be nested - no INCLUDE statements may appear inside the external file.

Defines initial velocities for transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
IVELO	G	U1	U2	U3	FI1	FI2	FI3		

Example:

IVELO	32	5.		0.5					

Field	**Contents**
G	Grid point identification number. (Integer > 0).
U1, U2, U3	Grid point initial linear velocities. (Real; Default = 0.0).
FI1, FI2, FI3	Grid point initial angular velocities in radian / s. (Real; Default = 0.0).

Remarks:

1. Defines initial velocities for transient response analysis.
2. Translation and rotation components are specified in grid DISPLACEMENT coordinate system.

LAYERS

Defines the properties of a multilayered composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
LAYERS	LID	Z0	NSM	MCID	QID	TREF	GE	LAM	
	MID1	T1	THETA1	FC1	MID2	T2	THETA2	FC2	
	MID3	T3	THETA3	FC3	-etc-				

Example:

LAYERS	181	-1.127						SYM	
	1	.144	+45.		1	.144	-45.		
	2	.211	90.		2	.211	0.		

Field	Contents
LID	Layer property identification number. (Integer > 0).
Z0	Z distance in element coordinate system from the reference plane to the bottom surface. (Real; Default = -1/2 the thickness of the element).
NSM	Non-structural mass per unit area . (Real).
MCID	Material coordinate system identification number. (Integer ≥ 0).
QID	QPRESS or DPRESS data entry identification number. (Integer ≥ 0).
TREF	Reference temperature. (Real).
GE	Structural element damping coefficient GE=2∗C/Co. (Real; Default=0.0).
LAM	Symmetric laminate option. If LAM="SYM" only plies on the one side of the element middle plane are specified. (Character or blank. If blank, all plies must be specified).
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. If MIDi = 0 or blank, ID is set equal to preceding layer material identification number. (Integer ≥ 0 or blank).
Ti	Thickness of the various plies. If equal to zero ply is ignored. (Real or blank).
THETAi	Ply material orientation angle in degrees. Angle is measured from element material system 0-axis to the ply material 1-axis. (Real or blank).
FCi	Failure Criterion (Integer = 1 ÷ 11 or blank)

Remarks:

1. TOP element surface is on the positive side of element local Z-axis. The plies input starts at bottom surface toward opposite side. Stresses and strains are output in ply 1-2 system.

2. Figure 1 shows symmetric laminate as defined in Data Entry Example. It is assumed that there is no offset of the reference plane from the grid point plane (ZOFFS=0.0 at element data entry).

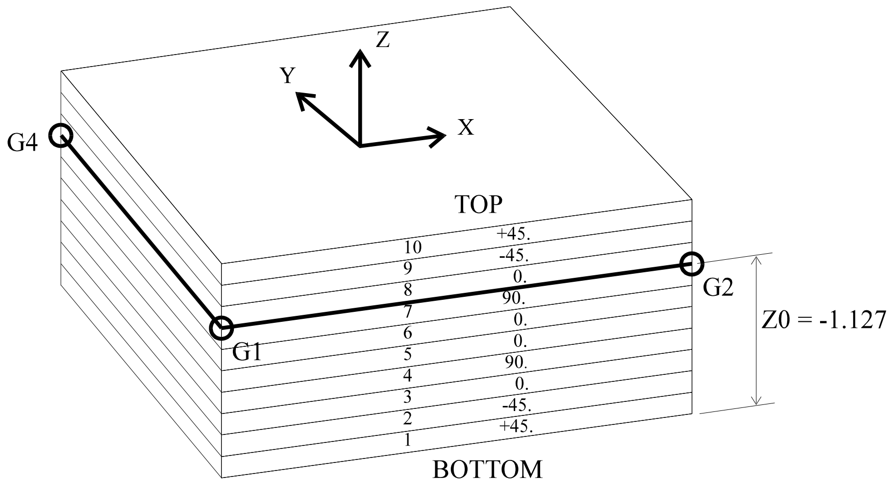


Figure 1. Laminate Definition.

3. Number of continuation lines is not limited. It is recommended to define one ply at each continuation entry, and make input more clear :

LAYERS	181	-1.127							
+	1	.144	+45.						
+	1	.144	-45.						
+	2	.211	90.						
+	2	.211	90.						
+	2	.211	90.						

4. TREF is used to calculate thermal loads and initial strain and stress for static sequences.

5. Coordinate system MCID is used if orientation field is left blank at element entry (i.e. CQUAD4). 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element.

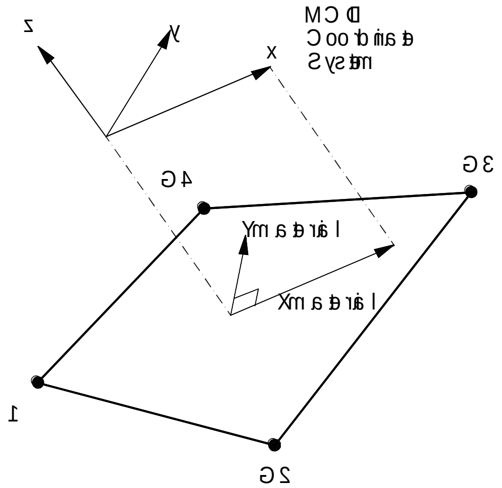


Figure 2. Material Coordinate System Definition.

6. On the output program prints in-plane stresses, strains and Margin of Safety for each ply. Output is controlled via PARAM PLATE Executive Data Entry. For each ply the Margin of Safety criterion is selected by FC option, or if that field is blank, criterion is established via appropriate input at MAT2 entry. The fibre M.S. is defined as minimal value from all margins calculated for that particular ply, i.e.

$$MS_{fiber} = \min(MS_1, MS_2, MS_{12})$$

Program identifies failure mode at the output as well. The following are output definitions

Failure Mode = 1 MS_1 is the lowest, direction 1 is critical
Failure Mode = 2 MS_2 is the lowest, direction 2 is critical
Failure Mode = 3 MS_{12} is the lowest, shear is critical

7. The following are the Margin of Safety criterions as used in the program:

Symbols Definition:

σ_1 , σ_2 , τ_{12}	: stresses in material 1-2 system
ε_1 , ε_2 , γ_{12}	: strains in material 1-2 system
F_1 = (Ft or Fc $)_1$	: stress allowable in 1-direction
F_2 = (Ft or Fc $)_2$	: stress allowable in 2-direction
F_{12}	: shear stress allowable
S_1 = (St or Sc $)_1$	: strain allowable in 1-direction
S_2 = (St or Sc $)_2$	: strain allowable in 2-direction
S_{12}	: shear strain allowable

<u>FC = 1 Maximum Stress Criterion</u>

$$MS_1 = \frac{F_1}{\sigma_1} - 1. \qquad MS_2 = \frac{F_2}{\sigma_2} - 1. \qquad MS_{12} = \frac{F_{12}}{\tau_{12}} - 1.$$

<u>FC = 2 Maximum Strain Criterion</u>

$$MS_1 = \frac{S_1}{\varepsilon_1} - 1. \qquad MS_2 = \frac{S_2}{\varepsilon_2} - 1. \qquad MS_{12} = \frac{S_{12}}{\gamma_{12}} - 1.$$

<u>FC = 3 Maximum Stress Criterion - Tape</u>

$$MS_1 = \frac{F_1}{\sigma_1} - 1. \qquad MS_{12} = \frac{F_{12}}{\tau_{12}} - 1.$$

<u>FC = 4 Maximum Strain Criterion - Tape</u>

$$MS_1 = \frac{S_1}{\varepsilon_1} - 1. \qquad MS_{12} = \frac{S_{12}}{\gamma_{12}} - 1.$$

<u>FC = 5 Quadratic Stress Criterion</u>

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\sigma_1}{F_1}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\sigma_2}{F_2}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1.$$

FC = 6 Quadratic Strain Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\varepsilon_1}{S_1}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\varepsilon_2}{S_2}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1.$$

FC = 7 Quadratic Stress Criterion - Tape

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\sigma_1}{F_1}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{F_2}{\sigma_2} - 1.$$

FC = 8 Quadratic Strain Criterion - Tape

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\varepsilon_1}{S_1}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{S_2}{\varepsilon_2} - 1.$$

FC = 9 Hashin Stress Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\sigma_1^t}{F_1^t}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1. \quad \text{or} \quad MS_1 = \frac{F_1^c}{\sigma_1^c} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\sigma_2}{F_2}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1.$$

FC = 10 Hashin Strain Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\varepsilon_1^t}{S_1^t}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1. \quad \text{or} \quad MS_1 = \frac{S_1^c}{\varepsilon_1^c} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\varepsilon_2}{S_2}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1.$$

FC = 11 Tsai-Hill Criterion

$$MS = \frac{1}{\sqrt{\left(\frac{\sigma_1}{F_1}\right)^2 - \frac{\sigma_1\sigma_2}{F_1^2} + \left(\frac{\sigma_2}{F_2}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1.$$

Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, PLOAD2, PLOAD4, QLOAD1, QLOAD2, QPRESS, AVECTOR, ROTATIO, TEMP, TEMPD, TLOAD and AESTAT entries.

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc-						

Example:

LOAD	101	1.5	1.0	5035	6.25	10			

Field	Contents
SID	Load set identification number. (Integer > 0).
S	Overall scale factor. (Real).
Si	Scale factor on Li. (Real).
Li	Load set identification numbers defined on entry types listed above. (Integer ≥ 0)

Remarks:

1. The load vector {P} is defined by

$$\{P\} = S\sum_i S_i P_{Li}$$

2. Load set IDs must be unique.

3. Load set ID is selected by Case Control command SUBCASE = SID.

4. LOAD entry cannot reference a set identification number defined by another LOAD entry.

LOADF

Scale factors for nodal static forces and moments.

Format:

1	2	3	4	5	6	7	8	9	10
LOADF	SID	Rx	Ry	Rz	Mx	My	Mz		

Example:

LOADF	2	1.0	0.	2.0	1.	1.	1.		

Field	Contents
SID	Load set identification number. (Integer > 0).
Rx, Ry, Rz	Force multipliers in BASIC coordinate system. (Real).
Mx, My, Mz	Moment multipliers in BASIC coordinate system. (Real).

Remarks:

1. Load multipliers are scale factors that are used to multiply nodal load components in X, Y and Z axis of the BASIC coordinate system. Nodal load is made from all FORCE, FORCE1, MOMENT and MOMENT1 Bulk Data Entries

2. If this entry is omitted, than default for all multipliers Rx through Mz is equal to one (1.0).

3. If this entry is defined and multiplier field is blank, multiplier is zero (0.0)

Scale factors for element static loads, such as pressure, force / unit length, aero pressures and super-element load.

Format:

1	2	3	4	5	6	7	8	9	10
LOADQ	SID	Rx	Ry	Rz	Mx	My	Mz		

Example:

LOADQ	2	1.0	0.	2.0	1.	1.	1.		

Field	Contents
SID	Load set identification number (Integer > 0).
Rx, Ry, Rz	Force multipliers in BASIC coordinate system. (Real).
Mx, My, Mz	Moment multipliers in BASIC coordinate system. (Real).

Remarks:

1. Load multipliers are scale factors that are used to multiply element load vector in X, Y and Z axis of the BASIC coordinate system. Element load is made from element pressure loads, CBAR and CBEAM distributed loads, CSTRIP aero load, and CSUPEL element loads.

2. If this entry is omitted, than default for all multipliers Rx through Mz is equal to one (1.0).

3. If this entry is defined and multiplier field is blank, multiplier is zero (0.0)

MAT1

Defines the material properties for linear isotropic material.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS						

Example:

MAT1	5	7400.		0.33	2.7E-6	13.E-6	70.		
	42.7	42.7	27.						

Field	**Contents**
MID	Material identification number. (Integer > 0).
E	Young’s modulus. (Real ≥ 0.0 or blank).
G	Shear modulus. (Real ≥ 0.0 or blank).
NU	Poisson’s ratio. (-1.0 < Real ≤ 0.5 or blank).
RHO	Mass density. (Real).
A	Thermal expansion coefficient. (Real).
TREF	Reference temperature for the calculation of thermal loads. (Real, Default=0.0 if A is specified).
GE	Structural element damping coefficient GE=2∗C/Co. (Real or blank).
ST, SC, SS	Stress limits for tension, compression, and shear used only to compute margins of safety in certain elements; they have no effect on the computational procedure. (Real).

Remarks:

1. The material identification number must be unique in respect to all other MAT1, MAT2 and MAT3 entries.

2. Mass density RHO is used to compute mass for all structural elements. Mass matrix is used in dynamic analysis. In combination with AVECTOR and ROTATIO accelerations, mass matrix is included in calculating inertia and rotational loads.

3. The following rules apply when E, G or NU are blank:

   a) E and G may not both be blank.

   b) If one of E, G, or NU is blank, than it will be computed from the equation :

$$G = \frac{E}{2(1 + NU)}$$

4. Stress limits for tension, compression, and shear used only to compute margins of safety in certain elements; they have no effect on the computational procedure.

5. A and TREF are used to calculate thermal loads and initial strain and stress. The initial strain is defined by following expression

$$\varepsilon_o = A(T - TREF)$$

   where T is applied temperature.

6. TREF and GE are ignored if the MAT1 entry is referenced by a LAYERS entry.

Defines the material properties for orthotropic linear material.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	FC	STRN						

Example:

MAT2	171	11790.	880.	.34	570.	300.	200.	1.56E-6	
	1.4E-6	2.E-4	70.	0.0090	0.0072	0.0090	0.0072	0.0144	
			STRAIN						

Field	Contents
MID	Material identification number. (Integer > 0).
E1	Modulus of elasticity in longitudinal direction, also defined as the fibre direction or 1-direction) . (Real ≠ 0.0).
E1	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction) . (Real ≠ 0.0).
NU12	Poisson's ratio. ($\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction). $\nu_{21} = \varepsilon_1/\varepsilon_2$ for uniaxial loading in 2-direction is calculated by relation $\nu_{12} E_2 = \nu_{21} E_1$
G12	In-plane shear modulus. (Real > 0.0).
G1Z, G2Z	Transverse shear modulus for shear in 1-Z and 2-Z planes, respectively. (Real ≥ 0.0; Blank or zero implies infinite transverse shear stiffness).
RHO	Mass density. (Real).
Ai	Thermal expansion coefficient in i-direction. (Real)
TREF	Reference temperature for the calculation of thermal loads. (Real, Default=0.0 if Ai are specified).
Xt, Xc Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the 1-direction (Xt, Xc), and 2-direcion (Yt, Yc). Required for Tsai-Hill and M.S. calculations. Default value for Xc is Xt, and default value for Yc is Yt. (Real).
S	Allowable stress or strain for in-plane shear. (Real).

GE	Structural element damping coefficient GE=2∗C/Co. (Real or blank).
FC	Failure Criterion (Integer = 1 ÷ 11 or blank. Default = 1).
STRN	Indicates whether Xt, Xc, Yt, Yc and S are stress or strain allowable: = "STRESS" for stress allowable (Default or blank) = "STRAIN" for strain allowable.

Remarks:

1. The material ID number must be unique in respect to all other MAT1, MAT2 , MAT3 entries.

2. If STRN = "STRAIN" indicates that Xt, Xc, Yt, Yc and S are strain allowable, UNA calculates stress allowable on the following way:

$$F_1^t = X_t E_1 \qquad F_2^t = Y_t E_2 \qquad F_{12} = S\ G_{12}$$
$$F_1^c = X_c E_1 \qquad F_2^c = Y_c E_2$$

   or, vice versa, if STRN = "STRESS" (blank) indicates that Xt - S are stress allowable, UNA calculates strain allowable on the following way:

$$S_1^t = \frac{X_t}{E_1} \qquad S_2^t = \frac{Y_t}{E_2} \qquad S_{12} = \frac{S}{G_{12}}$$
$$S_1^c = \frac{X_c}{E_1} \qquad S_2^c = \frac{Y_c}{E_2}$$

3. On the output program prints in-plane stresses, strains and Margin of Safety for each ply. Output is controlled via PARAM PLATE Executive Data Entry. For each ply the Margin of Safety criterion is selected by FC option. The fibre M.S. is defined as minimal value from all margins calculated for that particular ply, i.e.

$$MS_{fiber} = \min(MS_1, MS_2, MS_{12})$$

   Program identifies failure mode at the output as well. The following are output definitions

   Failure Mode = 1 MS_1 is the lowest, direction 1 is critical
   Failure Mode = 2 MS_2 is the lowest, direction 2 is critical
   Failure Mode = 3 MS_{12} is the lowest, shear is critical

4. The following are the Margin of Safety criterions as used in the program:

Symbols Definition:

σ_1 , σ_2 , τ_{12}	: stresses in material 1-2 system
ε_1 , ε_2 , γ_{12}	: strains in material 1-2 system
F_1 = (Ft or Fc $)_1$	: stress allowable in 1-direction
F_2 = (Ft or Fc $)_2$	: stress allowable in 2-direction
F_{12}	: shear stress allowable
S_1 = (St or Sc $)_1$	: strain allowable in 1-direction
S_2 = (St or Sc $)_2$	: strain allowable in 2-direction
S_{12}	: shear strain allowable

FC = 1 Maximum Stress Criterion

$$MS_1 = \frac{F_1}{\sigma_1} - 1. \qquad MS_2 = \frac{F_2}{\sigma_2} - 1. \qquad MS_{12} = \frac{F_{12}}{\tau_{12}} - 1.$$

FC = 2 Maximum Strain Criterion

$$MS_1 = \frac{S_1}{\varepsilon_1} - 1. \qquad MS_2 = \frac{S_2}{\varepsilon_2} - 1. \qquad MS_{12} = \frac{S_{12}}{\gamma_{12}} - 1.$$

FC = 3 Maximum Stress Criterion - Tape

$$MS_1 = \frac{F_1}{\sigma_1} - 1. \qquad MS_{12} = \frac{F_{12}}{\tau_{12}} - 1.$$

FC = 4 Maximum Strain Criterion - Tape

$$MS_1 = \frac{S_1}{\varepsilon_1} - 1. \qquad MS_{12} = \frac{S_{12}}{\gamma_{12}} - 1.$$

FC = 5 Quadratic Stress Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\sigma_1}{F_1}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\sigma_2}{F_2}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1.$$

FC = 6   Quadratic Strain Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\varepsilon_1}{S_1}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\varepsilon_2}{S_2}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1.$$

FC = 7   Quadratic Stress Criterion - Tape

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\sigma_1}{F_1}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{F_2}{\sigma_2} - 1.$$

FC = 8   Quadratic Strain Criterion - Tape

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\varepsilon_1}{S_1}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1. \qquad MS_2 = \frac{S_2}{\varepsilon_2} - 1.$$

FC = 9   Hashin Stress Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\sigma_1^t}{F_1^t}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1. \quad \text{or} \quad MS_1 = \frac{F_1^c}{\sigma_1^c} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\sigma_2}{F_2}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1.$$

FC = 10 Hashin Strain Criterion

$$MS_1 = \frac{1}{\sqrt{\left(\frac{\varepsilon_1^t}{S_1^t}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1. \quad \text{or} \quad MS_1 = \frac{S_1^c}{\varepsilon_1^c} - 1. \qquad MS_2 = \frac{1}{\sqrt{\left(\frac{\varepsilon_2}{S_2}\right)^2 + \left(\frac{\gamma_{12}}{S_{12}}\right)^2}} - 1.$$

FC = 11 Tsai-Hill Criterion

$$MS = \frac{1}{\sqrt{\left(\frac{\sigma_1}{F_1}\right)^2 - \frac{\sigma_1\sigma_2}{F_1^2} + \left(\frac{\sigma_2}{F_2}\right)^2 + \left(\frac{\tau_{12}}{F_{12}}\right)^2}} - 1.$$

MAT3

Defines the material properties for orthotropic 3-D linear material.

Format:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	E1	E2	E3	NU12	NU13	NU23	RHO	
	GE	G12	G13	G23	A1	A2	A3	TREF	

Example:

MAT3	15	0.	0.	15.5				6.41E-8	
		5.033	2.689	0.					

Field	**Contents**
MID	Material identification number. (Integer > 0).
E1	Modulus of elasticity in L-direction, also defined as 1-direction . (Real).
E2	Modulus of elasticity in W-direction, also defined as 2-direction . (Real).
E3	Modulus of elasticity in T-direction, also defined as 3-direction . (Real).
NU12, NU13, NU23	Poisson's ratios. (Real).
RHO	Mass density. (Real).
GE	Structural element damping coefficient GE=2∗C/Co. (Real or blank).
G12	Shear modulus in L-W plane, also defined as 1-2 plane. (Real).
G13	Shear modulus in L-T plane, also defined as 1-3 plane. (Real).
G23	Shear modulus in W-T plane, also defined as 2-3 plane. (Real).
Ai	Thermal expansion coefficient in i-direction. (Real)
TREF	Reference temperature for the calculation of thermal loads. (Real, Default=0.0 if Ai are specified).

Remarks:

1. The material identification number must be unique in respect to all other MAT1, MAT2 and MAT3 entries.

2. Mass density RHO is used to compute mass for all structural elements. Mass matrix is used in dynamic analysis. In combination with AVECTOR and ROTATIO accelerations, mass matrix is included in calculating inertia and rotational loads.

3. Orthotropic 3-D material is intended to be used with solid elements.

4. It is necessary to use Executive Data Entry

   PARAM SOLID 4 = 2

   in order to disable incompatible modes if this material is used. A large numerical error may happen otherwise.

5. Ai and TREF are used to calculate thermal loads and initial strain and stress. The initial strain is defined by following expression

$$\varepsilon_{o1} = A_1(T - TREF)$$
$$\varepsilon_{o2} = A_2(T - TREF)$$
$$\varepsilon_{o3} = A_3(T - TREF)$$

   where T is applied temperature. Shear strains γ_{12},, γ_{13},, γ_{23}, are not affected by temperature.

MOMENT

Defines a static concentrated moment at grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	F	M1	M2	M3		

Example:

MOMENT	2	5	6	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).
F	Scale factor. (Real).
M1, M2, M3	Components of a moment vector measured in coordinate system CID. (Real).

Remarks:

1. The static moment applied to grid point G is given by

$$\vec{M} = F\,(M1\,\vec{i} + M2\,\vec{j} + M3\,\vec{k})$$

2. A CID of zero or blank (default) references the Basic coordinate system. If CID > 0, moment components depends of CID system type :

CID system	M1	M2	M3
RECTANGULAR	M_X	M_Y	M_Z
CYLINDRICAL	M_R	M_θ	M_Z
SPHERICAL	M_R	M_θ	M_ϕ

Table 1. Moment Components Definition.

Defines a static concentrated moment by specification of a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example:

MOMENT1	2	31	-27.6	17	22				

Field	Contents
SID	Load set identification number. (Integer > 0).
G	Grid point identification number. (Integer > 0).
F	Magnitude of the moment. (Real).
G1, G2	Grid point identification numbers. (Integer > 0; G1 and G2 must not be coincident).

Remarks:

1. The static moment applied to grid point G is given by

$$\vec{M} = F \vec{n}$$

where $\vec{n}$ is a unit vector parallel to a vector from G1 to G2.

MPC

Defines a multipoint constraint equation of the form:

$$\sum_j A_j u_j = 0$$

where u_j represents degree of freedom C_j at grid point.

Format:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc-				

Example:

MPC	5	28	3	2.5	125	2	-1.0		
		5	2	0.5					

Field	**Contents**
SID	Set identification number. (Integer > 0).
Gj	Grid point identification number. (Integer ≥ 0.0).
Cj	Component number. (Any one of the integers 1 through 6).
Aj	Coefficient. (Real).

Remarks:

1. Multipoint constraint set must be selected with Executive Control command MPC = SID.

2. The first degree of freedom (G1, C1) in the sequence is defined to be dependent DOF.

3. Forces of the MPC equations are recovered under nodal reactions output section.

4. The following are conditions that MPC equation has to satisfy :
   - No SPC or SPC1 constraints are allowed at dependent degrees of freedom.
   - No dependent degrees of freedom are allowed at superelement boundary (SOL=4).
   - Dependent degrees of freedom cannot be dependent or independent in another MPC entry, COUPG entry or RBE2 rigid element.

Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

Format:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc-						

Example:

MPCADD	101	2	3	1	6				

Field	**Contents**
SID	Set identification number. (Integer > 0).
Sj	Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer ≥ 0.0).

Remarks:

1. Multipoint constraint set must be selected with Executive Control command MPC = SID.

2. The Sj must be unique and may not be the identification number of a multipoint constraint set defined by another MPCADD entry.

3. MPCADD entries take precedence over MPC entries. If both have the same SID, only the MPCADD entry will be used.

PBAR

Defines properties of a beam element (CBAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM	QID	
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2							

Example:

PBAR	39	6	2.9		5.97				
			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).
I1, I2	Area principal moments of inertia. (Real; I1 ≥ 0.0, I2 ≥ 0.0).
J	Torsional constant. (Real).
NSM	Non-structural mass per unit length. (Real).
QID	QLOAD1 or QLOAD2 load data entry pointer. (Integer ≥ 0).
K1, K2	Area factor for shear. (Real).
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0).

Remarks:

1. The property identification number must be unique in respect to all other PBAR entries.
2. PBAR entries may only reference MAT1 material entries.

3. See CBAR Data Entry for description of the bar element geometry.

4. The transverse shear stiffness per unit length in planes 1 and 2 are K1*A*G and K2*A*G, respectively, where G is the shear modulus. The default values for K1 and K2 are infinite; in other words, the transverse shear flexibilities are set equal to zero.

5. 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. The following are expressions used to calculate stresses at points due to bar end loads:

   End A: $$\sigma_x = -\frac{P_x}{A} + \frac{M_z}{I_z} y - \frac{M_y}{I_y} z$$

   End B: $$\sigma_x = +\frac{P_x}{A} - \frac{M_z}{I_z} y + \frac{M_y}{I_y} z$$

   where end forces and moments are given in element coordinate system.

PBEAM

Defines properties of a simple beam element (CBEAM entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	AX	AY	AZ	J	IY	IZ	
	C1	C2	D1	D2	E1	E2	F1	F2	
	QID	NSM							

Example:

PBEAM	39	6	2.9			0.1	4.0	5.97	
	2.	4.1							

Field	Contents
PID	Property identification number. (Integer > 0).
MID	Material identification number. (Integer > 0).
AX	Area of bar cross section. (Real).
AY, AZ	Shear areas. (Real).
J	Torsional constant. (Real).
IY, IZ	Area principal moments of inertia. (Real; $IY \geq 0.0$, $IZ \geq 0.0$).
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0).
QID	QLOAD1 or QLOAD2 load data entry pointer. (Integer ≥ 0).
NSM	Non-structural mass per unit length. (Real).

Remarks:

1. The property identification number must be unique in respect to all other PBEAM entries.
2. PBEAM entries may only reference MAT1 material entries.

3. See CBEAM Data Entry for description of the beam element geometry.

4. The transverse shear stiffness per unit length in planes x-y and x-z are AY*G and AZ*G, respectively, where G is the shear modulus. The default values for AY and AZ are infinite; in other words, the transverse shear flexibilities are set equal to zero.

5. The stress recovery coefficients C1 and C2, etc., are the y and z coordinates in the beam element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the beam. The following are expressions used to calculate stresses at points due to beam end loads:

   End A: $\sigma_x = -\frac{P_x}{A} + \frac{M_z}{I_z} y - \frac{M_y}{I_y} z$

   End B: $\sigma_x = +\frac{P_x}{A} - \frac{M_z}{I_z} y + \frac{M_y}{I_y} z$

   where end forces and moments are given in element coordinate system.

PLOAD2

Defines a uniform static pressure load applied to CMEMB, CSHEAR, CTRIA3, CQUAD4, CSHELL3 and CSHELL4 two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

Example:

PLOAD2	21	-3.6		4	16			2	

Alternate Format and Example:

PLOAD2	SID	P	EID1	"THRU"	EID2				

PLOAD2	1	30.4	16	THRU	48				

Field	Contents
SID	Load set identification number. (Integer > 0).
P	Pressure value. (Real).
EIDi	Element identification number. (Integer ≥ 0 or blank; for the "THRU" option, EID1 ≤ EID2).

Remarks:

1. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry (direction of the local z-axis).

Defines a static pressure load applied to CMEMB, CSHEAR, CTRIA3, CQUAD4, CSHELL3 and CSHELL4 two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4			

Example:

PLOAD4	2	1106	10.0	8.0	5.0				

Alternate Format and Example:

PLOAD4	SID	EID1	P1	P2	P3	P4	"THRU"	EID2	

PLOAD4	2	16	10.0				THRU	48	

Field	**Contents**
SID	Load set identification number. (Integer > 0).
EID, EID1, EID2	Element identification number. (Integer ≥ 0 or blank; for the "THRU" option, EID1 ≤ EID2).
P1, P2, P3, P4	Pressure load at corners. (Real or blank; If blank default for P2, P3, P4 is P1).

Remarks:

1. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry (direction of the local z-axis).

PROD

Defines properties of a rod element (CROD entry).

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

Example:

PROD	13	12	2.7	2.1					

Field	Contents
PID	Property identification number. (Integer > 0).
MID	Material identification number. (Integer > 0).
A	Area of the rod. (Real).
J	Torsion constant. (Real).
C	Coefficient to determine torsion shear stress. (Real; Default = 0.0).
NSM	Non-structural mass per unit length. (Real).

Remarks:

1. The property identification number must be unique in respect to all other PROD entries.
2. PROD entries may only reference MAT1 material entries.
3. Coefficient C is used to calculate torsion shear stress by the following expression :

$$.\tau = \frac{CM_\theta}{J}$$

where M_θ = torsional moment

PSHEAR

Defines properties of a shear element (CSHEAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	K13	K24			

Example:

PSHEAR	13	7	1.0		1	1			

Field	Contents
PID	Property identification number. (Integer > 0).
MID	Material identification number. (Integer > 0).
T	Default value for shear panel thickness. (Real).
NSM	Non-structural mass per unit area. (Real).
K13, K24	Option for automatic rod generation on element edges. (Integer ≥ 0).

Remarks:

1. The property identification number must be unique in respect to all other PSHEAR entries.
2. PSHEAR entries may reference MAT1 material entries.

1. Rod elements are generated on the shear panel edges on following way:

K13 = 0 Generation defined by Executive data Entry PARAM SHEAR 1 = [Num]
= 1 Generation requested for edges G_1G_2 , G_3G_4

K24 = 0 Generation defined by Executive data Entry PARAM SHEAR 2 = [Num]
= 1 Generation requested for edges G_2G_3 , G_4G_1

The effective cross section area is defined in a such way that equivalent moment of inertia is equal to element moment of inertia for in-plane bending.

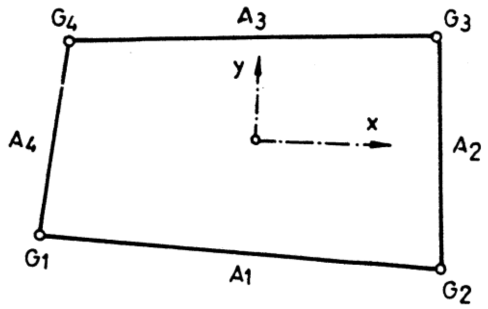


Figure 1. Generated rods on CSHEAR element

PSHELL

Defines the membrane, bending and transverse shear properties of thin shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T3	MID3	TS/T	NSM	
	MCID	QID							

Example:

PSHELL	203	4	1.5	4	1.3				

Field	Contents
PID	Property identification number. (Integer > 0).
MID1	Material identification number for the membrane. (Integer ≥ 0 or blank)
T	Default value for the membrane thickness. (Real).
MID2	Material identification number for bending. (Integer ≥ 0 or blank)
12I/T3	Bending stiffness parameter. (Real > 0.0; Default = 1.0).
MID3	Material identification number for transverse shear. (Integer ≥ 0 or blank)
TS/T	Transverse shear thickness divided by membrane thickness. (Real > 0.0; Default=0.8333333).
NSM	Non-structural mass per unit area. (Real).
MCID	Material coordinate system identification number. (Integer ≥ 0).
QID	QPRESS or DPRESS load data entry pointer. (Integer ≥ 0).

Remarks:

1. The property identification number must be unique in respect to all other PSHELL entries.
2. This entry is referenced by CMEMB, CTRIA3, CQUAD4, CSHELL3, CSHELL4 elements.
3. PSHELL entries may reference MAT1 and MAT2 material entries.

4. The results of leaving MID fields blank are:

   MID1 No membrane or coupling stiffness.
   MID2 No bending, coupling or transverse shear stiffness.
   MID3 No transverse shear flexibility (infinite transverse shear stiffness).

5. The structural damping (for dynamic runs) uses the values defined for the MID1 material.

6. Coordinate system MCID is used for orthotropic materials (MAT2) if orientation field is left blank at element entry (i.e. CQUAD4). 0-axis of the material coordinate system is determined by projecting the X-axis of the MCID coordinate system onto the surface of the element.

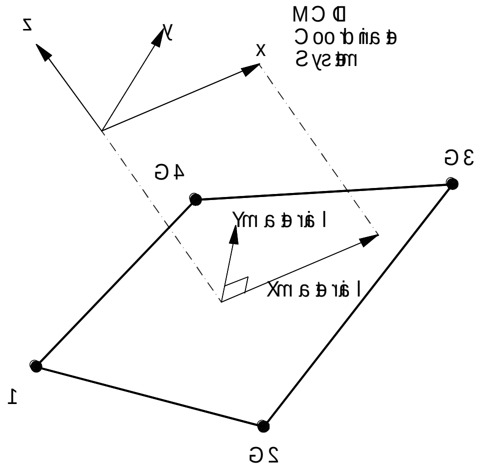


Figure 1. Material Coordinate System Definition

7. On the output program prints in-plane stresses, strain and Von-Misses strain and stress. Output is controlled via PARAM PLATE Executive entry. The following are definitions for output values:

   Von Misses stress : $$\sigma_{vmis} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$$

   Von Misses strain : $$\varepsilon_{vmis} = \sqrt{\frac{4}{9}\left(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x \varepsilon_y\right) + \frac{1}{3}\gamma_{xy}^2}$$

PSOLID

Defines properties of a solid elements (CSOLID entry).

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	MCID	QID	FID				

Example:

PSOLID	2	100	35		2				

Field	Contents
PID	Property identification number. (Integer > 0).
MID	Material identification number. (Integer > 0).
MCID	Material coordinate system definition. (Integer ≥ 0).
QID	QPRESS or DPRESS load data entry pointer. (Integer ≥ 0).
FID	Element face identification number. Pressure acts on this face. (Integer ≥ 0).

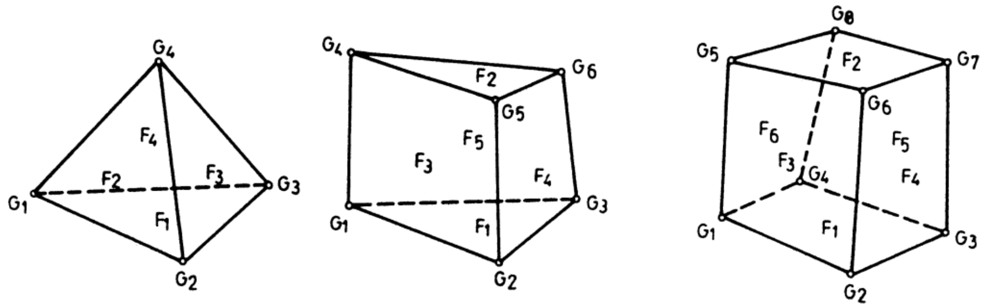
Remarks:

1. The property identification number must be unique in respect to all other PSOLID entries.

2. PSOLID entries may reference MAT1 and MAT3 material entries.

3. If a MCID coordinate system is used to define material orientation, then material coordinate system 1, 2 and 3-axis are equal to MCID x, y and z-axis, respectively. If MCID = 0 on this entry and material is orthotropic, it is assumed that material coordinate system is coincident with element local x, y, z system (1=x, 2=y, 3=z).

4. Pressure applied on the element is defined at QPRESS or DPRESS data entry. QID points to the proper entry and FID defines element face at which the pressure is applied (Table 1).



Type	Figure code	FID	Stress code	Face grids
	F_1	1	Z-	$G_1G_2G_3$
TETRA	F_2	2	Y-	$G_1G_2G_4$
	F_3	3	X+	$G_2G_3G_4$
	F_4	4	X-	$G_3G_1G_4$
	F_1	1	Z-	$G_1G_2G_3$
	F_2	2	Z+	$G_4G_5G_6$
PENTA	F_3	3	Y-	$G_1G_2G_5G_4$
	F_4	4	X+	$G_2G_3G_6G_5$
	F_5	5	X-	$G_3G_1G_4G_6$
	F_1	1	Z-	$G_1G_2G_3G_4$
	F_2	2	Z+	$G_5G_6G_7G_8$
HEXA	F_3	3	Y-	$G_1G_2G_6G_5$
	F_4	4	X+	$G_2G_3G_7G_6$
	F_5	5	Y+	$G_3G_4G_8G_7$
	F_6	6	X-	$G_4G_1G_5G_8$

Table 1. Solid Elements Face Identification

5. On the output program prints Von-Misses stress which is calculated by:

$$.Vmis = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{2} + \frac{(\sigma_y - \sigma_z)^2}{2} + \frac{(\sigma_z - \sigma_x)^2}{2} + 3\tau_{xy}^2 + 3\tau_{yz}^2 + 3\tau_{zx}^2}$$

PSTRIP

Defines a properties of an aerodynamic strip element (CSTRIP entry).

Format:

1	2	3	4	5	6	7	8	9	10
PSTRIP	PID	X/C				AREA	CLA	CMA	
	CL0	CM0							

Example:

PSTRIP	4105	0.15							

Alternate Format and Example:

PSTRIP	PID	CID	X	Y	Z	AREA	CLA	CMA	
	CL0	CM0							

PSTRIP	4105	5105	184.05	22.4	0.0	3.5E+5	0.0651	-0.011	
	0.029	0.001							

Field	Contents
PID	Property identification number. (Integer > 0).
X/C	Control point chord wise location. (Real or blank; Default=0.25).
AREA	Reference area of the aerodynamic panel. (Real; If blank or zero, then AREA=C∗B, where C is panel cord length, and B is panel width).
CLA, CMA	Coefficients of lift and pitching moment per angle of attack at control point. (Real or blank; Default CLA=2π, CMA=0.0).
CL0, CM0	Coefficients of lift and pitching moment for zero angle of attack at control point. (Real or blank; Default CL0=0.0, CM0=0.0).
CID	Coordinate system ID for defining control point location. (Integer ≥ 0).
X, Y, Z	Control point location in coordinate system CID. (Reals).

Remarks:

1. The property identification number must be unique in respect to all other PSTRIP entries.

2. X/C defines a control point location as a fraction of panel medium cord line length measured from the leading edge (0.0 ≤ X/C ≤ 1.0). All aerodynamic properties and loads that are based on strip panel aerodynamic theory will be derived at this control point.

3. X, Y, Z coordinates specify a control point location anywhere inside the panel. All aerodynamic properties and loads will be derived at this control point.

4. If CLA - CM0 are not specified (blank), a default values of CLA=2π, CMA=0.0 CL0=0.0, CMA=0.0 will be used regardless of the control point location (by default it is assumed that control point location is coincident with panel centre of pressure C.P.).

5. Aerodynamic lift Pz is oriented in z-axis direction, pitching moment My around y-axis, element local coordinate system (see CSTRIP). Stationary aerodynamic load is defined as follows:

$$P_Z = QS(C^W_{L0} + C^W_{L\alpha}\alpha) \quad \text{where} \quad C^W_{L0} = W_{L0}C_{L0}, \qquad C^W_{L\alpha} = W_{L\alpha}C_{L\alpha}$$

$$M_Y = QS(C^W_{M0} + C^W_{M\alpha}\alpha) \qquad C^W_{M0} = W_{M0}C_{M0}, \qquad C^W_{M\alpha} = W_{M\alpha}C_{M\alpha}$$

where

Q	: Dynamic pressure [P/L^2].
S	: Reference area as specified by AREA [L^2].
$C^W_{L\alpha}..C^W_{M0}$	: Corrected lift and moment coefficients CLA, CMA [1/rad], and CL0, CM0.
α	: Panel angle of attack [rad].

6. Correction factors WLA, WMA, WL0, WM0 are specified at WTFACT Bulk Data Entry. They are applied to aerodynamic properties in order to match desired data (experimental, analytical from CFD etc.). Factors will be applied in stationary and non-stationary aerodynamic regime in the form of scaling factors.

7. Aerodynamic loads are transferred from control point to structure via splining procedure (SPLINE Bulk Data Entry), or internally generated rigid link (if NODE is specified at aerodynamic element entry).

PVISC

Defines properties of a one-dimensional viscous element (CVISC entry).

Format:

1	2	3	4	5	6	7	8	9	10
PVISC	PID1	CE1	CR1		PID2	CE2	CR2		

Example:

PVISC	13	3.2	2.4						

Field	Contents
PIDi	Property identification number. (Integer ≥ 0).
CE1, CE2	Viscous coefficients for extension/compression. (Real).
CR1, CR2	Viscous coefficients for torsion. (Real).

Remarks:

1. The property identification number must be unique in respect to all other PVISC entries.
2. PVISC is used in frequency response type problems only (SOL=6,8,9).

QLOAD1

Defines Bar / Beam distributed load, form I.

Format:

1	2	3	4	5	6	7	8	9	10
QLOAD1	SID	QID	G1	G2	Q	X	Y	Z	

Example:

QLOAD1	3	2	101	130	0.5	-0.1			

Field	Contents
SID	Load set identification number. (Integer > 0).
QID	QLOAD1 entry identification number. (Integer > 0).
G1, G2	Grid point identification numbers. A vector defined from G1 to G2 represents direction of the applied distributed load. (Integer > 0).
Q	Force per unit length at grid G1. (Real).
X, Y, Z	Load gradients in Basic coordinate system. (Real).

Remarks:

1. The entry identification number QID must be unique in respect to all other QLOAD1 or QLOAD2 entries.

2. In order to be loaded elements have to point to QLOAD1 data entry via QID identification number at CBAR, CBEAM, PBAR or PBEAM data entries.

3. Q value represents force per unit length acting in direction defined by vector G1G2.

4. Load gradients X, Y, Z are used to create variable loading. At particular point in the space load is defined with following expression:

$$Q_p = Q + (x_p - x_{G1})X + (y_p - y_{G1})Y + (z_p - z_{G1})Z$$

where :	Q_p	: Force per unit length at point P
	Q	: Force per unit length at grid G1
	x_p, y_p, z_p	: Coordinates of the point P in Basic coordinate system
	x_{G1}, y_{G1}, z_{G1}	: Coordinates of the grid point G1 in Basic coordinate system
	X, Y, Z	: Load gradients

Bar element load at ends A and B is defined with the following expressions:

$$Q_A = Q + (x_A - x_{G1})X + (y_A - y_{G1})Y + (z_A - z_{G1})Z$$

$$Q_B = Q + (x_B - x_{G1})X + (y_B - y_{G1})Y + (z_B - z_{G1})Z$$

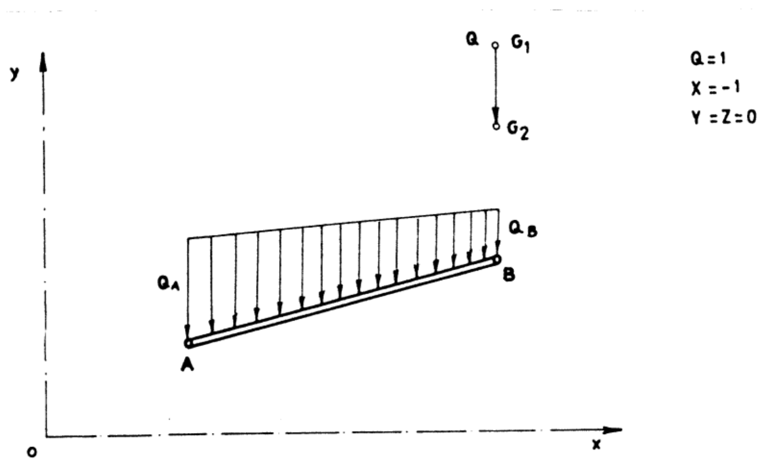


Figure 1. BAR element distributed load.

QLOAD2

Defines Bar / Beam distributed load, form II.

Format:

1	2	3	4	5	6	7	8	9	10
QLOAD2	SID	QID	CID	QXA	QXB	QYA	QYB	QZA	
	QZB								

Example:

QLOAD2	3	5	1			0.5	0.8	0.3	

Field	Contents
SID	Load set identification number. (Integer > 0).
QID	QLOAD2 entry identification number. (Integer > 0).
CID	Coordinate system for defining components QXA, QXB, .. QZB. (Integer ≥ 0). = 0 Basic coordinate system. = 1 Element coordinate system.
QXA, QXB	Force per unit length in x-direction at element ends A and B, respectively. x-direction is Basic or local element x-axis, as per CID option. (Real).
QYA, QYB	Force per unit length in y-direction at element ends A and B, respectively. y-direction is Basic or local element y-axis, as per CID option. (Real).
QZA, QZB	Force per unit length in z-direction at element ends A and B, respectively. z-direction is Basic or local element z-axis, as per CID option. (Real).

Remarks:

1. The entry identification number QID must be unique in respect to all other QLOAD1 or QLOAD2 entries.

2. In order to be loaded elements have to point to QLOAD2 data entry via QID identification number at CBAR, CBEAM, PBAR or PBEAM data entries.

3. QXA, QXB, ..., QZB values are defined as force per unit length at element ends A and B. If CID=1 than components are given in element local coordinate system (Figure 1). CID=0 indicates that components are given in Basic coordinate system (Figure 2).

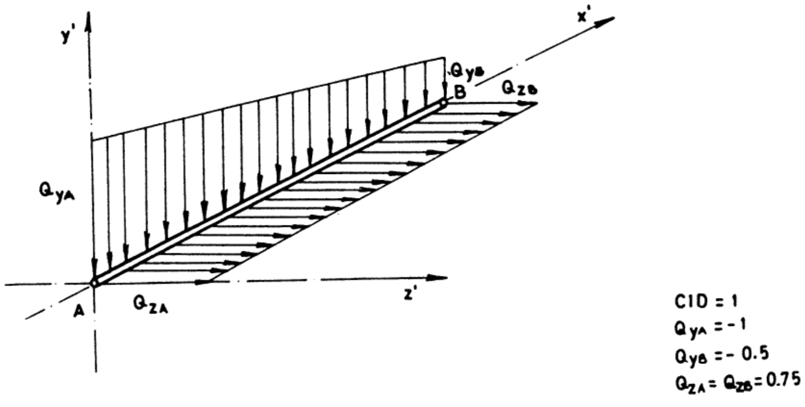


Figure 1. BAR distributed load, local system definition.

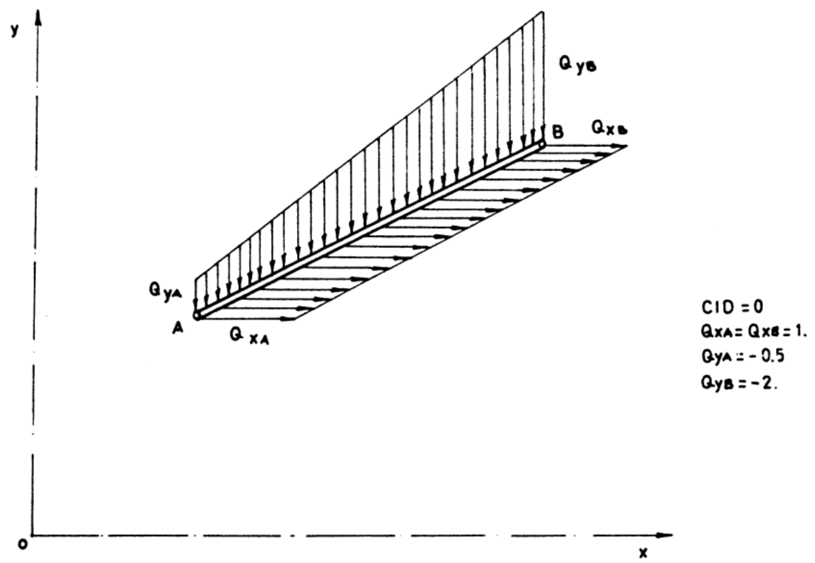


Figure 2. BAR distributed load, Basic system definition

QPRESS

Defines a static pressure load applied to CMEMB, CSOLID, CTRIA3, CQUAD4, SHELL3 and CSHELL4 elements.

Format:

1	2	3	4	5	6	7	8	9	10
QPRESS	SID	QID	G	Q	X	Y	Z	CID	

Example:

QPRESS	3	2	101	8.24E-4					

Field	Contents
SID	Load set identification number. (Integer > 0).
QID	QPRESS entry identification number. (Integer > 0).
G	Grid point identification number. Pressure Q is acting at this point in the space. (Integer ≥ 0).
Q	Pressure load. (Real).
X, Y, Z	Pressure gradients in CID coordinate system. (Real).
CID	Coordinate system identification number. Pressure gradients are defined in this coordinate system. (Integer ≥ 0; Blank or zero defines Basic coordinate system).

Remarks:

1. The entry identification number QID must be unique in respect to all other QPRESS entries.

2. In order to be loaded element has to point to QPRESS data entry via QID identification number at CMEMB, CTRIA3, CQUAD4, CSHELL3, CSHELL4 or PSOLID data entry.

3. Q value represents pressure load. Pressure is applied normal to the element surface. Load gradients X, Y , Z are used to create variable loading in the space. At particular element C.G. pressure is defined with following expression:

$$Q_{CG} = Q + (x_{CG} - x_G)X + (y_{CG} - y_G)Y + (z_{CG} - z_G)Z$$

where :

Q_{CG}	:	Pressure at element C.G.
Q	:	Pressure at grid point G.
x_{CG}, y_{CG}, z_{CG}	:	Coordinates of the element C.G. in CID coordinate system.
x_G, y_G, z_G	:	Coordinates of the grid point G in CID coordinate system.
X, Y, Z	:	Pressure gradients in CID coordinate system

Direction of the pressure is defined with a vector connecting grid point G and element C.G. If Q is positive, pressure is acting toward the element surface which can be seen from the grid point G. If negative, pressure is acting in opposite direction (Figure 1).

4. In the case when pressure point G is not specified (Figure 2), convention for positive pressure direction is as follows:

- For plane elements positive pressure direction is element local Z-axis.
- For solid elements positive direction is normal to element face and toward element.

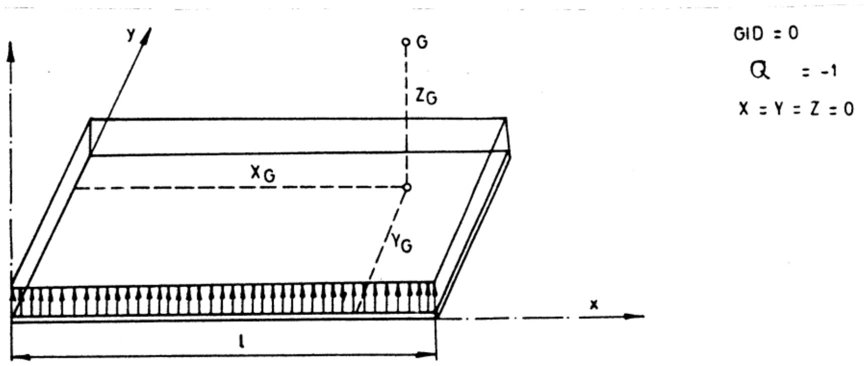


Figure 1. Pressure Direction, Pressure Point Convention

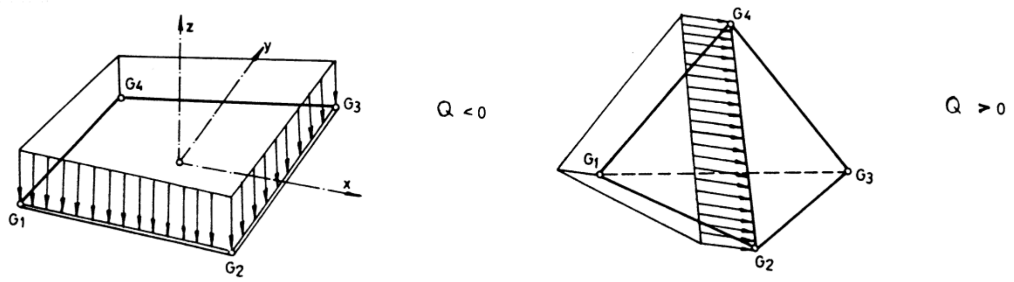


Figure 2. Pressure Direction, Element Convention

RANDPS

Defines load set power spectral density factors for use in random type analysis (SOL=8,9). The frequency dependent form is defined by:

$$\phi_{jk}(f) = (X + iY) \cdot G(f)$$

where f is frequency in [Hz].

Format:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

Example:

RANDPS	10	101	101	3.14	0.	99			

Field	Contents
SID	Random analysis set identification number. (Integer > 0).
J	Identification number of the excited load set. (Integer > 0).
K	Identification number of the applied load set. (Integer ≥ 0, K ≥ J).
X, Y	Components of the complex number. (Real).
TID	Identification number of TABRNDi entry that defines G(*f*). (Integer ≥ 0).

Remarks:

1. Set identification number must be selected with Executive Control command RANDOM=SID.
2. In order to combine and apply different PSD factors, create multiple RANDPS with same SID.
3. For auto spectral density J = K, X must be greater than zero, and Y must be equal to zero. Cross spectral density is defined with K > J.
4. J and K are pointing to source set number J at RLOAD Bulk Data entry, or SID identification number at GUST Bulk Data entry.
5. For TID = 0, G(*f*) = 1.0.

Defines a rigid 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:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc-					

Example:

RBE2	9	8	12	10	12	14	15	16	
	20								

Field	Contents
EID	Element identification number. (Integer > 0).
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned. (Integer > 0).
CM	Component numbers of the dependent degrees of freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0).
GMi	Grid point identification numbers at which dependent degrees of freedom are assigned. (Integer ≥ 0).

Remarks:

1. The components indicated by CM are made dependent at GMi grids.

2. Forces of the RBE2 element are recovered under nodal reactions output section.

3. The following are conditions that RBE2 element has to satisfy :
   - No SPC or SPC1 constraints are allowed at dependent degrees of freedom.
   - No dependent degrees of freedom are allowed at superelement boundary (SOL=4).
   - Dependent degrees of freedom cannot be dependent or independent in another MPC entry, COUPG entry or RBE2 rigid element.

RLOAD

Defines a frequency-dependent dynamic load of the form:

$$P(f) = \sum_j P_j(f) = \sum_j P_j^{stat} F_j(f) \cdot e^{i \cdot [\Omega t + \varphi_j(f)]}$$

where P_j^{stat} : source shape (static load set).
$F_j(f)$: frequency dependent scaling function.
$\varphi_j(f)$: frequency dependent phase function.
$\Omega = 2\pi f$: excitation frequency.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD	SID	J	TF	TP					

Example:

RLOAD	2	101	7						

Field	Contents
SID	Dynamic load set identification number. (Integer > 0).
J	Source number. It points to static set that gives source shape P_j^{stat}. (Integer > 0).
TF	Set identification number of TABLE entry that gives $F_j(f)$ versus frequency in [Hz]. (Integer ≥ 0). For TF = 0, $F_j(f)$ = 1.0.
TP	Set identification number of TABLE entry that gives phase $\varphi_j(f)$ in degrees versus frequency in [Hz]. (Integer ≥ 0). For TP = 0, $\varphi_j(f)$= 0.0.

Remarks:

1. Dynamic load sets must be selected with Executive Control command DLOAD = SID.

2. All sources Pj(f) from multiple RLOAD entries with the same SID identification number create dynamic load set P(f).

3. J points to SID number of the static load set made from FORCE, FORCE1, MOMENT, MOMENT1, PLOAD2, PLOAD4, QPRESS, QLOAD1 and QLOAD2 Bulk Data entries. This static set defines the shape P_j^{stat} for dynamic source Pj(f).

Defines a static loading condition due to angular velocity and/or acceleration.

Format:

1	2	3	4	5	6	7	8	9	10
ROTATIO	SID	GA	GB	RVEL	RACC				

Example:

ROTATIO	2	13	21	5.0	0.5				

Field	Contents
SID	Load case identification number. (Integer > 0).
GA, GB	Grid point identification numbers. Direction of rotation vector V is from GA to GB. (Integer > 0).
RVEL	Angular velocity. (Real).
RACC	Angular acceleration. (Real).

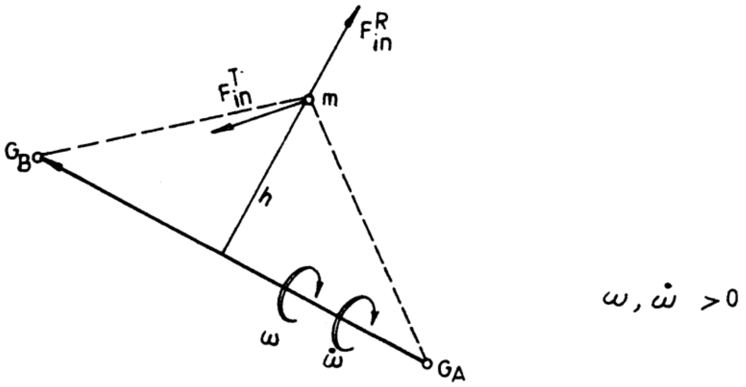


Figure 1. Rotation axis and inertial loads

Remarks:

1. Rotational axis vector V direction is from GA to GB.

2. Direction of ω and ω' are per right-hand rule definition.

3. Angular velocity ω and angular acceleration ω' are sources of radial and tangential inertial forces as per following expressions :

$A_R = h\omega^2$: Radial acceleration

$A_T = h\omega'$: Tangential acceleration

$\vec{P}_R^{in} = -m\vec{A}_R$: Radial inertial load

$\vec{P}_T^{in} = -m\vec{A}_T$: Tangential inertial load

Used to manually order the grid points. This entry is used to redefine the sequence of grid points to optimize bandwidth.

Format:

1	2	3	4	5	6	7	8	9	10
SEQGP	ID1	SEQID1	ID2	SEQID2	ID3	SEQID3	ID4	SEQID4	

Example:

SEQGP	5292	15.6	596	0.2			3	2	

Field	Contents
Idi	Grid point identification number. (Integer ≥ 0).
SEQIDi	Sequenced identification number. (Integer, Real).

Remarks:

1. Original grid number is not changed. Displacement values, nodal stress etc., will be refereed to this original number.
2. Assigned number can be any real or integer value. For example, if user wants to put grid with label 5392 between grid points 15 and 16 (in global system of equations), he can assign number 15.6 to this grid point.
3. SEQID has to be unique, and cannot be equal to any original ID grid numbers.
4. This data entry cannot be followed by continuation (+) entry.
5. It is possible to define one to four relations on each SEQGP data entry

SET

Defines a list of structural items (elements, grids etc.)

Format:

1	2	3	4	5	6	7	8	9	10
SET	SID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc-							

Example:

SET	3	31	62	93	124	16	17	18	
	19								

Alternate Format and Example:

SET	SID	ID1	"THRU"	ID2					

SET	3	31	THRU	62					

Field	Contents
SID	Unique set identification number. (Integer > 0)
IDi	List of structural items. (Integer ≥ 0 or "THRU"; For the "THRU" option ID1 $\leq$ ID2).

Defines a set of single-point constraints and enforced displacements.

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example:

SPC	3	101	135	-1.0					

Field	Contents
SID	Identification number of the single-point constraint set. (Integer > 0).
Gi	Grid point identification numbers. (Integer ≥ 0).
Ci	Component numbers. (Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0).
Di	Value of enforced displacements for all degrees of freedom designated by Gi and Ci. (Real).

Remarks:

1. Single-point constraint set must be selected with the Executive Control Entry SPC = SID.

SPC1

Defines a set of single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc-					

Example:

SPC1	3	2	1	3	10	9	6	5	
	2	8							

Alternate Format and Example:

SPC1	SID	C	G1	"THRU"	G2				

SPC1	3	12456	6	THRU	22				

Field	**Contents**
SID	Identification number of the single-point constraint set. (Integer > 0).
C	Component numbers. (Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0).
Gi	Grid point identification numbers. (Integer ≥ 0 or "THRU"; For the "THRU" option G1 ≤ G2).

Remarks:

1. Single-point constraint set must be selected with the Executive Control Entry SPC = SID.

Defines a linear spline for interpolating aero to structural mesh.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE	SID	CID							
	"AERO"	A1	A2	A3	A4	A5	A6	A7	
	A8	A9	A10	-etc-					
	"NODE"	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	G11	G12	G13	-etc-		

Example:

SPLINE	4101	101							
	AERO	31	32	33	34				
	NODE	3101	3102	3103	3105	3111	3107		

Alternate Format and Example:

SPLINE	SID	CID							
	"AERO"	A1	"THRU"	A2					
		A3	"THRU"	A4	-etc-				
	"NODE"	G1	"THRU"	G2					
		G3	"THRU"	G4	-etc-				

SPLINE	4101	101							
	AERO	31	THRU	34					
	NODE	3101	THRU	3111					

Field	**Contents**
SID	Spline identification number. (Integer > 0).
CID	Coordinate system identification number, y-axis defines spline axis (Integer > 0)
"AERO"	List of aerodynamic grids (GRIDA) starts.
A1, A2, ... , An	Aerodynamic grid points identification numbers. (Integer ≥ 0).
"NODE"	List of structural grids (GRID) starts.
G1, G2, ... , Gn	Structural grid points identification numbers. (Integer ≥ 0).

Remarks:

1. Spline identification number must be unique with respect to all other SPLINE entries.

2. Aero grid cannot be called in more than one SPLINE entry.

3. For a single aerodynamic panel all four corner grids must be part of one SPLINE entry in order to be regularly linked with structure. Mixed or partial splining (some of corner grids rather than all) is not allowed.

4. If all four corner aero grids are not splined to structure, they will be grounded with no deflections and no interaction between aero panel and structural entities. The exception is when panel control point is directly linked to structure via NODE input field at panel Bulk Data Entry. Direct linking overrides any link defined by SPLINE entry.

5. The CID is rectangular coordinate system which Y-axis specifies spline axis. It is recommended to align Y-axis with approximate spanwise locations for structural shear centres (wing, fin, horizontal tail), or with fuselage centreline.

TABLE

Defines a tabular function by discrete point method.

Format:

1	2	3	4	5	6	7	8	9	10
TABLE	TID								
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	
	X5	Y5	X6	Y6	-etc-				

Example:

TABLE	15								
	1.	0.	1.5	2.	1.5	4.	2.	4.	
	2.6	4.	3.0	3.	5.0	2.	5.0	-2.	
	6.	-2.	6.5	-1.0					

Field	Contents
TID	Table identification number. (Integer > 0).
Xi, Yi	Tabular values. (Real, $X_1 \le X_2 \le X_3 \le .... \le X_N$).

Remarks:

1. This entry is used to define arbitrary function (i.e. force-time).

2. TABLE uses the algorithm

$$y = y_T(x)$$

   where x is input to the table and y is returned. Linear interpolation is performed between the points.

3. If the last $X_N = 0.0$, it must be substituted by some small number (i.e. 10^{-30})

4. Figure 1 represents a typical x-y entry as per TABLE example from previous page.

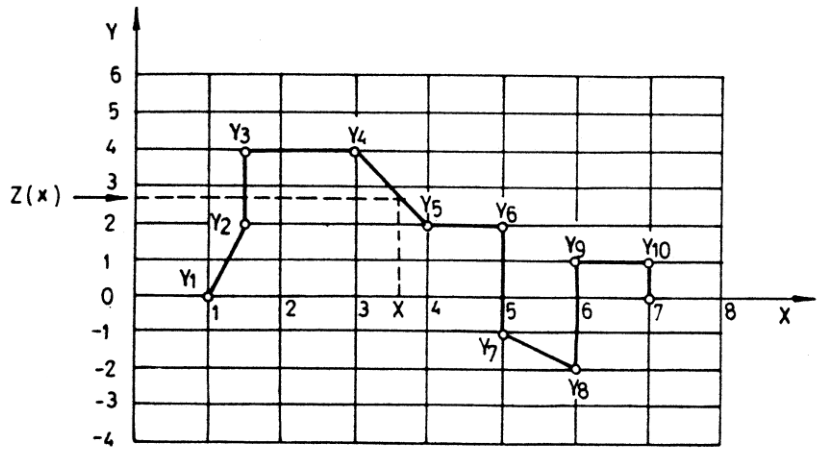


Figure 1. Example of Table definition.

TABRND1

Defines the power spectral density as a tabular function of frequency for use in random analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TABRND1	TID								
	f1	g1	f2	g2	f3	g3	-etc-		

Example:

TABRND1	15								
	2.5	0.0105	4.6	0.0155					

Field	Contents
TID	Table identification number. (Integer > 0).
fi	Frequency value in [Hz]. (Real, $f_1 \le f_2 \le f_3 \le .... \le f_N$).
gi	Power spectral density. (Real).

Remarks:

1. TABRND1 identification number TID must be referenced by RANDPS Bulk Data entry.

2. TABRND1 uses the algorithm

$$g = g_T(f)$$

where f is input to the table and g is returned. Linear interpolation is performed between the points, and linear extrapolation outside the table.

Defines the power spectral density (PSD) of a gust for aeroelastic response analysis (SOL=9).

Format:

1	2	3	4	5	6	7	8	9	10
TABRNDG	TID	TYPE	L/U	WG					

Example:

TABRNDG	902	1	5.838	85.					

Field	**Contents**
TID	Table identification number. (Integer > 0).
TYPE	PSD type: von Karman (TYPE=1) or Dryden model (TYPE=2). (Integer = 1 or 2)
L/U	Scale of turbulence divided by velocity (units of time). (Real).
WG	Root-mean-square gust velocity. (Real).

Remarks:

1. TABRNDG identification number TID must be referenced by RANDPS Bulk Data entry.

2. The power spectral density is given by

$$\phi_g(\Omega)=2(WG)^2(L/U)\frac{1+2(p+1)k^2(L/U)^2\Omega^2}{\left[1+k^2(L/U)^2\Omega^2\right]^{p+3/2}}$$

where

Type	p	k
1 = von Karman	1/3	1.339
2 = Dryden	1/2	1.0

and $\Omega=2\pi f$. The units of $\phi_g(\Omega)$ are velocity squared per frequency (f).

TEMP

Defines temperature at grid points for determination of thermal load.

Format:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

Example:

TEMP	3	94	316.2	49	219.8				

Field	Contents
SID	Temperature set identification number. (Integer > 0).
Gi	Grid point identification number. (Integer ≥ 0).
Ti	Temperature. (Real).

Remarks:

1. The temperature set SID is identification number of the static load set. Thermal loads that are generated by this temperature and thermal expansion factors, will be added to all static loads (FORCE, MOMENT etc.) that have the same SID.

2. Nodal temperatures are used to calculate element temperature by simply averaging the connecting grid values. This temperature is used if no element direct temperature is specified by TLOAD entry.

3. Temperature loads are generated for CROD, CBAR, CBEAM, CMEMB, CSOLID, CTRIA3, CQUAD4, CSHELL3 and CSHELL4 elements. Thermal nodal loads are generated by utilizing the assumption that temperature is constant inside the volume of a single element (no thermal gradients between element boundaries or through the thickness).

Defines a temperature value for all grid points of the structural model which have not been given a temperature on a TEMP entry.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID	T							

Example:

TEMPD	3	550.							

Field	Contents
SID	Temperature set identification number. (Integer > 0).
Gi	Grid point identification number. (Integer ≥ 0).
Ti	Temperature. (Real).

Remarks:

1. The temperature set SID is identification number of the static load set. Thermal loads that are generated by this temperature and thermal expansion factors, will be added to all static loads (FORCE, MOMENT etc.) that have the same SID.

2. Nodal temperatures are used to calculate element temperature by simply averaging the connecting grid values. This temperature is used if no element direct temperature is specified by TLOAD entry.

3. Temperature loads are generated for CROD, CBAR, CBEAM, CMEMB, CSOLID, CTRIA3, CQUAD4, CSHELL3 and CSHELL4 elements. Thermal nodal loads are generated by utilizing the assumption that temperature is constant inside the volume of a single element (no thermal gradients between element boundaries or through the thickness).

TLOAD

Defines a temperature value for structural elements.

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD	SID	T	EID1	EID2	EID3	EID4	EID5	EID6	

Example:

TLOAD	21	-120.0		4	16			2	

Alternate Format and Example:

TLOAD	SID	T	EID1	"THRU"	EID2				

TLOAD	1	550.	16	THRU	48				

Field	**Contents**
SID	Temperature set identification number. (Integer > 0).
T	Temperature. (Real).
EIDi	Element identification number. (Integer ≥ 0 or blank; for the "THRU" option, EID1 ≤ EID2).

Remarks:

1. The temperature set SID is identification number of the static load set. Thermal loads that are generated by this temperature and thermal expansion factors, will be added to all static loads (FORCE, MOMENT etc.) that have the same SID.

2. Directly specified element temperature has priority over element temperature obtained by averaging connecting grid temperatures (TEMP, TEMPD entries).

3. Temperature loads are generated for CROD, CBAR, CBEAM, CMEMB, CSOLID, CTRIA3, CQUAD4, CSHELL3 and CSHELL4 elements. Thermal nodal loads are generated by utilizing the assumption that temperature is constant inside the volume of a single element (no thermal gradients between element boundaries or through the thickness).

Defines an aerodynamic correction factors to be applied on aero elements (CSTRIP).

Format:

1	2	3	4	5	6	7	8	9	10
WTFACT	WID								
	EID1	WLA1	WMA1	WL01	WM01				
	EID2	WLA2	WMA2	-etc-					

Example:

WTFACT	1405								
	84101	0.1555	0.00	1.00	1.00				
	84102	0.3455	0.00						

Field	Contents
WID	Set identification number. (Integer > 0).
EIDi	Aero element numbers. (Integer ≥ 0 or blank).
WLAi, WL0i	Correction factor for lift coefficients CLA and CL0. (Reals).
WMAi, WM0i	Correction factor for moment coefficients CMA and CM0. (Reals).

Remarks:

1. The set identification number must be unique in respect to all other WTFACT entries. WTFACT set must be selected with Executive Control command WTFACT=WID. Only factors from selected set will be applied to aero elements.

2. Aerodynamic properties for elements that are not listed at WTFACT entry are not changed.

3. Correction factors WLA, WMA, WL0 and WM0 are applied to aerodynamic properties in order to match desired data (experimental, analytical from CFD etc.). Factors will be applied in stationary and non-stationary aerodynamic regime in the form of scaling factors, i.e.:

$$C_{L\alpha}^{W} = W_{L\alpha} C_{L\alpha}\ , \qquad C_{L0}^{W} = W_{L0} C_{L0}$$

$$C_{M\alpha}^{W} = W_{M\alpha} C_{M\alpha}\ , \qquad C_{M0}^{W} = W_{M0} C_{M0}$$

Section 3

Solution sequences and special techniques

3.1 STATIC ANALYSIS

The static analysis (SOL=1) is based on Finite Element Displacement Method. The basic solution is obtained by solving the following system:

$$[K]\{U\} = \{P\} \tag{3.1-1}$$

where K : stiffness matrix in the "skyline" form - upper matrix half under the skyline profile in the form of multiple blocks. The number of blocks depends on the available RAM.

U : displacements (translations and rotations).

P : static load vector.

UNA PROCEDURE:

The following are steps as performed by UNA during the static solution run :

1. Create element stiffness matrices (ROD-BAR).
2. Assemble elements to form global stiffness matrix (ADDSTF).
3. Check for local singularities (FIXOTO).
4. Impose single and multi-point constraints (ADDCON).
5. Decompose the stiffness matrix (DECOMP).
6. Solve the system for multiple load cases and calculate deflections (SOLVER).
7. Multiply original stiffness matrix with deflections in order to recover reactions (REACT).
8. Multiply substructure stiffness with deflections in order to recover free body loads (FBODY).
9. Calculate element internal loads and stresses from nodal relative deflections (STRESS).
10. Post process 1-D and 2-D elements and calculate end loads (POST).
11. Convert output to format suitable for graphic postprocessing.

Free Body Analysis

The free body analysis is a part of static solution run (SOL=1). It provides a simple user interface to the capability of extracting the arbitrary portion of structure, and finding the set of static forces and moments that load that substructure in the self-balanced manner. In that sense, a "*free body*" may be considered as a substructure, and "*free body load*" as a load sub-set compromising from applied load and reactions (if any), and substructure / the rest of structure interface load. Another useful definition for the "*free body load*" is that it represents the load set which will, when applied to substructure, result with the same deformations and internal load as when substructure is a part of the entire structure.

To explain it let the equation (3.1-1) be partitioned so that there are substructure stiffness part and remaining part. Thus 3.1.-1 becomes

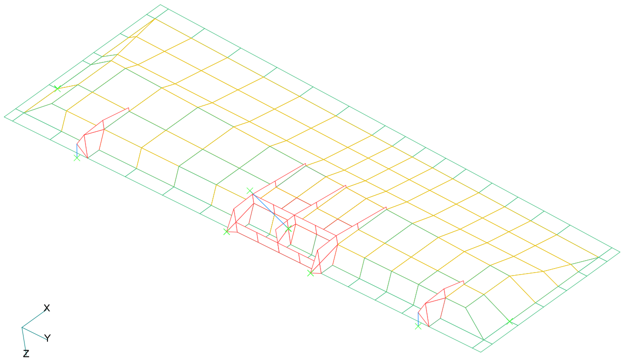
$$\left[K_{fb}\right]\{U\}+\left[K_r\right]\{U\}=\{P\}=\left\{P_{fb}\right\}+\left\{P_r\right\} \qquad (3.1.1\text{-}1)$$

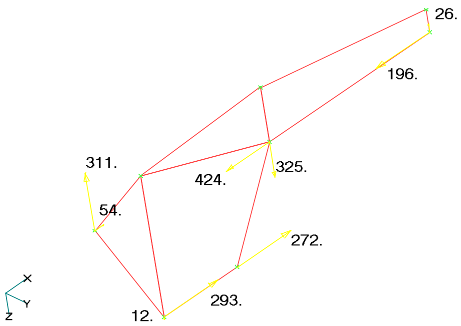
or

$$\left[K_{fb}\right]\{U\}=\left\{P_{fb}\right\} \qquad (3.1.1\text{-}2)$$

where K_{fb} : substructure stiffness matrix.
U : displacements (translations and rotations).
P_{fb} : free body loads.

The free body load is calculated by multiplying the substructure stiffness matrix with deflections. The example of a substructure (spoiler fitting) with a free body loads is shown on the following figure. Fitting plate elements are listed on the FREEBODY Data Entry. Free body load is shown in the form of grid point forces at fitting interface with spoiler and hinge.





The UNA user interface for a free body load analysis is provided in the form FREEBODY Bulk Data Entry. The elements that are listed on that entry create a substructure K_{fb}. A multiple substructures may be defined in the single static run, see FID number definition at FREEBODY Bulk Data Entry.

Free Body Section Cut

By definition sum of forces and moments of the free body P_{fb} load is equal to zero, i.e.

$$\sum P_{fb}^{x} = \sum P_{fb}^{y} = \sum P_{fb}^{z} = 0 \qquad \sum M_{fb}^{x} = \sum M_{fb}^{y} = \sum M_{fb}^{z} = 0 \qquad (3.1.1\text{-}3)$$

for any arbitrary coordinate system. In some cases it is of use to account only a portion of the P_{fb} and to calculate sum of forces and moments based on it. Typically, this is used to calculate internal load which acts from one side of the structure section, such as wing rib station or fuselage frame. The procedure is utilized by listing the substructure nodes of interest on the FREEBODY Bulk Data Entry. Program calculates and prints load and sum of forces and moments for those nodes only. The selection of the coordinate system for output and summation is performed via CID number at FREEBODY Bulk Data Entry.

UNA procedure:

The following are steps required for UNA input file preparation:

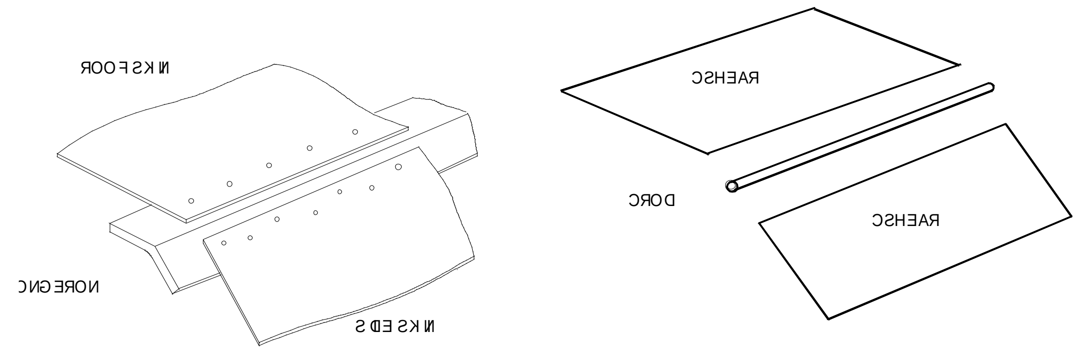
1. List all elements that create free body substructure at FREEBODY Bulk Data Entry.
2. Specify coordinate system CID for outputting the nodal loads and summary values.
3. Select and list substructure nodes for output. Default is all substructure nodes.
4. Repeat steps 1-3 for all desired free-body substructures.
5. Activate output by FREEBODY Executive Control Command.

Post-Processing for 1-D and 2-D Elements

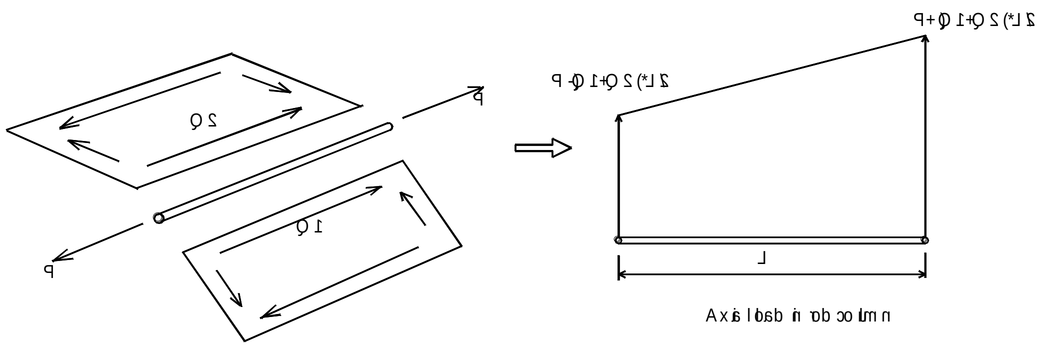
The post-processing analysis is a part of static solution run (SOL=1). It is aimed for one dimensional (1-D) elements, i.e. CROD, CBAR and CBEAM. The basic post-processing represents calculation of axial forces at element ends on the following way :

For all linear elements axial strain is constant along the span, because of UNA utilization of the Finite Element Displacement Method. The result of that constant strain is constant axial force which, in reality, represents the average value between two nodes. In a typical thin-walled aircraft structure idealisation, rods and bars are combined with plane elements such as shear panels, membranes and shells. The design of such structures involves the consideration of internal "*column end loads*" which, effectively, represent rod or bar axial forces corrected for the effect of adjacent shear flows and membrane axial loads.

Typical aircraft idealization is shown on the following figure. Fuselage longeron is idealized with a rod element, and roof and side skins are modelled as a shear panels.



The finite element output is shown on the next figure, axial force for rod and shear flow for panels.



The column end forces are calculated from CROD axial load P and panel shear flows Q1 and Q2 on the following way

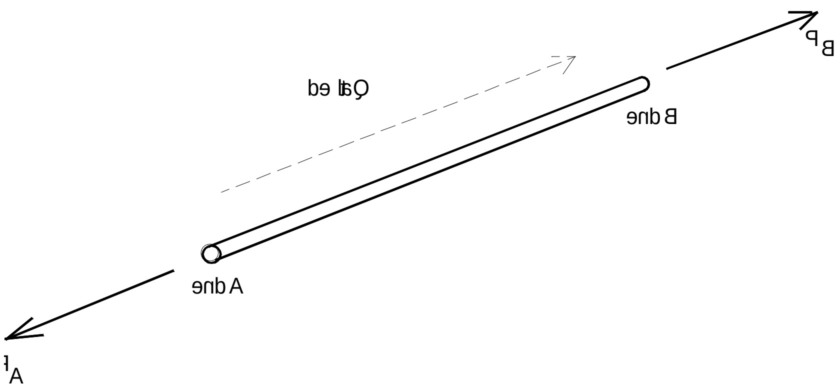
$$P_A = P - (Q_1 + Q_2)\frac{L}{2} \qquad\qquad P_B = P + (Q_1 + Q_2)\frac{L}{2}$$

where P_A : axial load at end A
P_B : axial load at end B
L : rod element lenght

The free body load set is completed by adding the balancing shear flow equal to

$$\Delta Q = \frac{(P_A - P_B)}{L}$$

A convention for P_A, P_B and ΔQ positive directions is shown on the following figure.

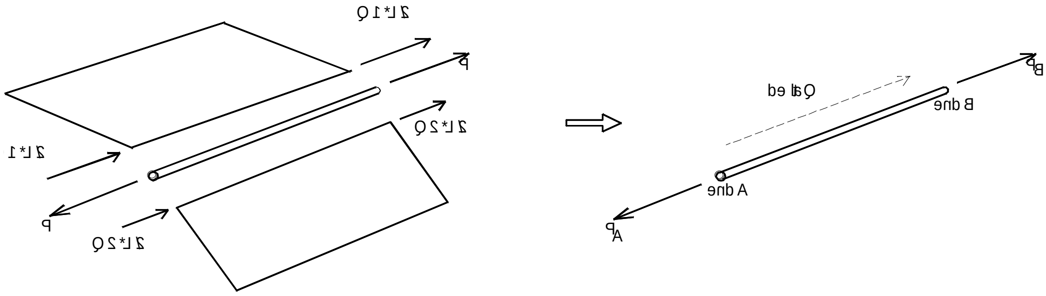


Ends A and B are defined by element convention, i.e. A and B are the first and second node at CROD Bulk Data Entry. The following is example of UNA post-processing output, where |MAX| = max ($|Q_1|$, $|Q_2|$, ... , $|Q_N|$) represents the maximum absolute value of all adjacent shear flows.

ROD ELEMENTS POST-PROCESSING

ELEMENT ID.	LOAD CASE	-AXIAL FORCE- END-A	END-B	-SHEAR FLOW- DELTA	|MAX|
11109106	4510	6332.33	6229.84	0.25	4.04
11109106	5031	-16336.04	-15753.86	-1.42	17.39

The actual UNA postprocessing is based on element grid point force (GPF) balance rather than on shear flow summation. The procedure is that of summing the GPF's along the CROD or CBAR line at both ends, resulting with the same end loads as in the case of shear flow summation.



The difference is made in the case when 2-D elements are CMEMB or CSHELL* elements rather than CSHEAR panels. In that case axial membrane component is collected as well, and end force represents a real column load. User has to be careful in utilizing the output, and keep in mind that end load represents *sum of line element axial and plane element shear and axial (if exists) load components*. Furthermore, any un-regular modelling may affect the post-processing, and erroneous results may occur, such as:

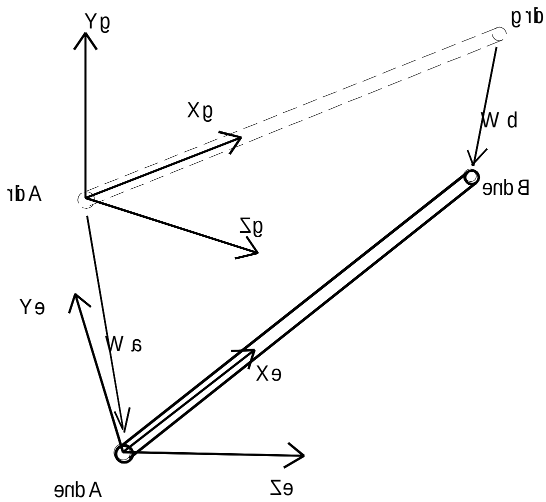
- Two or more line elements share the same line. The first (lowest I.D.) element will collect all 2-D contributions, remaining elements will be left with no change.
- 2-D element spans over two line elements, or line element spans over two 2-D elements along the same edge. Results are erroneous.
- Triangle 2-D element contribution is ambiguous, and this element should be avoided.

CBAR and CBEAM post-processing

Beam element post-processing is very much the same procedure as utilized for rod elements. The summary of the GP Forces is calculated in axial direction (direction between two grid points), by taking into the account contributions from line element and all 2-D elements that share the same edge with CBAR or CBEAM element. Axial end loads and balancing shear flow have the same meaning as already explained for rod post-processing. The only difference is that CBAR end locations may be offset from the grid points.

The basic post-processing and load calculation is performed at grid locations in a "*grid coordinate system*" (*Xg*, *Yg*, *Zg* at following picture), which is defined as follows :

Xg	:	axis vector from grid point A to grid point B.
Xg-Yg	:	plane defined with Xg and element orientation vector V.
Zg	:	axis perpendicular to Xg-Yg plane, right hand system.



From grid locations load is then transformed to element end locations by utilizing the Wa and Wb rigid link transformations. Finally, end load is rotated to "*element coordinate system*" (*Xe, Ye, Ze* at above picture). For conventions defining the orientation vector V, bar element coordinate system, end point offsets and internal loads, see CBAR Bulk Data Entry.

UNA post-processing output is available at both locations :

(G) output : grid point locations in "g*rid coordinate system*" (*Xg, Yg, Zg*).
(E) output : element offset ends in "*element coordinate system*" (*Xe, Ye, Ze*).

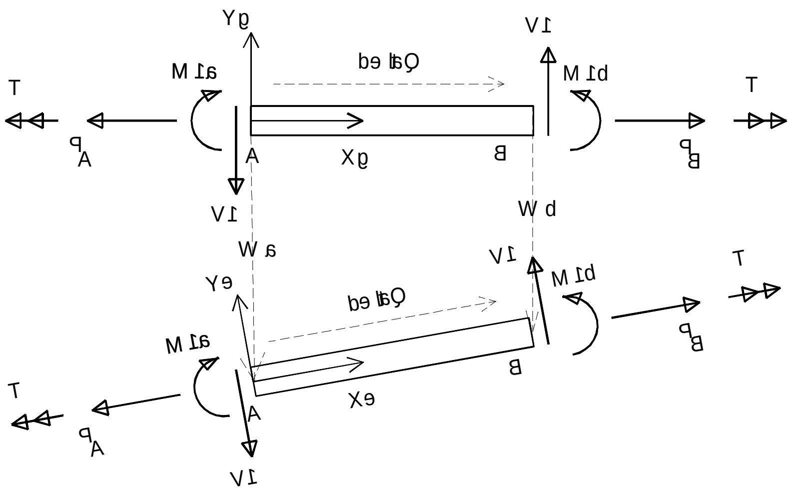
Output location is chosen by Executive Control Command PARAM POST {4} = Num. The following is example of UNA post-processing output in grid system.

grid location
↓

```
BAR ELEMENTS POST-PROCESSING (G)
--------------------------------
```

ELEMENT ID.	LOAD SEARCH	-AXIAL FORCE- END-A	-AXIAL FORCE- END-B	-SHEAR FLOW- DELTA	-SHEAR FLOW- |MAX|
11102081	MAX	82.55	1271.60	-0.99	11.13
11102081	MIN	25.95	386.88	-3.28	3.87

A convention for Plane 1 load positive direction at grid location (grid coordinate system), and offset location (element coordinate system), is shown on the following figure. *<u>Remember that load components do not have the same values at grid and offset locations</u>*, P_A (grid) ≠ P_A (elem) ... etc.



CBAR Element Internal Forces and Moments, (Plane 1)

2-D elements post-processing

UNA provides post-processing output for 2-D elements in the form of shear flows or shear forces at all edges. This option is user controlled by Executive Command PARAM POST {2} = Num. Only four nodes CMEMB, CSHEAR, CQUAD4 and CSHELL4 elements are considered.

Post-processing Max/Min search

The post-processing output is available in two forms, a) All load cases, b) Max/Min search. In the case of default choice UNA prints output for all analysed load cases. Max/Min search is required by Executive Control Command PARAM POST {1} = Num. If required, UNA search and prints Maximum and Minimum values for each load component, together with i.d. number of the critical load case.

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. In most of the cases only a few are required. The following commands are static solutions related:

1. DISP : Displacement output.
2. FORCE : Element internal load and stress output.
3. FREEBODY: Free body forces output.
4. GPFORCE : Element grid point forces output.
5. PARAM : Element output control.
6. POST : Request post-processing for 1-D and 2-D elements.
7. REACT : SPC reactions output.

TIPS:

- Read Executive Control Section of the manual which describes PARAM commands. They provide a great variety of options for controlling FE output such as stresses, locations etc.
- Check sum of forces and moments for reactions and compare with input load. If difference is found, it usually points to modelling errors (load exits, mechanism etc.).
- CTRIA3, CQUAD4, CSHELL3 and CSHELL4 do not have "drilling" rotations defined (perpendicular to surface). Use FIXOTO = ROT command to eliminate singularities.
- Eliminate "incompatible" modes for CSOLID elements when modelling sandwich core. Use PARAM SOLID 4 = 2 command to achieve this.

Executive Control Section Example

```
$
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "STATIC ANALYSIS EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 1                  ! select static solution
SPC = 10                 ! select SPC and SPC1 Bulk Data constraint sets
MPC = 100                ! select MPC and COUPG Bulk Data sets
VER = 7.2                ! specify Bulk Data file version
$
FIXOTO = ROT             ! check rotational stiffness at nodes
$
SUBCASE =  4510          ! select static subcase set
SUBCASE =  5031          ! select static subcase set
SUBCASE = 13023          ! select static subcase set
$
$
$ OUTPUT REQUEST
$ --------------
$
DISP TRA = SET 12      ! print deflections (translations) from SET 12
REACT    = ALL         ! print all reactions (forces and moments)
GPFOR    = SET 14      ! print grid point forces, elements from SET 14
FORCE    = SET 14      ! print element internal load and stresses, SET 14
FREEBODY = ALL         ! print forces and moments for all free bodies
POST     = SET 14      ! print post-processed internal load, SET 14
$
$
$ ALTER OUTPUT DEFAULTS
$ ---------------------
$
PARAM BAR    {1} = 2      ! BAR internal load output in X-Y plane
PARAM BAR    {2} = 1      ! BAR internal load output at grid locations
PARAM PLATE  {2} = 1      ! Plate stresses at neutral plane required
PARAM PLATE  {5} = 2      ! Plate incompatible modes supressed
PARAM POST   {1} = 3      ! POST-processing Max/Min search required
$
$
$ FEMAP OUTPUT REQUEST
$ --------------------
$
FEMAP DISP   = ON         ! create FEMAP file with deflections
FEMAP FORCE  = ON SET 14  ! create FEMAP file with internal loads
FEMAP STRESS = ON COMP 17 ! create FEMAP file with stresses, 17 vectors
$
BEGIN BULK                ! end of Executive Control section
```

Bulk Data Section Example

```
$
$
$  BULK DATA SECTION
$
$
$ SET 12 : NODES FOR DISPLACEMENT OUTPUT
$ SET 14 : ELEMENTS FOR STRESS OUTPUT AND FEMAP INTERNAL LOADS
$ ------------------------------------------------------------
$
SET           12  110381  110395  150251  150265
+                 110201  110215  110251  110265
+                 110301  110315  110351  110365
+                 110401  110415  110451  110465
$
SET           14        11109102  THRU  11109114
+                       11109152  THRU  11109164
+                       11109201        11109215
+                       11109251        11109265
+                       15109101  THRU  15109114
+                       15109151  THRU  15109164
$
$
$ FREE BODY GROUP
$ ---------------
$
FREEBODY    1100
+           ELEM        12110381        22110381       ! list of elements
+                       12110395        22110395
+                       12150251        22150251
+                       12150265        22150265
+                       14102001  THRU  14102014
+                       24102001  THRU  24102014
+                       13152251        23152251
+                       13152265        23152265
+
+           NODE  110381  110395  150251  150265       ! list of nodes
+                 110281    THRU  110295
+                 210381  210395  250251  250265
+                 210281    THRU  210295
$
$
$ INCLUDE FILE WITH FE MODEL
$ --------------------------
$
INCLUDE     "WING_S44.UNA"
$
ENDDATA
```

Output Example

NODAL TRANSLATIONS - L.case : 5031

Node	Dsys	U1	U2	U3
110201	0	-2.787	-1.940	3.563
110215	0	-2.634	-2.340	-1.435
110251	0	-0.722	1.752	3.480

NODAL REACTIVE FORCES - L.case : 5031

Node	Dsys	R1	R2	R3	Rtot
110281	0	-279.69	-6.82	-3.13	279.79
110282	0	-272.73	-5.92	2.24	272.80
110283	0	-160.75	-2.80	1.10	160.77

F R E E B O D Y F O R C E S

FREE BODY I.D. : 1100
LOAD CASE : 5031

Node	Dsys	P1	P2	P3	Ptot
110381	0	0.00	0.00	-29157.11	29157.11
110395	0	0.00	0.00	-7612.70	7612.70
150251	0	0.00	267.37	-446.04	520.03
150265	0	0.00	-4207.09	5319.65	6782.21

ELEMENT GRID POINT FORCES - L.case : 5031 (continue)

Elem	Node	Dsys	P1	P2	P3	Ptot
11109106	110906	0	0.00	16044.95	0.00	16044.95
	111006	0	0.00	-16044.95	0.00	16044.95
11109107	110907	0	0.00	17265.41	0.00	17265.41
	111007	0	0.00	-17265.41	0.00	17265.41
11109108	110908	0	0.00	26116.05	0.00	26116.05
	111008	0	0.00	-26116.05	0.00	26116.05

ROD ELEMENTS - L.case : 5031

Elem	Force	Torque	Sig	Tau
11109106	-16044.95	0.00	-31.09	0.00
11109107	-17265.41	0.00	-32.03	0.00
11109108	-26116.05	0.00	-32.36	0.00

SHEAR ELEMENTS - L.case : 5031

Elem	Taxy	Q-21	Q-23	Q-43	Q-41
15109105	-5.44	-17.39	-17.39	-17.39	-17.39
15109106	-4.99	-15.97	-15.97	-15.97	-15.97
15109107	-3.81	-16.02	-16.02	-16.02	-16.02
15109108	-1.14	-4.78	-4.78	-4.78	-4.78

ROD ELEMENTS POST-PROCESSING

ELEMENT	LOAD	-AXIAL FORCE-		-SHEAR FLOW-	
ID.	SEARCH	END-A	END-B	DELTA	|MAX|
11109106	MAX	6332.33	6229.84	0.25	17.39
	l/c	4510	4510	4510	5031
11109106	MIN	-16336.04	-15753.86	-1.42	4.04
	l/c	5031	5031	5031	4510
11109107	MAX	6688.97	6534.31	0.38	16.02
	l/c	4510	4510	4510	5031
11109107	MIN	-17255.06	-17275.75	0.05	4.42
	l/c	5031	5031	5031	4510
11109108	MAX	10883.21	9218.80	4.06	16.02
	l/c	4510	4510	4510	5031
11109108	MIN	-28419.89	-23812.21	-11.24	5.76
	l/c	5031	5031	5031	13023

3.2 MODAL ANALYSIS

The modal analysis solution (SOL=2) calculates natural frequencies and modal shapes. The basic free vibration solution is achieved by utilizing software implementation of the LANCZOS algorithm, which is probably the best available for FE applications. Eigen-value module is capable to manage free body modes by so called *spectral shift* approach. The following system is solved

$$\left([K]-\omega^2[M]\right)\cdot\{\phi\}=0 \qquad (3.2\text{-}1)$$

resulting with free body and flexible modes orthonormalized on the following way

$$\left[M_{HH}\right]=[\phi]^T[M][\phi]=[1] \qquad \left[K_{HH}\right]=[\phi]^T[K][\phi]=\left[\omega^2\right] \qquad (3.3\text{-}2)$$

Modal mass matrix is unit matrix, and modal stiffness matrix is diagonal with frequency [rad/sec] squared on it.

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. The following are modal related commands:

1. FZERO : Specify tolerance for rigid body modes in [Hz].
2. GRAV : Scale mass matrix in order to produce compatible system of units.
3. LAMBDA : Specify desired number of modes. (Default = 5)
4. MASTYP : Select mass matrix type, consistent (recommended) or lumped.
5. MGROUP : Selects a group of elements to be included in mass matrix. (Default is all).
6. MODES : Request modal displacement printout.
7. SHIFT : Stiffness matrix shift in [Hz]. Negative shift is used to extract rigid body modes.

TIPS:

- Free vibration frequencies are in units of cycles per second [Hz].
- MGROUP Executive Control command is used to control vibration area of interest. Only elements selected by this command will be used to form mass matrix. ***This allows free vibration analysis of the isolated part of the large structure.*** The non-selected elements form elastic support for the analysed part (elastic boundary conditions).

Executive Control Section Example

```
$
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "MODAL ANALYSIS EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 2                    ! select modal solution
SPC = 20                   ! select SPC and SPC1 Bulk Data constraint sets
VER = 7.2                  ! specify Bulk Data file version
$
LAMBDA = 12                ! specify number of modes
SHIFT  = -1.               ! shift stiffness matrix by -1.0 [Hz]
FZERO  = 0.01              ! tolerance for rigid body modes in [Hz]
GRAV   = 386.4             ! mass matrix scaling in Imperial System
$
$
$ OUTPUT REQUEST
$ --------------
$
FORMAT = 1                 ! print output using E (exponential) format
PAGES  = 0                 ! no output paging
ECHO   = ON                ! print input data echo
MODES  = RANGE 1 6         ! print modal deflections for modes 1 thru 6
$
$
$ FEMAP OUTPUT REQUEST
$ --------------------
$
FEMAP MODES  = ON          ! create FEMAP file with modal deflections
$
$
BEGIN BULK                 ! end of Executive Control section
$
```

Output Example

FREQUENCIES

MODE	FREQUENCY (in Hz)	MODAL MASS	ERROR (%) (estimate)	NODE	COMP
1	0.0	1.00000E+00	7.82208E-17	3116	X2-tra
2	0.0	1.00000E+00	1.30437E-16	1001	X2-tra
3	0.0	1.00000E+00	7.30395E-18	2124	X3-tra
4	2.58033E+00	1.00000E+00	2.31845E-15	3116	X2-tra
5	3.40938E+00	1.00000E+00	4.68039E-15	3116	X3-tra
6	3.89320E+00	1.00000E+00	7.39316E-13	2124	X3-tra
7	4.84876E+00	1.00000E+00	1.78808E-09	3116	X1-tra
8	5.22337E+00	1.00000E+00	4.14786E-09	3116	X1-tra
9	6.66220E+00	1.00000E+00	1.20689E-06	3116	X3-tra
10	7.86067E+00	1.00000E+00	8.33142E-05	3116	X3-tra
11	9.63533E+00	1.00000E+00	1.36495E-01	3116	X3-tra
12	9.85148E+00	1.00000E+00	3.31143E-01	3116	X1-tra

MODE : 1 Freq = 0.0000E+00 Mod.mas = 1.0000E+00

Node	Dsys	U1	U2	U3	Fi1	Fi2	Fi3
1001	0	0.000E+00	3.236E+00	0.000E+00	3.342E-03	0.000E+00	-3.409E-03
1002	0	0.000E+00	2.946E+00	0.000E+00	3.342E-03	0.000E+00	-3.409E-03
1003	0	0.000E+00	2.813E+00	0.000E+00	3.342E-03	0.000E+00	-3.409E-03

MODE : 4 Freq = 2.5803E+00 Mod.mas = 1.0000E+00

Node	Dsys	U1	U2	U3	Fi1	Fi2	Fi3
1001	0	0.000E+00	2.339E-01	0.000E+00	3.295E-03	0.000E+00	4.115E-04
1002	0	0.000E+00	2.688E-01	0.000E+00	3.295E-03	0.000E+00	4.115E-04
1003	0	0.000E+00	2.848E-01	0.000E+00	3.295E-03	0.000E+00	4.116E-04

The elastic buckling solution (SOL=3) calculates critical buckling loads and buckling shapes. The analysis represents a linearized buckling solution, in which the following problem is solved

$$\det\left([K]+\lambda\cdot\left[K_\sigma\right]\right)=0 \qquad \rightarrow \qquad \left([K]+\lambda\cdot\left[K_\sigma\right]\right)\cdot\{\phi\}=0 \tag{3.3-1}$$

K and K_σ are, respectively, linear and nonlinear strain (geometric) stiffness matrices that usually correspond to the initial configuration of the structure. The eigenvalue problem is solved by LANCZOS algorithm. The buckling load is obtained as

$$R_{buckling}=\lambda_1\cdot R \tag{3.3-2}$$

where λ_1 is lowest eigenvalue found and R original load vector applied. The linearized buckling analysis is appropriately used to predict the load level at which structure becomes unstable if the pre-buckling displacements and their effects are negligible.

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. The following are buckling related commands:

1. LAMBDA : Specify desired number of eigenvalues. (Default = 5).
2. MGROUP : Selects elements to be included in geometric stiffness matrix. (Default is all).
3. MODES : Request buckling displacement printout.
4. SHIFT : Request stiffness matrix shift. Used to bypass lower eigenvalues.

TIPS:

- Only CROD, CBEAM, CBAR, CSHELL3 and CSHELL4 are used to form geometric stiffness matrix and calculate buckling load. Other types may be in the Bulk Data Deck, but they provide only linear stiffness matrix (elastic support for buckling area).
- MGROUP Executive Control command is used to control buckling area of interest. Only elements selected by this command will be used to form geometric stiffness matrix. *This allows buckling analysis of the isolated part of the large structure.* The non-selected elements form elastic support for the analysed part (elastic boundary conditions).

Executive Control Section Example

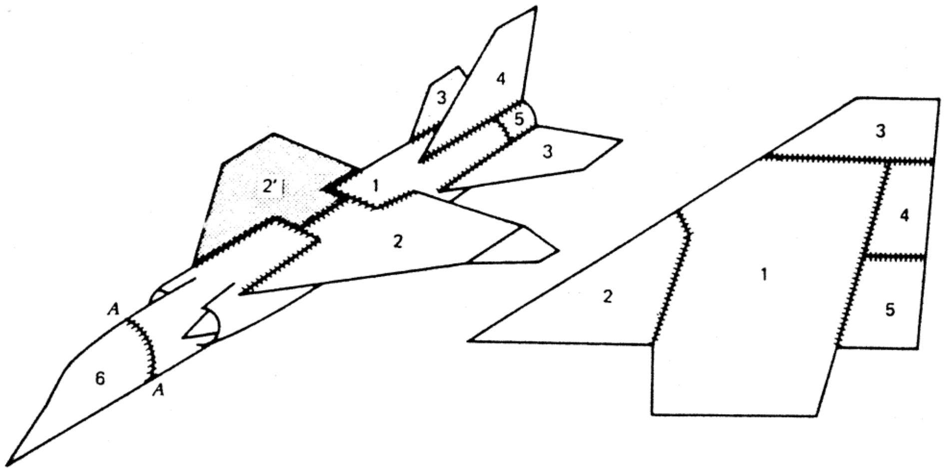
```
$
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "ELASTIC BUCKLING EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 3                    ! select elastic buckling solution
SPC = 30                   ! select SPC and SPC1 Bulk Data constraint sets
VER = 7.2                  ! specify Bulk Data file version
$
LAMBDA = 3                 ! specify number of eigenvalues
MGROUP = ALL               ! selects elements for buckling (all)
$
SUBCASE = 1                ! select static subcase set
SUBCASE = 2                ! select static subcase set
SUBCASE = 3                ! select static subcase set
$
$
$ OUTPUT REQUEST
$ --------------
$
MODES  = ALL               ! print all buckling shapes
$
$
$ PATRAN OUTPUT REQUEST
$ ---------------------
$
PATRAN MODES  = ON         ! create PATRAN files with buckling shapes
$
$
BEGIN BULK                 ! end of Executive Control section
$
```

Output Example

```
          CRITICAL LOAD MULTIPLIERS - L.case :       1
          --------------------------------------------

       MODE         LAMBDA      ERROR (%)     NODE    COMP

          1    2.41927E+00    1.14791E-02       11  X2-tra
          2    6.40954E+00    5.30155E+00        7  X2-tra
          3    1.16664E+01    3.18205E+01        6  X2-tra



          CRITICAL LOAD MULTIPLIERS - L.case :       2
          --------------------------------------------

       MODE         LAMBDA      ERROR (%)     NODE    COMP

          1    4.48655E-01    4.48517E-02       11  X2-tra
          2    9.01878E-01    3.42738E+00        6  X2-tra
          3    1.63204E+00    2.55495E+01        3  X2-tra



          CRITICAL LOAD MULTIPLIERS - L.case :       3
          --------------------------------------------

       MODE         LAMBDA      ERROR (%)     NODE    COMP

          1    4.05153E+00    2.06732E-02       11  X2-tra
          2    9.49229E+00    4.38036E+00        7  X2-tra
          3    1.80275E+01    3.23495E+01        5  X2-tra


      BUCKLING MODE :   1   L.case :       1   Lambda = 2.4193E+00
      ------------------------------------------------------------

Node    Dsys        U1        U2        U3       Fi1       Fi2       Fi3

   1       0     0.000     0.000     0.000     0.000     0.000    -0.058
   2       0     0.000    -0.058     0.000     0.008     0.000    -0.058
   3       0     0.000    -0.117     0.000     0.017     0.000    -0.058
```

3.4 SUBSTRUCTURING

The substructuring solution (SOL=4) generates "superelement" or "macro element" with many nodes on its boundary and many interior d.o.f. The process represents stiffness matrix and load vector condensation. Mathematically, a substructure is a partially solved portion of the complete set of structural equations. Physically, a substructure is one of two or more parts into which a structure or a finite element mesh is divided. Multilevel substructuring is possible.



UNA PROCEDURE:

Elements on the above figure are substructures or "superelements" having a few d.o.f. After the division of a structure into substructures has been selected, static analysis proceeds as follows:

1. Evaluate [K] and {r} for each substructure, where [K] and {r} are stiffness matrix and load vector based on substructure d.o.f. In the UNA run that means creating a separate Bulk Data file for each substructure (file contains substructure elements, interior and boundary nodes). Boundary nodes are identified by ISUP field at GRID Bulk Data Entry.

2. Eliminate internal d.o.f. by condensation. This is done by UNA solution SOL=4 applied to every substructure, one by one. The condensed substructure became a "superelement" with stiffness and load defined at boundary nodes only. The condensation is performed by starting Gauss elimination solution of equations for unknowns but stopping before the substructure stiffness matrix has been fully reduced. This procedure is called *static condensation*.

To explain it let the equations $[K]\{U\}=\{P\}$ represents a substructure which is portion of entire structure. Let d.o.f. $\{U\}$ be partitioned so that $\{U\}=\begin{bmatrix}U_r & U_c\end{bmatrix}^T$ where $\{U_r\}$ are boundary d.o.f. to be retained and $\{U_c\}$ are internal d.o.f. to be eliminated by condensation. Thus $[K]\{U\}=\{P\}$ becomes

$$\begin{bmatrix}K_{rr} & K_{rc}\\ K_{cr} & K_{cc}\end{bmatrix}\begin{Bmatrix}U_r\\ U_c\end{Bmatrix}=\begin{Bmatrix}P_r\\ P_c\end{Bmatrix} \qquad (3.4\text{-}1)$$

The lower partition is solved for $\{U_c\}$

$$\{U_c\}=-\left[K_{cc}\right]^{-1}\left(\left[K_{cr}\right]\{U_r\}-\{P_c\}\right) \qquad (3.4\text{-}2)$$

Next $\{U_c\}$ is substituted into the upper partition of above matrix equation. Thus

$$\left(\left[K_{rr}\right]-\left[K_{rc}\right]\left[K_{cc}\right]^{-1}\left[K_{cr}\right]\right)\{U_r\}=\{P_r\}-\left[K_{rc}\right]\left[K_{cc}\right]^{-1}\{P_c\}$$

or

$$\left[K_{rr}^{s}\right]\{U_r\}=\left\{P_r^{s}\right\} \qquad (3.4\text{-}3)$$

where $\left[K_{rr}^{s}\right]$ and $\left\{P_r^{s}\right\}$ are condensed stiffness matrix and load vector, respectively. UNA run SOL=4 finishes here. *The condensed matrix and load vector became a "superelement" stiffness and load.* The data are written into external file and saved on hard drive. The name of the file is defined by SUPNAME Executive Control Command. This step is repeated for every substructure.

3. From this point superelement is treated in the standard fashion. It is attached to the structure via CSUPEL Bulk Data Entry, and used as ordinary element in any available UNA solution sequence (SOL=1,2,..., 9). Stiffness matrix is assembled into the structure, boundary conditions applied and system of equation solved.

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. There is only one exclusively related to superelements:

1. SUPNAME : Specify name for file created during SOL=4. This file contains reduced stiffness matrix and load vector (default = SUPELEM.BIN).

BULK DATA:

Used to define finite element model. There is only one exclusively related to superelements:

1. CSUPEL : Defines a Superelement.

TIPS:

- There are separate Bulk Data Files for each substructure plus main structure. Interface (boundary) nodes have to be repeated in both files. The node numbers may be different between files, but physical location in the space must be the same.
- Sort substructure grid points so that interface nodes are at the end of system of equations. It reduces band and increase decomposition speed during SOL=4 condensation.
- Static condensation SOL=4 does not reduce mass matrix. Superelement may be used in dynamic solutions, but only substructure stiffness will be included, mass will not.
- Substructure and main structure may be defined in different coordinate systems. In order to correlate two structures use CID system identification as defined at CSUPEL Bulk Data Entry.
- UNA calculates and prints superelement interface load during main structure static run. To control this output see PARAM SUPEL Executive Data Command.

The transient response solution (SOL=5) is a dynamic analysis with has inertia and elastic forces defined as a time dependent variables. The following is form of equations of equilibrium governing linear, non-damped dynamic response of system of finite elements

$$[M]\{X''\}+[K]\{X\}=\{P(t)\} \tag{3.5-1}$$

where M and K are mass and stiffness matrices; P(t) is the time dependent external load vector; and X and X" are displacement and acceleration. This system is based on Newton law of motion. Mathematically, system represents a system of linear differential equations of second order and can be solved by standard procedure for solution of differential equations with constant coefficients. However, the procedure for solving of general system of differential equations became very expensive in the case of large number of unknowns such as in finite element models. The procedure to reduce system and solve it in reduced form is called *Mode Superposition Method*, and it is utilized in UNA transient response analysis.

Method reduces system to generalized form in the few steps. First, a free vibration solution is sought by solving the Eigen-value problem of the following type

$$\left([K]-\omega^2[M]\right)\cdot\{\phi\}=0 \tag{3.5-2}$$

Stiffness and mass matrix size is N x N. Solution of the system produces natural frequencies and free-vibration modes. Usually, the limited number of frequencies and modes is calculated, i.e. total of H modes. This number may be significantly smaller than full system size N.

Second, a *modal transformation matrix* is created by taking modal vectors and storing them column-wise. The free body and flexible modes are orthonormalized, and by using potential and kinetic energy of dynamic system, it can be shown that following transformations are valid

$$[M_{HH}]=[\phi]^T[M][\phi]=[1] \qquad [K_{HH}]=[\phi]^T[K][\phi]=[\omega^2] \tag{3.5-3}$$

Matrices M_{HH} and K_{HH} are diagonal matrices with dimension H x H. Modal mass matrix represents unit diagonal matrix, and modal stiffness matrix is diagonal with frequency squared on it. Modal transformation matrix ϕ can be used to change from finite element coordinate basis to basis of eigenvectors of the generalized problem, i.e.

$$\{X\}=[\phi]\{X_H\} \tag{3.5-4}$$

where X is vector of finite element nodal displacements and X_H generalized modal displacements.

If the same transformation is applied to equations of equilibrium governing the dynamic response of system of finite elements (3.5-1), the system is transformed to modal generalized form, i.e.

$$[M]_{HH}\{X''_H\}+[K_{HH}]\{X_H\}=\{P_H(t)\} \tag{3.5-5}$$

or by taking into account equation (3.5-3), system reduces to H individual equations of the form

$$x_h''(t)+\omega_h^2 x_h(t)=p_h(t) \qquad h=1,2,\ldots,H \tag{3.5-6}$$

where $p_h(t)=\{\phi_h\}^T P(t)$. The initial conditions on the motions of this system are obtained by

$$\begin{aligned} x_h(0)&=\{\phi_h\}^T[M]\{X(0)\} \\ x_h'(0)&=\{\phi_h\}^T[M]\{X'(0)\} \end{aligned} \tag{3.5-7}$$

The solution to each equation in (3.5-6) can be obtained by using some of numerical integration algorithms, or can be calculated by Duhamel integral

$$x_h(t)=\frac{1}{\omega_h}\int_0^t p_h(\tau)\sin\omega_h(t-\tau)d\tau+\alpha_h\sin\omega_h t+\beta_h\sin\omega_h t \tag{3.5-8}$$

where α_h and β_h are determined from the initial conditions (3.5-7). For the complete response, the solution to all H equations in (3.5-6) must be calculated, and then the finite element nodal displacement are calculated by superposition of the response of the each mode; i.e. by using (3.5-4), we get

$$\{X(t)\}=\sum_{h=1}^{H}\{\phi_h\}x_h(t) \tag{3.5-9}$$

The internal load and stresses are calculated by utilizing standard finite element approach, i.e. from relative nodal deflections.

UNA PROCEDURE :

The transient response analysis is based on mode superposition method. The following are steps as performed by program:

1. Solve for the eigenvalues and eigenvectors of the problem by LANCZOS algorithm. All explanations made for modal analysis (section 3.2) are valid for this step as well.

2. Transform initial conditions to modal base (3.5-7).

3. For each time step performs the following:
   a) transform load vector to modal basis (3.5-6).
   b) solve decoupled equilibrium equations in (3.5-6) by using Duhamel integral (3.5-8).
   c) backward superposition of the response in each eigenvector (3.5-9). Calculate internal load and stresses from nodal deflections.

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. The following are modal and transient response (highlighted) related commands:

1. FZERO : Specify tolerance for rigid body modes in [Hz].

2. GRAV : Scale mass matrix in order to produce compatible system of units.

3. INCR : Specify number of time increments.

4. LAMBDA : Specify desired number of modes. (Default = 5)

5. MASTYP : Select mass matrix type, consistent (recommended) or lumped.

6. MGROUP : Selects a group of elements to be included in mass matrix. (Default is all).

7. MODES : Request modal displacement printout.

8. RESPON : Specify time span.

9. SHIFT : Stiffness matrix shift in [Hz]. Negative shift is used to extract rigid modes.

BULK DATA:

Used to define finite element model. The following are exclusively transient response entries:

1. <u>DFORCE</u> : Dynamic concentrated force at grid point.
2. <u>DMOMENT</u> : Dynamic concentrated moment at grid point.
3. <u>DPRESS</u> : Dynamic pressure.
4. <u>IDISP</u> : Initial nodal displacements.
5. <u>IVELO</u> : Initial nodal velocities.

TIPS:

- Dynamic load is applied by DFORCE, DMOMENT and DPRESS cards. The load-time function can be selected from 18 pre-defined shapes, or can be created via TABLE Bulk Data entry. Multiple load cards with a different starting and ending times can be applied in the single run.
- Use GRAV Executive Control Command in order to match compatible system of units. Failure to achieve this leads to erroneous natural frequencies and structural response. Calculated free vibration frequencies are in units of cycles per second [Hz].
- Experiment with a different number of calculated modes (LAMBDA), time interval (RESPONSE), and number of load steps (INCREMENTS). Generally speaking, for a short time impacts a larger number of modes and shorter time steps are required in order to represents structural response properly.
- MGROUP Executive Control command is used to control vibration area of interest. Only elements selected by this command will be used to form mass matrix. <u>*This allows free vibration analysis of the isolated part of the large structure.*</u> The non-selected elements form elastic support for the analysed part (elastic boundary conditions).

Executive Control Section Example

```
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "TRANSIENT RESPONSE EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 5                    ! select transient response solution
VER = 7.2                  ! specify Bulk Data file version
$
LAMBDA   = 2               ! specify number of modes
RESPONSE = 0.0  3.36       ! specify time values for start and end
INCREM   = 12              ! specify number of time steps
$
$
$ OUTPUT REQUEST
$ --------------
$
DISP TRA = ALL             ! print deflections, translations only
FORCE    = ALL             ! print element internal loads
$
$
$ FEMAP OUTPUT REQUEST
$ --------------------
$
FEMAP MODES  = ON          ! create FEMAP file with modal shapes
FEMAP DISP   = ON          ! create FEMAP file with nodal deflections
$
BEGIN BULK                 ! end of Executive Control section
```

Bulk Data Section Example

```
GRID           3       0     0.0     0.0     0.0       0  123456
GRID           1       0     1.0     0.0     0.0       0   23456
GRID           2       0     2.0     0.0     0.0       0   23456
GRID           4       0     3.0     0.0     0.0       0  123456
$
CELAS2         1      4.       3       1       1       1
CELAS2         2      2.       1       1       2       1
CELAS2         3      2.       2       1       4       1
$
CMASS          4       1       0      2.
CMASS          5       2       0      1.
$
DFORCE         1       2       0       1     10.      0.  ! dynamic force
$
ENDDATA
```

Output Example

```
                          FREQUENCIES
                          -----------

  MODE      FREQUENCY     MODAL MASS      ERROR (%)      NODE    COMP

     1    2.25079E-01    1.00000E+00    2.74692E-32         2  X1-tra
     2    3.55881E-01    1.00000E+00    3.08072E-30         2  X1-tra


               NODAL TRANSLATIONS - Time : 8.40000E-01
               ---------------------------------------

        Node    Dsys           U1           U2           U3

           1       0        0.176        0.000        0.000
           2       0        2.781        0.000        0.000
           3       0        0.000        0.000        0.000
           4       0        0.000        0.000        0.000

               NODAL TRANSLATIONS - Time : 1.12000E+00
               ---------------------------------------

        Node    Dsys           U1           U2           U3

           1       0        0.486        0.000        0.000
           2       0        4.094        0.000        0.000
           3       0        0.000        0.000        0.000
           4       0        0.000        0.000        0.000


                  ELAS2 ELEMENTS - Time : 8.40000E-01
                  -----------------------------------

     Elem    Dof         Load            Elem    Dof         Load

        1      1        -0.70               2      1        -5.21
        3      1         5.56

                  ELAS2 ELEMENTS - Time : 1.12000E+00
                  -----------------------------------

     Elem    Dof         Load            Elem    Dof         Load

        1      1        -1.94               2      1        -7.22
        3      1         8.19
```

The frequency response solution (SOL=6) is a dynamic analysis with has inertia, damping and elastic forces defined as a time dependent variables. It is assumed that all transient effects have died out after some time, and that harmonic excitation is steady or independent of time. The solution of dynamic system is sought in the *frequency domain* rather than in the *time domain.* The goal of analysis is to determine the way how some physical entities, i.e. structural internal load, acceleration etc. inside the dynamic system, respond to harmonic excitation. Excitations may be of various nature (force, acceleration), but all of them can be defined as a harmonic functions of the following type

$$p(t) = p_o \cdot e^{i\Omega(t+\tau)} = p_o \cdot e^{i\Omega\tau} \cdot e^{i\Omega t} = \bar{p}_o \cdot e^{i\Omega t} \qquad (3.6\text{-}1)$$

The dynamic system of equations is based on the Newton law of motion. By adding viscous damping and harmonic forcing function to it, the single degree of freedom system can written as

$$mx'' + cx' + kx = \bar{p}_o \cdot e^{i\Omega t} \qquad (3.6\text{-}2)$$

where m : mass
c : viscous damping
k : stiffness
$\bar{p}_o$: load complex amplitude
Ω : excitation frequency

The solution has two parts, transient and steady state. Transient part is not of the interest in this case, and after the transient motion are dampened out system will settles down to the steady response state. If the solution is search in the form of

$$x(t) = \bar{x} \cdot e^{i\Omega t} \qquad (3.6\text{-}3)$$

and substituted to the previous equation, the system can be written as

$$\left(-\Omega^2 m + i\Omega c + k\right)\bar{x} \cdot e^{i\Omega t} = \bar{p}_o \cdot e^{i\Omega t} \qquad (3.6\text{-}4)$$

or, by dividing both sides by $e^{i\Omega t}$, a purely algebraic system of equation is obtained

$$\left(-\Omega^2 m + i\Omega c + k\right)\bar{x} = \bar{p}_o \qquad (3.6\text{-}5)$$

The solution is in the complex form, i.e. there is real and imaginary part. In the case of single degree of freedom the solution is

$$x(t) = \frac{\bar{p}_o}{-\Omega^2 m + i\Omega c + k} \cdot e^{i\Omega t} = (x_R + ix_I) \cdot e^{i\Omega t} = |\bar{x}| \cdot e^{i\Omega(t+\varphi)} \qquad (3.6\text{-}6)$$

where x_R : real part

x_I : imaginary part

$|\bar{x}| = \sqrt{x_R^2 + x_I^2}$: amplitude

$\varphi = \tan^{-1}(x_I / x_R)$: phase

Transfer function

The transfer function, or complex frequency response, is defined as a complex ratio between physical entity response and input excitation, i.e.

$$\bar{H}(\Omega) = \frac{\bar{x}(\Omega)}{\bar{p}_o(\Omega)} \qquad (3.6\text{-}7)$$

Such a function can be calculated by harmonic analysis approach for any particular load or for set of different loads applied on the structure. In the case of single degree of freedom transfer function is defined as follows

$$\bar{H}(\Omega) = \frac{\bar{p}_o}{-\Omega^2 m + i\Omega c + k} = H_R(\Omega) + iH_I(\Omega) \qquad (3.6\text{-}8)$$

and its amplitude is given as

$$\left|\bar{H}(\Omega)\right| = \sqrt{H_R^2(\Omega) + H_I^2(\Omega)} \qquad (3.6\text{-}9)$$

Transfer function plays a very important role in defining the random vibration response or aerodynamic gust analysis.

Generalized problem

The algebraic system of equation (3.6-5) can be expanded in order to include structural damping as well. The following is form of equations governing damped linear system response of system of finite elements

$$\left[-\Omega^2 M + i\Omega C + \left(1+ig\right)K\right] \cdot \overline{X}(\Omega) = \overline{P}(\Omega) \tag{3.6-10}$$

where M, C and K are mass, viscous damping and stiffness matrices; g is complex stiffness damping factor, $P(\Omega)$ and $X(\Omega)$ are complex load and solution vectors. Mathematically, system represents a system of linear equations with complex coefficients. It can be solved by any standard linear matrix solution method like Gaussian elimination. However, the procedure for solving it became very expensive in the case of large number of various exciting frequencies Ω such as in frequency response analysis. The procedure to reduce system and solve it in reduced form is called *Mode Superposition Method*, and it is utilized in UNA frequency response analysis.

Method reduces system to generalized form in the few steps. First, a free vibration solution is sought by solving the Eigen-value problem of the following type

$$\left([K]-\omega^2[M]\right)\{\phi\} = 0 \tag{3.6-11}$$

Stiffness and mass matrix size is N x N. Solution of the system produces natural frequencies and free-vibration modes. Usually, the limited number of frequencies and modes is calculated, i.e. total of H modes. This number may be significantly smaller than full system size N.

Second, a *modal transformation matrix* is created by taking modal vectors and storing them column-wise. The free body and flexible modes are orthonormalized, and by using potential and kinetic energy of dynamic system, it can be shown that following transformations are valid

$$\left[M_{HH}\right] = [\phi]^T[M][\phi] = [1] \qquad \left[K_{HH}\right] = [\phi]^T[K][\phi] = \left[\omega^2\right] \tag{3.6-12}$$

Matrices M_{HH} and K_{HH} are diagonal matrices with dimension H x H. Modal mass matrix represents unit diagonal matrix, and modal stiffness matrix is diagonal with frequency squared on it. Modal transformation matrix ϕ can be used to change from finite element coordinate basis to basis of eigenvectors of the generalized problem, i.e.

$$\{X\} = [\phi]\left\{X_H\right\} \tag{3.6-13}$$

where X is vector of finite element nodal displacements and X_H generalized modal displacements.

If the same transformation is applied to system of equations governing the dynamic response of system of finite elements (3.6-10), the system is transformed to modal generalized form, i.e.

$$\left[-\Omega^2 M_{HH} + i\Omega C_{HH} + (1+ig) K_{HH}\right] \cdot \overline{X}_H(\Omega) = \overline{P}_H(\Omega) \tag{3.6-14}$$

This system is solved for every excitation frequency Ω over the frequency range. The results are vectors of complex generalized deflections $\overline{X}_H(\Omega)$. The global solution vector is recovered by modal transformation as well, i.e.

$$\left\{\overline{X}(\Omega)\right\} = [\phi]\left\{\overline{X}_H(\Omega)\right\} \tag{3.6-15}$$

which allows to calculate grid velocities and accelerations

$$X'(t) = \left(\overline{X}(\Omega) \cdot e^{i\Omega t}\right)' = i\Omega \cdot \overline{X}(\Omega) \cdot e^{i\Omega t} \tag{3.6-16}$$

$$X''(t) = \left(\overline{X}(\Omega) \cdot e^{i\Omega t}\right)'' = -\Omega^2 \cdot \overline{X}(\Omega) \cdot e^{i\Omega t} \tag{3.6-17}$$

Structure complex internal loads are calculated by utilizing standard finite element approach, i.e. from relative nodal deflections.

UNA PROCEDURE :

The frequency response analysis is based on mode superposition method. The following are steps as performed by program:

1. Solve for the eigenvalues and eigenvectors of the problem by LANCZOS algorithm. All explanations made for modal analysis (section 3.2) are valid for this step as well.

2. Transform load vector to the modal basis

$$\left\{\overline{P}_H(\Omega)\right\} = [\phi]^T \left\{\overline{P}(\Omega)\right\}$$

3. Create and solve modal system of equations (3.6.14). The generalized system of equations has the following appearance

$$\left[-\Omega^2 M_{1HH}+i\Omega\cdot\left(B_{2HH}+S_{2HH}\right)+i\cdot\left(B_{3HH}+S_{3HH}\right)+K_{4HH}\right]\cdot\bar{X}_H(\Omega)=\bar{P}_H(\Omega)$$

where

$\left[M_{1HH}\right]=[\phi]^T\left[\Sigma m_e\right][\phi]$: modal mass, all elements mass.

$\left[B_{2HH}\right]=[\phi]^T\left[\Sigma b_{2e}\right][\phi]$: modal viscous damping, all elements.

$\left[S_{2HH}\right]=\left[\Sigma g_h\omega_h m_h\right]$: modal viscous damping, direct modal definition.

$\left[B_{3HH}\right]=[\phi]^T\left[\Sigma b_{3e}\right][\phi]$: modal complex stiffness damping, all elements.

$\left[S_{3HH}\right]=\left[\Sigma g_h\omega_h^2 m_h\right]$: modal complex stiffness damping, direct modal definition.

$\left[K_{4HH}\right]=[\phi]^T\left[\Sigma k_{4e}\right][\phi]$: modal structural stiffness, all structural elements.

$\left\{\bar{P}_H(\Omega)\right\}=[\phi]^T\left\{\bar{P}(\Omega)\right\}$: modal mechanical load vector.

$\left\{\bar{X}_H(\Omega)\right\}=[\phi]^T\left\{\bar{X}(\Omega)\right\}$: modal solution vector.

Ω : excitation frequency [rad/sec].

Direct modal viscous damping is defined as

$$i\Omega\cdot S_{2hh}=i\Omega\cdot\frac{g_h}{\omega_h}k_h=i\Omega\cdot g_h\omega_h m_h \qquad h=1,2,....,H$$

and direct modal complex stiffness damping is defined as

$$i\cdot S_{3hh}=i\cdot g_h k_h=i\cdot g_h\omega_h^2 m_h \qquad h=1,2,....,H$$

where Ω is excitation frequency, k_h , m_h and ω_h are modal stiffness, modal mass and undamped vibration frequency of the h-th mode. g_h is modal damping factor.

4. Recover global solution vector by modal transformation (3.6-15). Calculate velocities and accelerations (3.6-16), (3.6-17). Calculate structure complex internal loads from relative nodal deflections.

5. Repeat steps 2-4 for each excitation frequency of interest.

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. The following are modal and frequency response (highlighted) related commands:

1. DLOAD : Frequency response load selection.
2. FREQ : Frequency response forcing set selection.
3. FRESP : Frequency response output request and control.
4. FZERO : Specify tolerance for rigid body modes in [Hz].
5. GRAV : Scale mass matrix in order to produce compatible system of units.
6. LAMBDA : Specify desired number of modes. (Default = 5).
7. LMODES : Number of lowest modes to be used in generalized problem.
8. MASTYP : Select mass matrix type, consistent (recommended) or lumped.
9. MGROUP : Selects a group of elements to be included in mass matrix. (Default is all).
10. MODES : Request modal displacement printout.
11. SHIFT : Stiffness matrix shift in [Hz]. Negative shift is used to extract rigid modes.
12. SDAMP : Modal damping definition.
13. SYSTEM : Service printing, i.e. generalized matrices M_{1HH} - K_{4HH}, modal deflections etc.

BULK DATA:

Used to define finite element model. The following are frequency solutions entries:

1. FREQ : Frequency response frequency set.
2. FREQ1 : Frequency response frequency set.
3. FREQ2 : Frequency response frequency set.
4. RLOAD : Frequency response dynamic load.

TIPS:

- Dynamic load is applied by RLOAD cards. At least one static load set P_j^{stat} such as FORCE, MOMENT, QPRESS, etc., is required in order to create dynamic load set. Static load defines load amplitude (shape), while RLOAD card combines it with frequency and time dependent functions and create source *j*. Total dynamic load is composed from all sources *j*, i.e.

$$P(f) = \sum_j P_j^{stat} F_j(f) \cdot e^{i \cdot [\Omega t + \varphi_j(f)]}$$

where P_j^{stat} : source shape (static load set).
$F_j(f)$: frequency dependent scaling function.
$\varphi_j(f)$: frequency dependent phase function.
$\Omega = 2\pi f$: excitation frequency.

- Use GRAV Executive Control Command in order to match compatible system of units. Failure to achieve this leads to erroneous natural frequencies and structural response. Calculated free vibration frequencies are in units of cycles per second [Hz].

- Experiment with a different number of calculated modes (LAMBDA). Generally speaking, for a higher excitation frequencies a larger number of modes are required in order to represents structural response properly.

- MGROUP Executive Control command is used to control vibration area of interest. Only elements selected by this command will be used to form mass matrix. *This allows free vibration analysis of the isolated part of the large structure.* The non-selected elements form elastic support for the analysed part (elastic boundary conditions).

Executive Control Section Example

```
$
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "FREQUENCY RESPONSE EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 6                    ! select frequency response solution
VER = 7.2                  ! specify version of the Bulk Data file
$
LAMBDA = 5                 ! specify number of modes
SHIFT  = -0.1              ! shift -0.1 [Hz], extract rigid body modes
FZERO  =  0.001            ! tolerance for rigid body modes in [Hz]
$
$
$ FRERQUENCY RESPONSE CONTROL
$ ---------------------------
$
SDAMPING CRIT = 0.01       ! viscous modal damping C/Co=0.01
$
FREQ   = 1                 ! select frequency response forcing set (FREQ*)
DLOAD  = 1                 ! select frequency response load set (RLOAD)
$
$
$ OUTPUT REQUEST
$ --------------
$
FRESP ACCE = RANGE   4   4   COMP   1  2      ! print accelerations
FRESP BAR  = RANGE 103 103   COMP  11 11      ! print CBAR internal loads
$
$
$ FEMAP OUTPUT REQUEST
$ --------------------
$
FEMAP MODES = ON            ! create FEMAP file with modal shapes
$
$
$ MODAL matrices
$ --------------
$
SYSTEM  M1HH = FULL         ! print modal mass matrix
SYSTEM  S2HH = FULL         ! print modal viscous damping matrix
SYSTEM  K4HH = FULL         ! print modal stiffness matrix
$
BEGIN BULK
```

Bulk Data Section Example

```
$
$ Grid points
$
GRID          1       0      1.      2.      0.       0     345
GRID          2       0      0.      2.      0.       0     345
GRID          3       0      0.      1.      0.       0     345
GRID          4       0      0.      0.      0.       0    3456
$
$ Elements
$
CBAR        101     100       1       2      0.      1.      0.
CBAR        102     100       2       3     -1.      0.      0.
CBAR        103     100       3       4     -1.      0.      0.
$
CMASS         1       1       0     0.5
CMASS         2       2       0      1.
CMASS         3       3       0      1.
CMASS         4       4       0   1.e+6
$
$ Properties
$
PBAR        100       1    1.e5      1.      1.      1.
$
$ Materials
$
MAT1          1      1.      1.    0.33
$
$ Load sources Ps
$
FORCE       101       4       0   1.e+6      1.      0.      0.
FORCE       102       4       0   1.e+6      0.      1.      0.
$
$
$ Frequency response dynamic load P = Ps x cos (wt).
$
RLOAD         1     101
RLOAD         1     102
$
$
$ Forcing frequencies (FREQ1, FREQ2) to be used in frequency response
$
FREQ1         1   .0699   .0001      30
FREQ1         1   .2485   .0005      30
FREQ1         1   .7290   .0010      30
$
FREQ2         1     .01    1.01     300
$
ENDDATA
```

Output Example

```
                         FREQUENCIES

  MODE      FREQUENCY     MODAL MASS       ERROR (%)     NODE    COMP

     1            0.0    1.00000E+00    5.53292E-18         2  X2-tra
     2            0.0    1.00000E+00    6.04051E-18         2  X1-tra
     3    7.07198E-02    1.00000E+00    7.65620E-29         1  X1-tra
     4    2.55021E-01    1.00000E+00    5.73232E-27         1  X3-rot
     5    7.43954E-01    1.00000E+00    7.39568E-24         1  X3-rot



                   C O M P L E X   A C C E L E R A T I O N

                GRID   :        4    COMPONENT  1  ( T1-tra )
                ---------------------------------------------

              FREQUENCY (Hz)        AMPLITUDE     PHASE (deg)

               1.000000E-02      9.999975E-01          0.000
               1.015503E-02      9.999975E-01          0.000
               1.031246E-02      9.999975E-01          0.000



                  C O M P L E X   I N T E R N A L   L O A D

                BAR    :      103    COMPONENT 11  (M1-end B)
                ---------------------------------------------

              FREQUENCY (Hz)        AMPLITUDE     PHASE (deg)

               7.050000E-02      1.299888E+02        287.648
               7.054984E-02      1.322118E+02        283.887
               7.060000E-02      1.339173E+02        279.982
               7.070000E-02      1.354632E+02        271.964
               7.080000E-02      1.343447E+02        263.895
               7.090000E-02      1.307273E+02        256.085
```

The random response solution (SOL=8) is a dynamic analysis of the random process. A random process is a family, or ensemble, of *n* random variables related to a similar phenomenon which may be function of one or more independent variables. For example, when aircraft is flying throughout continuous gust field it is experiencing a load which is of the stochastic nature. In general, gust profile cannot be explicitly described as a function of time, because gust speed and intensity varies in the randomly manner over wide range of frequencies. Fortunately gust random profiles exhibits a certain regular average values. This characteristic of random phenomena is called *statistical regularity*. Wide accepted approach in random gust analysis is to idealize its profile as a "stationary Gaussian random process".

The analysis of the random process is based on application of frequency response technique (section 3.6). This requires that system is linear and that the excitation is stationary with respect to time. The following are few important quantities from the random analysis theory:

The autocorrelation function $R(\tau)$ of a physical variable x is defined as

$$R(\tau)=\lim_{T\to\infty}\frac{1}{T}\int_0^T x(t)x(t-\tau)dt \qquad (3.8\text{-}1)$$

$R(0)$ is the time average value of x^2. The power spectral density function $\phi(\Omega)$ of x is defined by

$$\phi(\Omega)=\lim_{T\to\infty}\frac{2}{T}\left|\int_0^T e^{-i\Omega t}x(t)dt\right|^2 \qquad (3.8\text{-}2)$$

It can be shown that autocorrelation function and power spectral density are Fourier transforms of each other. Thus

$$R(\tau)=\frac{1}{2\pi}\int_0^\infty \phi(\Omega)\cos(\Omega\tau)d\Omega \qquad (3.8\text{-}3)$$

from which follows the mean-square theorem

$$\bar{x}^2=R(0)=\frac{1}{2\pi}\int_0^\infty \phi(\Omega)d\Omega \qquad (3.8\text{-}4)$$

The mean-square value is of great importance in the random analysis of structures.

The number of the zero crossings (apparent frequency) is given by

$$N^2 = \frac{\int_0^{\infty} \left(\frac{\Omega}{2\pi}\right)^2 \phi(\Omega) d\Omega}{\int_0^{\infty} \phi(\Omega) d\Omega} \tag{3.8-5}$$

The next step is to establish the relation between input (excitation) and output values. For linear systems power spectral density of the response is related to the power spectral density of the source by

$$\phi_o(\Omega) = |H(\Omega)|^2 \cdot \phi_i(\Omega) \tag{3.8-6}$$

This equation is very important because is permits determining the statistical properties of structural response (acceleration, internal loads etc.) directly from statistical properties of excitation source profile. $|H(\Omega)|$ is complex frequency response amplitude of the output as defined in equation 3.6-7. Mean-square value is calculated by multiplying square of transfer function amplitude with excitation source PSD, and integrating the product over frequency range, equation (3.8-4).

$$\bar{x}_o^2 = \frac{1}{2\pi} \int_0^{\infty} \phi_o(\Omega) d\Omega = \frac{1}{2\pi} \int_0^{\infty} |H(\Omega)|^2 \cdot \phi_i(\Omega) d\Omega \tag{3.8-7}$$

UNA PROCEDURE :

In UNA, Random Response analysis is utilized as a data postprocessing that is applied to the results of a Frequency Response analysis. The Frequency Response analysis is explained in section 3.6. The following are steps as performed by program:

1. Solve for the eigenvalues and eigenvectors of the problem by LANCZOS algorithm. All explanations made for modal analysis, (section 3.2), are valid for this step as well.

2. Create and solve modal system of equations (3.6.14) for all excitation sources *j*. A multiple excitation sources *j* may be applied to the system simultaneously. The generalized system of equations has the following appearance

$$\left[-\Omega^2 M_{1HH} + i\Omega\cdot\left(B_{2HH} + S_{2HH}\right) + i\cdot\left(B_{3HH} + S_{3HH}\right) + K_{4HH}\right]\cdot \bar{X}_{jH}(\Omega) = \bar{P}_{jH}(\Omega)$$

3. Calculate structure complex response such as displacements, velocities, acceleration and internal loads. Define frequency responses H_{oj} for all output variables x_o of interest due to all excitation sources j, i.e.

$$x_{oj}(\Omega) = H_{oj}(\Omega) P_j(\Omega) \qquad j=1,2,...,J$$

4. Calculate Power Spectral Density of the response by superimposing the frequency response and source PSD on the following way :

   - If the sources P_1, P_2, P_3, etc., are statistically independent, then power spectral density of the total response is equal to the sum of power spectral densities due to individual sources

   $$\phi_o(\Omega) = \sum_j \phi_{oj}(\Omega) = \sum_j \left|H_{oj}(\Omega)\right|^2 \phi_j(\Omega)$$

   - If the sources are statistically correlated, the degree of correlation is expressed by a cross-spectral density ϕ_{jk}, and power spectral density of the response is evaluated as

   $$\phi_o(\Omega) = \sum_j \sum_k H_{oj}(\Omega) H^*_{ok}(\Omega) \phi_{jk}(\Omega)$$

   where H^*_{ok} is the complex conjugate of H_{ok}. Repeat steps 2-4 for all excitation frequencies Ω_n , n=1,2,..., N. Plot power spectral densities ϕ_o versus frequency Ω.

5. Calculate the root mean square (r.m.s) of the response as the square root of a trapezoidal approximation to equation (3.8-4), i.e.

$$x_{r.m.s.} = \sqrt{\frac{1}{4\pi} \sum_{n=1}^{N-1} \left[\phi_o(\Omega_{n+1}) + \phi_o(\Omega_n)\right] (\Omega_{n+1} - \Omega_n)}$$

6. Calculate number of zero crossings as trapezoidal approximation to equation (3.8-5).

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. The following are modal, frequency response and random response (highlighted) related commands:

1. DLOAD : Frequency response load selection.
2. FREQ : Frequency response forcing set selection.
3. FRESP : Frequency response output request and control.
4. FZERO : Specify tolerance for rigid body modes in [Hz].
5. GRAV : Scale mass matrix in order to produce compatible system of units.
6. LAMBDA : Specify desired number of modes. (Default = 5).
7. LMODES : Number of lowest modes to be used in generalized problem.
8. MASTYP : Select mass matrix type, consistent (recommended) or lumped.
9. MGROUP : Selects a group of elements to be included in mass matrix. (Default is all).
10. MODES : Request modal displacement printout.
11. PSDRESP : Auto PSD response output
12. RANDOM : Random vibration PSD input selection
13. SHIFT : Stiffness matrix shift in [Hz]. Negative shift is used to extract rigid modes.
14. SDAMP : Modal damping definition.
15. SYSTEM : Service printing, i.e. generalized matrices M_{1HH} - K_{4HH}, modal deflections etc.

BULK DATA:

Used to define finite element model. The following are frequency response and random response (highlighted) related entries:

1. FREQ : Frequency response frequency set.
2. FREQ1 : Frequency response frequency set.
3. FREQ2 : Frequency response frequency set.
4. RANDPS : PSD specification
5. RLOAD : Frequency response dynamic load.
6. TABRND1 : PSD table

TIPS:

- Input PSD is combined with dynamic excitation source P_j . It is possible to combine different PSD sets with various sources *j* in the same run by repeating RANDPS Bulk Data Entry, see enclosed Bulk Data Example.

- Dynamic load is applied by RLOAD cards. At least one static load set P_j^{stat} such as FORCE, MOMENT, QPRESS, etc., is required in order to create dynamic load set. Static load defines load amplitude (shape), while RLOAD card combines it with frequency and time dependent functions and create source *j*. Total dynamic load is composed from all sources *j*.

- Use GRAV Executive Control Command in order to match compatible system of units. Failure to achieve this leads to erroneous natural frequencies and structural response. Calculated free vibration frequencies are in units of cycles per second [Hz].

- Experiment with a different number of calculated modes (LAMBDA). Generally speaking, for a higher excitation frequencies a larger number of modes are required in order to represents structural response properly.

Executive Control Section Example

```
$
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "RANDOM RESPONSE EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 8                   ! select random response solution
VER = 7.2                 ! specify version of the Bulk Data file
$
LAMBDA = 5                ! specify number of modes
SHIFT  = -0.1             ! shift -0.1 [Hz], extract rigid body modes
FZERO  =  0.001           ! tolerance for rigid body modes in [Hz]
$
$
$ FREQUENCY AND RANDOM RESPONSE CONTROL
$ -------------------------------------
$
SDAMPING CRIT = 0.01      ! viscous modal damping C/Co=0.01
$
FREQ   = 1                ! select frequency response forcing set (FREQ*)
DLOAD  = 1                ! select frequency response load set (RLOAD)
RANDOM = 1                ! select random response PSD sets (RANDPS)
$
$
$ OUTPUT REQUEST
$ --------------
$
FRESP ACCE = RANGE   4   4  COMP  1  2        ! print complex accele.
FRESP BAR  = RANGE 103 103  COMP 11 11        ! print CBAR complex loads
$
PSD  ACCE = RANGE   4   4  COMP  1  2  FULL   ! print PSD accelerations
PSD  BAR  = RANGE 103 103  COMP 11 11  FULL   ! print CBAR PSD loads
$
$
BEGIN BULK
```

Bulk Data Section Example

```
$
Grid points
$
GRID           1       0      1.      2.      0.       0     345
GRID           2       0      0.      2.      0.       0     345
```

```
GRID           3       0      0.      1.      0.       0     345
GRID           4       0      0.      0.      0.       0    3456
$
$ Elements
$
CBAR         101     100       1       2      0.      1.      0.
CBAR         102     100       2       3     -1.      0.      0.
CBAR         103     100       3       4     -1.      0.      0.
$
CMASS          1       1       0     0.5
CMASS          2       2       0      1.
CMASS          3       3       0      1.
CMASS          4       4       0   1.e+6
$
$ Properties
$
PBAR         100       1    1.e5      1.      1.      1.
$
$ Materials
$
MAT1           1      1.      1.     0.3
$
$ Load sources Ps
$
FORCE        101       4       0   1.e+6      1.      0.      0.
FORCE        102       4       0   1.e+6      0.      1.      0.
$
$ Frequency response dynamic load P = Ps x cos (wt)
$
RLOAD          1     101
RLOAD          1     102
$
$ Forcing frequencies (FREQ1, FREQ2) to be used in frequency response
$
FREQ1          1   .0699   .0001      30
FREQ1          1   .2485   .0005      30
FREQ1          1   .7290   .0010      30
$
FREQ2          1     .01    1.01     300
$
$ PSD definition for random analysis
$
$            SID       J       K       X       Y     TID
$
RANDPS         1     101     101 12.5664      0.       1     ! auto-PSD
RANDPS         1     102     102 12.5664      0.       2     ! auto-PSD
RANDPS         1     101     102 12.5664      0.       3     ! cross-PSD
$
TABRND1        1
+             0.      1.      1.      1.
TABRND1        2
+             0.      .5      1.      .5
TABRND1        3
+             0.   .7071      1.   .7071
$
ENDDATA
```

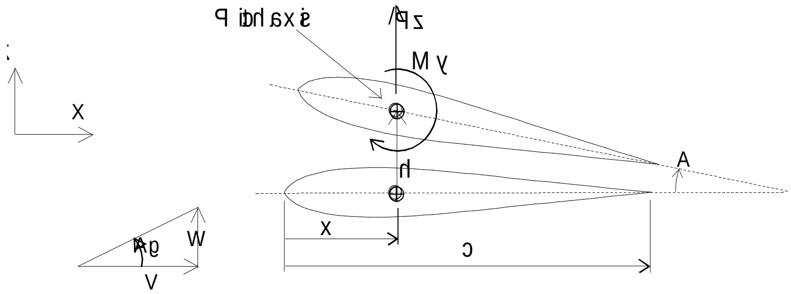
Output Example

```
                              FREQUENCIES

  MODE      FREQUENCY     MODAL MASS      ERROR (%)     NODE    COMP

     1            0.0    1.00000E+00    5.53292E-18        2  X2-tra
     2            0.0    1.00000E+00    6.04051E-18        2  X1-tra
     3    7.07198E-02    1.00000E+00    7.65620E-29        1  X1-tra
     4    2.55021E-01    1.00000E+00    5.73232E-27        1  X3-rot
     5    7.43954E-01    1.00000E+00    7.39568E-24        1  X3-rot


           A U T O   P S D  -  A C C E L E R A T I O N

           GRID   :        4    COMPONENT  1  ( T1-tra )
           ---------------------------------------------

                FREQUENCY (Hz)                PSD

                 1.000000E-02        1.256634E+01
                 1.015503E-02        1.256634E+01
                 1.031246E-02        1.256634E+01


           A U T O   P S D  -  I N T E R N A L   L O A D

           BAR    :      103    COMPONENT 11  (M1-end B)
           ---------------------------------------------

                FREQUENCY (Hz)                PSD

                 7.060000E-02        2.649432E+05
                 7.070000E-02        2.711797E+05
                 7.080000E-02        2.668032E+05


          A U T O   P S D  -  O U T P U T   S U M M A R Y
          -----------------------------------------------

 TYPE      I.D.   OUTPUT     COMPONENT           RMS      NO. POSITIVE
                            NO.     TYPE         VALUE        CROSSINGS

 GRID        4     ACCE      1    T1-tra    3.54491E+00   5.86043E-01
 GRID        4     ACCE      2    T2-tra    2.50663E+00   5.86043E-01
 BAR       103     FORCE    11  M1-end B    2.67024E+01   1.33792E-01
```

The aeroelastic response solution (SOL=9) is a dynamic analysis of the random aerodynamic gust process. For example, when aircraft is flying throughout continuous gust field it is experiencing a load which is of the stochastic nature. In general, gust profile cannot be explicitly described as a function of time, because gust speed and intensity varies in the randomly manner over wide range of frequencies. Fortunately gust random profiles exhibits a certain regular average values. This characteristic of random phenomena is called *statistical regularity*. Wide accepted approach in random gust analysis is to idealize its profile as a "stationary Gaussian random process".

The aeroelastic response analysis is based on frequency and random response techniques. Inertia, damping and elastic forces are the same, plus additional aerodynamic "damping" and "stiffness". It is assumed that all transient effects have died out after some time, and that harmonic gust excitation is steady or independent of time. The solution of dynamic system is sought in the *frequency domain* and combined with gust Power Spectral Density in order to calculate random response for the variables of interest, i.e. acceleration, internal load etc. The explanations made in chapters 3.6 and 3.8 are generally valid for gust random response as well.

The aerodynamic part of dynamic system is based on "strip" aerodynamic theory. The following pictures shows nomenclature for the oscillatory pitching and heaving motion of a flat plate:



The lift and pitching moment at location X are given as:

$$P_z = QSC_{L0} + QSC_{L\alpha}\left[C(k)\alpha - C(k)\frac{\dot{h}}{V} + C(k)\left(\frac{3}{4}c - x\right)\frac{\dot{\alpha}}{V} + \frac{c}{4}\frac{\dot{\alpha}}{V}\right] \quad (3.9\text{-}1)$$

$$M_y = QSC_{M0} + QSC_{M\alpha}\left[C(k)\alpha - C(k)\frac{\dot{h}}{V} + C(k)\left(\frac{3}{4}c - x\right)\frac{\dot{\alpha}}{V}\right] + QSC_{L\alpha}\frac{c}{4}\left(x - \frac{3c}{4}\right)\frac{\dot{\alpha}}{V}$$

where Q : dynamic pressure.
S : reference area.
$C_{L0}, C_{L\alpha}$: coefficients of lift (1/rad), default $C_{L0} = 0$, $C_{L\alpha} = 2\pi$
$C_{M0}, C_{M\alpha}$: coefficients of moment (1/rad), default $C_{M0} = 0$, $C_{M\alpha} = 0$
V : aircraft forward speed.
c : panel average chord length
x : pitch axis chordwise location
$C(k)$: Theodorsen non-stationary correction factor
α, h : Angle of attack and vertical deflection

Is assumed that $C_{L\alpha}$ and $C_{M\alpha}$ remain constant over the frequency range. C_{L0} … $C_{M\alpha}$ factors can be modified by correction factors W_{L0} ... W_{MA}. They are applied to aerodynamic properties in order to match desired data (experimental, analytical from CFD etc.). Factors will be applied in stationary and non-stationary aerodynamic regime in the form of scaling factors.

The angle of attack is composed from two parts: nominal gust angle $w(t)/V$, and contribution from rigid body rotation and elastic angle of twist $\theta(t)$. Gust speed $w(t)$ is assumed to performs a steady harmonic oscillation over the frequency range

$$w(t) = w_o \cdot e^{i\Omega(t+\tau)} = w_o \cdot e^{i\Omega\tau} \cdot e^{i\Omega t} = \overline{w}_o(\Omega) \cdot e^{i\Omega t} \qquad (3.9\text{-}2)$$

Time delay is used to model gust penetration effect. Gust pressure which acts at the nose of aircraft at one moment, changes a phase until another aircraft part arrives at the same point (i.e. tail). If nose of the aircraft is used as a reference point being in the phase with the gust pressure, time lag for all other points on the aircraft is defined by

$$\tau = -\frac{X_{cp}}{V} \qquad (3.9\text{-}3)$$

where X_{cp} is control point (pitching axis) location measured from nose, and V is aircraft speed.

Control point vertical speed and elastic/rigid body angle of attack are result of fluid / structure interaction and cannot be defined in advance as in the case of the gust angle. In this stage we can recall the section 3.6 which explains that if input is steady state excitation, output values are steady state oscillating variables as well. By assuming the solutions for vertical velocity and elastic/rigid body angle in the following form

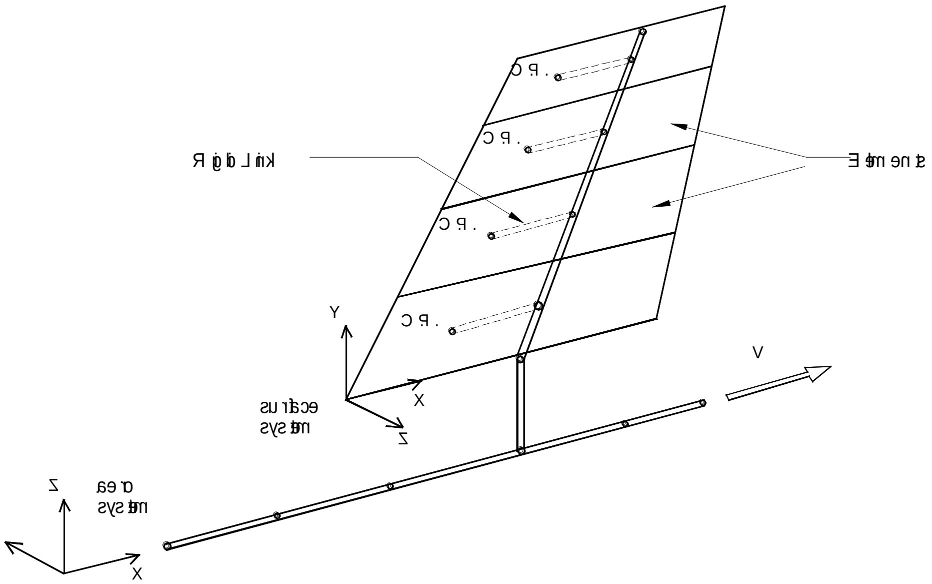
$$\dot{h} = V_Z(t) = \left(\overline{U}_z(\Omega)\cdot e^{i\Omega t}\right)' = \overline{U}_z(\Omega) \cdot i\Omega \cdot e^{i\Omega t} = i\Omega \cdot \overline{U}_z(\Omega) \cdot e^{i\Omega t} \qquad (3.9\text{-}4)$$

$$\theta(t) = \overline{\theta}(\Omega) \cdot e^{i\Omega t} \qquad (3.9\text{-}5)$$

and substituting to equations 3.9-1, aerodynamic lift and pitching moment are expressed as a functions of gust speed and control point location deflections. Aerodynamic load in this form is integrated into dynamic system of equations, and solution is sought for complex frequency response.

Aero modelling

Aerodynamic stiffness and damping elements are developed from equation (3.9-6). Aerodynamic modelling is utilized by splitting the lifting surface on finite number of strips or elements. Panel is defined by four corner aero grids. Grids have to lay in one plane thus defining element local x-y plane. Lift is acting in local z-direction and pitching moment around local y-axis. For each element the aero stiffness, damping and gust load are developed at control point location and then transformed to FE structural grids via splining procedure or internal rigid link.



Typical aerodynamic mesh is shown on the above picture. Aero lifting surface is split into several strip sections each representing one aero element. Local strip x-axis must be coo-linear with aero system x-axis that specifies down-stream direction.. All matrices are derived at control point and then transformed to the structural grids via splining or internal rigid link.

Generalized problem

The generalized system of equations has the same form as presented in section 3.6. There are three new elements of the system - aerodynamic damping A_{2HH}, aerodynamic stiffness A_{4HH} and gust load vector $P_w(\Omega)$, with all of them derived from equation (3.9-6). The generalized system of equations is defined as

$$\left[-\Omega^2 M_{1HH} + i\Omega\cdot\left(\frac{Q}{V} A_{2HH} + B_{2HH} + S_{2HH}\right) + i\cdot\left(B_{3HH} + S_{3HH}\right) + Q.A_{4HH} + K_{4HH}\right]\cdot \bar{X}_H(\Omega) = $$
$$= Q\cdot \bar{P}_{wH}(\Omega)$$

where (3.9-7)

$\left[M_{1HH}\right] = [\phi]^T\left[\sum m_e\right][\phi]$	: modal mass, all elements mass.
$\left[A_{2HH}\right] = [\phi]^T\left[\sum a_{2e}\right][\phi]$	: modal aerodynamic damping, all aero elements.
$\left[B_{2HH}\right] = [\phi]^T\left[\sum b_{2e}\right][\phi]$	: modal viscous damping, all elements.
$\left[S_{2HH}\right] = \left[\sum g_h \omega_h m_h\right]$	: modal viscous damping, direct modal definition.
$\left[B_{3HH}\right] = [\phi]^T\left[\sum b_{3e}\right][\phi]$	: modal complex stiffness damping, all elements.
$\left[S_{3HH}\right] = \left[\sum g_h \omega_h^2 m_h\right]$	: modal complex stiffness damping, direct modal definition.
$\left[A_{4HH}\right] = [\phi]^T\left[\sum a_{4e}\right][\phi]$	: modal aerodynamic stiffness, all aero elements.
$\left[K_{4HH}\right] = [\phi]^T\left[\sum k_{4e}\right][\phi]$	: modal structural stiffness, all structural elements.
$\left\{\bar{P}_{wH}(\Omega)\right\} = [\phi]^T\left\{\bar{P}_w(\Omega)\right\}$	: modal gust load vector
$\left\{\bar{X}_H(\Omega)\right\} = [\phi]^T\left\{\bar{X}(\Omega)\right\}$	: modal solution vector.

Ω	: Excitation frequency [rad/sec]
Q	: Dynamic pressure
V	: Velocity

UNA PROCEDURE :

In UNA, Aeroelastic Response is utilized as a random vibration analysis with a gust harmonic excitation. The Frequency and Random Response analysis are explained in sections 3.6 and 3.8, respectively. The following are steps as performed by program:

1. Solve for the eigenvalues and eigenvectors of the problem by LANCZOS algorithm. All explanations made for modal analysis (section 3.2) are valid for this step as well.

2. Create and solve modal system of equations (3.9.7). Gust excitation is applied via GUST Entry.

3. Calculate structural complex response, such as displacements, velocities, acceleration and internal loads. Define frequency responses H_{ow} for all output variables x_o of interest due to gust load, i.e.

   $$x_o(\Omega) = H_{ow}(\Omega) P_w(\Omega)$$

4. Calculate Power Spectral Density of the response by superimposing the frequency response and gust based PSD (Von Karman, Dryden or user defined) on the following way :

   $$\phi_o(\Omega) = \left| H_{ow}(\Omega) \right|^2 \phi_w(\Omega)$$

   Repeat steps 2-4 for all excitation frequencies Ω_n , n=1,2,..., N. Plot power spectral densities ϕ_o versus frequency Ω.

5. Calculate the root mean square (r.m.s) of the response as the square root of a trapezoidal approximation to equation (3.8-4), i.e.

   $$x_{r.m.s.} = \sqrt{\frac{1}{4\pi} \sum_{n=1}^{N-1} \left[\phi_o(\Omega_{n+1}) + \phi_o(\Omega_n) \right] (\Omega_{n+1} - \Omega_n)}$$

6. Calculate the number of zero crossings as trapezoidal approximation to equation (3.8-5).

EXECUTIVE CONTROL COMMANDS:

Used to control solution process. The following are modal, frequency response, random response and aerodynamic gust (highlighted) related commands:

1. FREQ : Frequency response forcing set selection.
2. FRESP : Frequency response output request and control.
3. FZERO : Specify tolerance for rigid body modes in [Hz].
4. GRAV : Scale mass matrix in order to produce compatible system of units.
5. GUST : Frequency response gust load selection.
6. LAMBDA : Specify desired number of modes. (Default = 5).
7. LMODES : Number of lowest modes to be used in generalized problem.
8. MASTYP : Select mass matrix type, consistent (recommended) or lumped.
9. MGROUP : Selects a group of elements to be included in mass matrix. (Default is all).
10. MODES : Request modal displacement printout.
11. PSDRESP : Auto PSD response output.
12. RANDOM : Random gust PSD input selection.
13. SHIFT : Stiffness matrix shift in [Hz]. Negative shift is used to extract rigid modes.
14. SDAMP : Modal damping definition.
15. SYSTEM : Service printing, i.e. generalized matrices M_{1HH} - K_{4HH}, modal deflections etc.

BULK DATA:

Used to define finite element model. The following are frequency response, random response and aerodynamic gust (highlighted) related entries :

1. AERO : Aerodynamic parameters for gust analysis.
2. CSTRIP : Aerodynamic strip element.
3. FREQ : Frequency response frequency set.
4. FREQ1 : Frequency response frequency set.
5. FREQ2 : Frequency response frequency set.
6. GRIDA : Aerodynamic grids for defining panel corners.
7. GUST : Defines the stationary gust load.
8. PSTRIP : Aerodynamic strip panel properties.
9. RANDPS : PSD specification.
10. SPLINE : Connection of aero and structural mesh.
11. TABRND1 : User defined PSD table.
12. TABRNDG : Gust PSD functions : Von-Karman or Dryden spectrums.
13. WTFACT : Aerodynamic correction factors.

TIPS:

- Gust load is applied by GUST Bulk Data Entry.
- Rigid Aircraft solution is achieved by setting LMODES = No. of rigid modes. This type of run isolates a rigid aircraft portion of response (i.e. Duch-Roll mode) from elastic contribution.
- Experiment with a different numbers of calculated modes (LAMBDA). Typically, by accounting for frequencies up to 10 [Hz] provides a satisfactory convergence because of decrease of Von-Karman PSD in the higher excitation range.
- Use GRAV Executive Control Command in order to match compatible system of units.

Executive Control Section Example

```
$
$
$  EXECUTIVE CONTROL SECTION
$
$
TITLE = "AEROELASTIC RESPONSE EXAMPLE"
$
$
$ FE RUN CONTROL
$ --------------
$
SOL = 9                       ! select aeroelastic response solution
VER = 7.2                     ! specify version of the Bulk Data file
SPC = 20                      ! selects anti-symmetric boundary conditions
$
LAMBDA = 12                   ! specify number of modes
SHIFT  = -1.                  ! shift -1. [Hz], extract rigid body modes
FZERO  = 0.01                 ! tolerance for rigid body modes in [Hz]
FORMAT = 1                    ! specify Exponent output format
$
$
$ FREQUENCY AND RANDOM RESPONSE CONTROL
$ --------------------------------------
$
SDAMPING G = 0.02  STRUCTURAL ON        ! structural modal damping g=0.02
$
FREQ   = 1                     ! select frequency response set (FREQ*)
GUST   = 102                   ! select gust response load set (GUST)
RANDOM = 102                   ! select random response PSD sets (RANDPS)
$
$
$ OUTPUT REQUEST
$ --------------
$
FRESP ACCE =  SET 2  COMP 6  6          ! print complex accelerations
FRESP BAR  =  SET 3  COMP 9 10          ! print CBAR complex loads
$
PSD ACCE   =  SET 2  COMP 6  6          ! print PSD accelerations
PSD BAR    =  SET 3  COMP 9 10          ! print CBAR PSD loads
PSD STRIP  =  SET 4  COMP 7  7          ! print STRIP PSD loads
$
$
$ MODAL MATRICES
$ --------------
$
SYSTEM  M1HH = ROW 1 6 COLUMN 1 6
SYSTEM  A2HH = ROW 1 6 COLUMN 1 6
SYSTEM  S3HH = ROW 1 6 COLUMN 1 6
SYSTEM  A4HH = ROW 1 6 COLUMN 1 6
SYSTEM  K4HH = ROW 1 6 COLUMN 1 6
$
BEGIN BULK
```

Bulk Data Section Example

```
$
$
$  BULK DATA SECTION
$
$
$ AERO DATA for M=0.385, 1000 [ft], UNIT GUST W=1/V
$
$           ASID    RSID    REFC    REFB    REFS       Q    VELO    MACH
$
AERO          88      88    241.  17000.  3.25+5 1.013-3  13051.   0.381
GUST         102 2.335-3      0.      88
TABLE         88
+             0.      0.   0.005      1.     10.      1.
$
$
$ FIN AERO ELEMENTS
$ -----------------
$
PSTRIP     94101    4100  102.08      0.      0.  3.25+5  .01190      0.
PSTRIP     94102    4100  128.61   46.22      0.  3.25+5  .03475      0.
PSTRIP     94103    4100  153.34  138.66      0.  3.25+5  .13664      0.
PSTRIP     94105    4100  222.93   231.1      0.  3.25+5  .16493      0.
$
CSTRIP     94101   94101  104101  104102  104104  104103    4101
CSTRIP     94102   94102  104103  104104  104106  104105    4102
CSTRIP     94103   94103  104105  104106  104108  104107    4103
CSTRIP     94105   94105  104107  104108  104110  104109    4105
$
$
$ FREQUENCIES FOR FREQUENCY RESPONSE ANALYSIS
$ -------------------------------------------
$
FREQ1          1     0.0    0.05     199
$
$
$ VON KARMAN PSD GUST = 85 ft/sec
$
$            SID       J       K       X       Y     TID
$
RANDPS       102     102     102      1.      0.     900
$
$            TID    TYPE     L/U      WG
$
TABRNDG      900       1  5.8380     85.
$
INCLUDE     "GUST_V27.UNA"
$
ENDDATA
```

Output Example

FREQUENCIES

MODE	FREQUENCY	MODAL MASS	ERROR (%)	NODE	COMP
1	0.0	1.00000E+00	4.65469E-11	1001	X2-tra
2	0.0	1.00000E+00	1.65603E-11	2124	X3-tra
3	0.0	1.00000E+00	1.13963E-11	3116	X2-tra
4	2.56955E+00	1.00000E+00	2.20783E-15	3116	X3-tra
5	3.38815E+00	1.00000E+00	1.76444E-13	3116	X3-tra
6	3.88604E+00	1.00000E+00	1.39603E-10	2124	X3-tra
7	4.55343E+00	1.00000E+00	2.55687E-06	3116	X1-tra
8	5.16872E+00	1.00000E+00	1.70820E-07	3116	X3-tra
9	6.13777E+00	1.00000E+00	1.52045E-05	3116	X3-tra
10	7.27915E+00	1.00000E+00	2.56594E-04	3116	X3-tra
11	8.68589E+00	1.00000E+00	1.72532E-02	3116	X3-tra
12	9.61772E+00	1.00000E+00	1.81298E-01	3116	X1-tra

A U T O   P S D - O U T P U T   S U M M A R Y

TYPE	I.D.	OUTPUT	COMPONENT NO.	COMPONENT TYPE	RMS VALUE	NO. POSITIVE CROSSINGS
GRID	5001	ACCE	6	R3-rot	1.29000E+00	4.72333E+00
GRID	99999	ACCE	6	R3-rot	5.72149E-01	4.01479E+00
BAR	112119	FORCE	9	Torque	1.60177E+04	2.51646E+00
BAR	112119	FORCE	10	M1-end A	1.11478E+04	2.13566E+00
BAR	114100	FORCE	9	Torque	3.80424E+03	2.11441E+00
BAR	114100	FORCE	10	M1-end A	3.01068E+03	7.58413E+00
STRIP	91005	FORCE	6	Pz	3.96728E+00	7.05104E-01
STRIP	94105	FORCE	6	Pz	1.18277E+01	8.08577E-01

Section 4

Verification Examples

Title : Lateral buckling of a simply supported beam

Reference: Stephen Timoshenko, James Gere
"Theory Of Elastic Stability"
McGraw-Hill Book Company, 1961

Description: The simply supported beam is analysed for buckling load. The beam is idealized with twenty CBAR elements. Total of three load cases are considered:

1. Force at midspan Pz = -20.

2. Moment at both ends My = +/- 400.

3. Continuous load Qz = -1.

Geometry, properties, boundary conditions and load case 1 are shown at Figure 1.

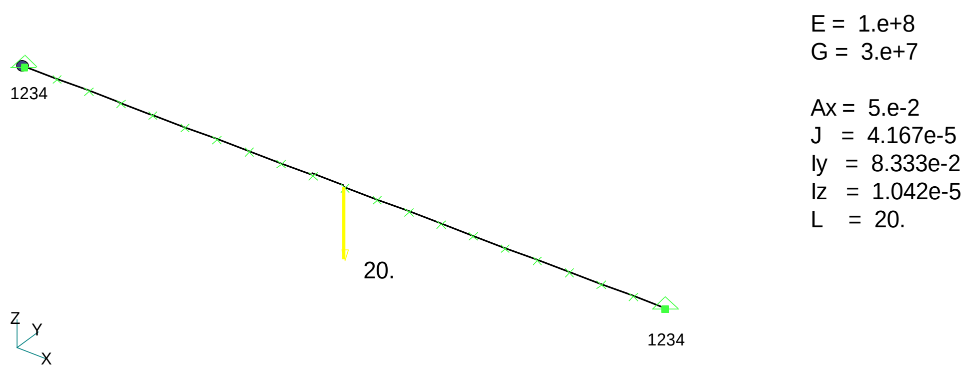


Figure 1. Simply supported beam

Solution:

The following are expressions for buckling load as per mentioned reference:

1. $Pc = 16.94\frac{\sqrt{EI_Z GJ}}{L^2}$ (eq. 6-36)

2. $Mc = 3.14\frac{\sqrt{EI_Z GJ}}{L}$ (eq. 6-10)

3. $Qc = 28.3\frac{\sqrt{EI_Z GJ}}{L^3}$ (eq. 6-39)

The following table compares finite element and analytical values:

Load Case	UNA		Analytical
	Lmin	Load	Load
1	2.41927	48.38	48.33
2	0.44866	179.46	179.28
3	4.05153	4.052	4.037

Table 1. FEM and Analytical Buckling Load

TV002

Title : In-plane bending of 2-D elements

Reference: S. Timoshenko and J.N. Goodier
"Theory Of Elasticity"
McGraw-Hill Book Company, 1951

Description: In-plane bending of the CSHELL4 elements is analysed. Cantilevered beam is modelled with 3 elements and total of three load cases are considered:

1. Compression force at free end Px = -1.

2. Moment at free end Mz = 0.5

3. Transversal force at free end Py = 1.

Geometry, properties, boundary conditions and load case 3 are shown at Figure 1.

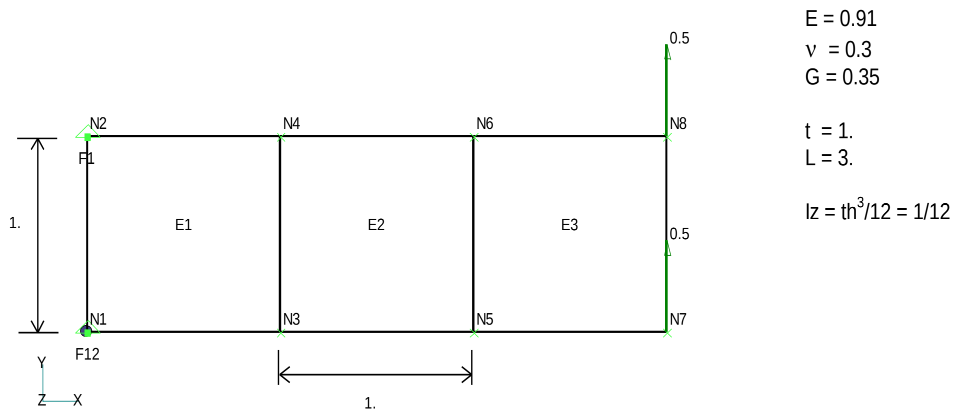


Figure 1. Cantilevered beam

Solution:

The following are expressions for displacement at free end:

1. $U_X = \frac{P_X L}{EA} = -3.30$

2. $U_Y = \frac{M_Z L}{2EI_Z} = 29.67$

3. $U_Y = \frac{P_Y L^3}{3EI_Z} + \frac{(4+5\nu)P_Y L}{2Eh} = 127.75$

The second term in the equation 3) represents transverse shear effect. The FEM values are shown with and without incompatible modes, see PARAM PLATE {5} for explanation. The following table compares analytical and finite element displacements:

Load Case	Analytical	UNA	
		incop=YES	incop=NO
1	-3.30	-3.30	-3.30
2	29.67	29.67	20.00
3	127.75	124.87	87.12

Table 1. Displacement comparison

Title : Tapered cantilever beam

Reference: E.H. Bruhn
"Analysis and Design of Flight Vehicle Structures"
Tri-State Offset Company, 1973

Description: A tapered plain web is analysed for shear flows on the web edges. Web represents beam in the flexure. Geometry and loads are given on the following figure:

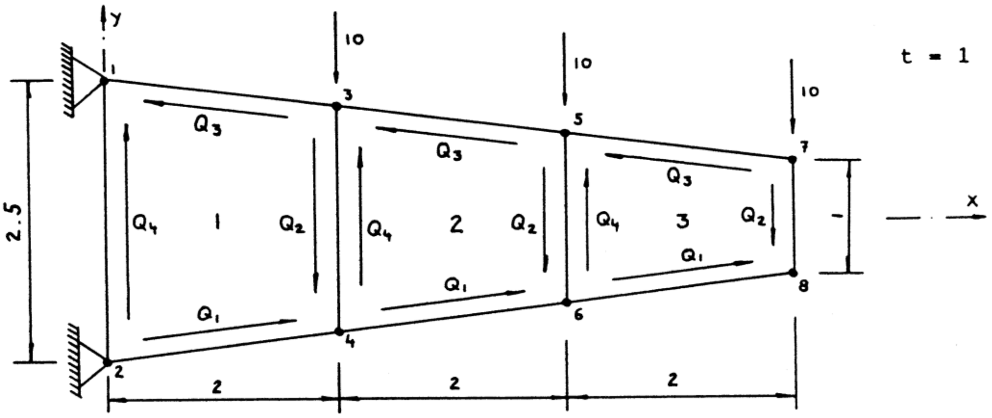


Figure 1. Cantilevered Beam

Solution:

Program generates web flanges with effective area of bt/6 where b stands for element average height and t for element thickness. Finite element results are compared with analytical solution (Shear Panels, page A7.23). Both methods gave the same results. Shear flow values are given in the following table

Element	Q1	Q2	Q3	Q4
1	-9.00	-11.25	-9.00	-7.20
2	-8.33	-11.11	-8.33	-6.25
3	-6.67	-10.00	-6.67	-4.44

Table 1. Shear Flows

UNA input file:

```
TITLE = "TV003 Tapered cantilevered beam"
$
SOL   = 1
VER   = 7.2
$
PARAM SHEAR {1} = 1
PARAM SHEAR {2} = 1
PARAM SHEAR {3} = 1
$
SUBCASE = 1
$
DISP TRA = ALL
FORCE    = ALL
$
BEGIN BULK
GRID           1       0      0.    1.25      0.       0  123456
GRID           2       0      0.   -1.25      0.       0  123456
GRID           3       0      2.    1.00      0.       0    3456
GRID           4       0      2.   -1.00      0.       0    3456
GRID           5       0      4.    0.75      0.       0    3456
GRID           6       0      4.   -0.75      0.       0    3456
GRID           7       0      6.    0.50      0.       0    3456
GRID           8       0      6.   -0.50      0.       0    3456
$
CSHEAR         1       1       2       4       3       1
CSHEAR         2       1       4       6       5       3
CSHEAR         3       1       6       8       7       5
$
PSHEAR         1       1      1.
MAT1           1   7200.   2800.
$
FORCE          1       3       0      1.      0.    -5.0     0.0
FORCE          1       4       0      1.      0.    -5.0     0.0
FORCE          1       5       0      1.      0.    -5.0     0.0
FORCE          1       6       0      1.      0.    -5.0     0.0
FORCE          1       7       0      1.      0.    -5.0     0.0
FORCE          1       8       0      1.      0.    -5.0     0.0
$
ENDDATA
```

Title : Three disks shaft torsional vibrations

Reference: Walter Hurty, Moshe Rubinstein
"Dynamics of Structures"
Prentice-Hall, 1976

Description: Problem is to determine fundamental torsional frequencies of mass-less beam shaft with three equal spaced disks. Geometry and structure data are given on following figure:

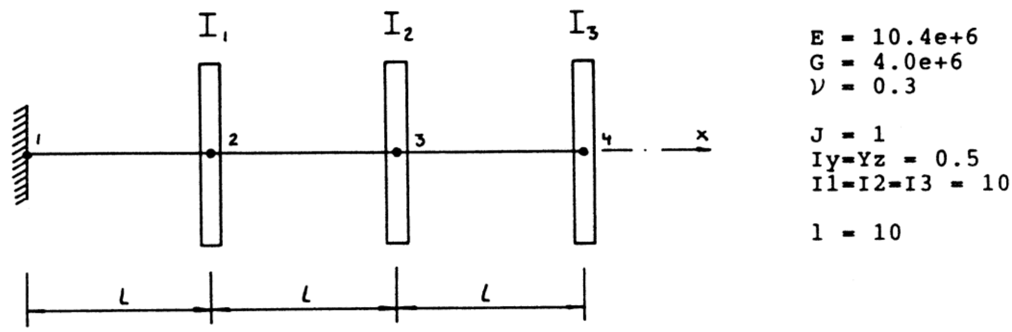


Figure 1. Three Disks Shaft

Solution:

In the case of equal disk moments of inertia and equal spaced disks, analytical solutions for frequencies are

$$f_1 = \frac{\omega_1}{2\pi} = \frac{0.445\sqrt{GJ/IL}}{2\pi}$$

$$f_2 = \frac{\omega_2}{2\pi} = \frac{1.247\sqrt{GJ/IL}}{2\pi}$$

$$f_3 = \frac{\omega_3}{2\pi} = \frac{1.802\sqrt{GJ/IL}}{2\pi}$$

The following table compares analytical and finite element frequencies (Hz):

Frequency	Analytical	UNA
1	14.165	14.166
2	39.693	39.693
3	57.359	57.358

Table 1. Natural Frequencies

UNA input file:

```
TITLE  = "TV004 - Three disks shaft torsional vibrations"
SOL    = 2
VER    = 7.2
$
LAMBDA = 3
BEGIN BULK
GRID           1       0     0.0     0.0     0.0       0  123456
GRID           2       0    10.0     0.0     0.0       0   12356
GRID           3       0    20.0     0.0     0.0       0   12356
GRID           4       0    30.0     0.0     0.0       0   12356
GRID           5       0     0.0    10.0     0.0       0  123456
MAT1           1  10.4E6   4.0E6     0.3
PBEAM          1       1     1.0     0.0     0.0     1.0     0.5     0.5
CBEAM          1       1       1       2       5
CBEAM          2       1       2       3       5
CBEAM          3       1       3       4       5
CMASS          4       2       0      0.      0.      0.      0.
+            10.      0.
CMASS          5       3       0      0.      0.      0.      0.
+            10.      0.
CMASS          6       4       0      0.      0.      0.      0.
+            10.      0.
ENDDATA
```

TV005

Title : Spring-mass system time response

Reference: Klaus-Jurgen Bathe
"Finite Element Procedures in Engineering Analysis"
Prentice-Hall, 1982

Description: Spring-mass system is analysed for time response. A constant force is acting at node 2. System is shown on the following figure:

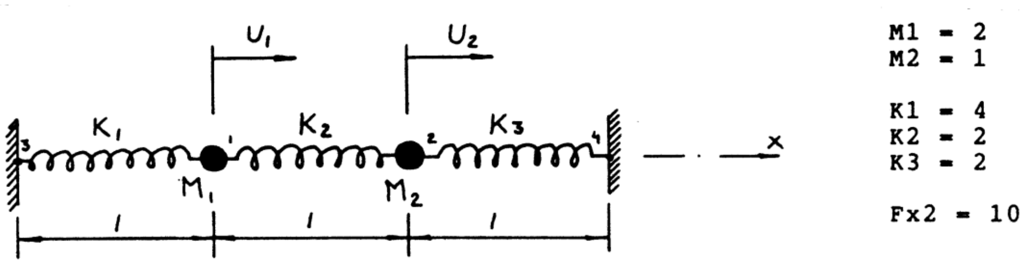


Figure 1. Spring-Mass System

Solution:

Governing equilibrium equations for the defined system is

$$\begin{bmatrix} 2 & 0 \\ 0 & 1 \end{bmatrix} \begin{Bmatrix} U_1'' \\ U_2'' \end{Bmatrix} + \begin{bmatrix} 6 & -2 \\ -2 & 4 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 10 \end{Bmatrix}$$ (ref. examples 9.1-9.7)

Analytical solutions for the free oscillations is:

$$f_1 = \frac{\omega_1}{2\pi} = \frac{\sqrt{2}}{2\pi} \qquad f_2 = \frac{\omega_2}{2\pi} = \frac{\sqrt{5}}{2\pi}$$

For initial displacement and velocity equal to zero, the displacement versus time functions are defined by following equations:

$$U_1 = \frac{5}{3}(1 - \cos\sqrt{2}\cdot t) + \frac{2}{3}(-1 + \cos\sqrt{5}\cdot t)$$

$$U_2 = \frac{5}{3}(1 - \cos\sqrt{2}\cdot t) - \frac{4}{3}(-1 + \cos\sqrt{5}\cdot t)$$

Finite element and analytical solutions are identical. Finite element output for time range 0.0 - 3.36 [sec] is shown on the following figure:

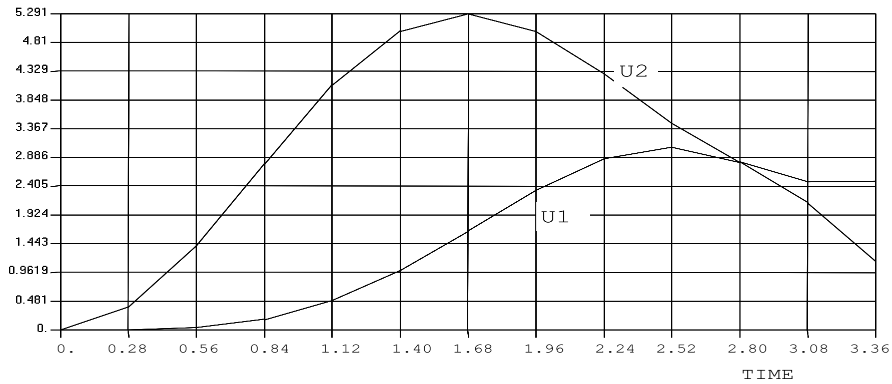


Figure 2. Deflection Time Response

UNA input file:

```
TITLE = "TV005 - Spring-mass system force response"
SOL   = 5
VER   = 7.2
LAMBDA = 2
INCREM = 12
RESPONSE = 0. 3.36
DISP TRA = ALL
FEMAP DISP  = ON
FEMAP MODES = ON
BEGIN BULK
GRID           3       0       0.      0.      0.       0  123456
GRID           1       0       1.      0.      0.       0   23456
GRID           2       0       2.      0.      0.       0   23456
GRID           4       0       3.      0.      0.       0  123456
CELAS2         1      4.       3        1       1        1
CELAS2         2      2.       1        1       2        1
CELAS2         3      2.       2        1       4        1
CMASS          4       1       0       2.
CMASS          5       2       0       1.
DFORCE         1       2       0        1     10.       0.
ENDDATA
```

Title : Bending of simply supported isotropic plate

Reference: J.N. Redy
"Energy And Variational Methods In Applied Mechanics"
John Willey & Sons, 1984

Description: This example considers a square isotropic plate which is simply supported on all edges and subjected to pressure load Qz. Normal deflection W in plate centre is required. Due to the symmetry of geometry and boundary conditions, only upper right part is modelled using 2x2 mesh of CSHELL4 elements. Structure and idealization are shown on the following figure:

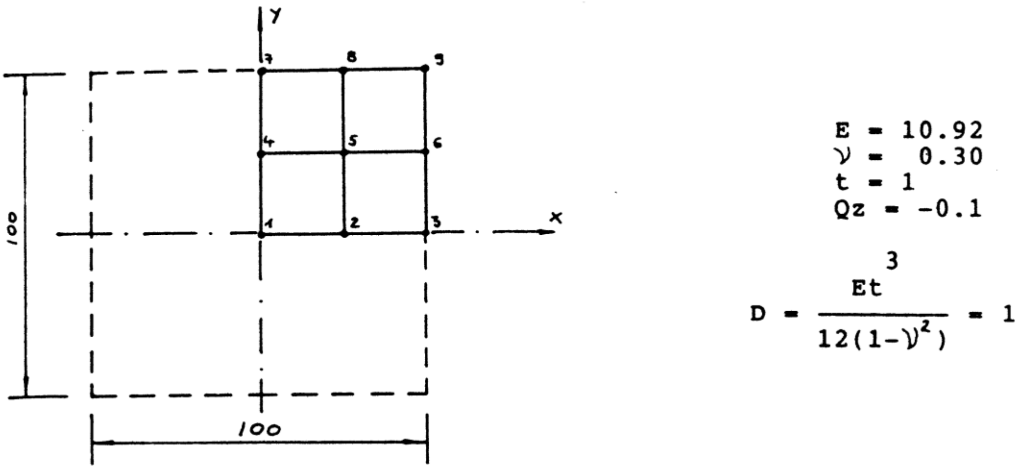


Figure 1. Plate under bending

Solution:

Navier's solution for simply supported rectangular plate is given in reference equations 4.1.61. Using square plate case and performing necessary summations, a simple analytical expression for transverse deflection in the plate centre is obtained

$$W = 40624 \frac{Q_Z L^4}{D} 10^{-7}$$

The following table compares analytical and finite element deflections:

Analytical	UNA	difference (%)
-40624.	-40456.	-0.4

Table 1. Plate Centre Deflections

UNA input file:

```
TITLE = "TV006 - Bending of simply supported isotropic plate"
$
SOL = 1
VER = 7.2
$
SUBCASE = 1
$
DISP  = ALL
FORCE = ALL
$
BEGIN BULK
GRID           1       0      0.      0.      0.       0   12456
GRID           2       0     25.      0.      0.       0    1246
GRID           3       0     50.      0.      0.       0   12346
GRID           4       0      0.     25.      0.       0    1256
GRID           5       0     25.     25.      0.       0     126
GRID           6       0     50.     25.      0.       0   12346
GRID           7       0      0.     50.      0.       0   12356
GRID           8       0     25.     50.      0.       0   12356
GRID           9       0     50.     50.      0.       0  123456
CSHELL4        1       1       1       2       5       4
CSHELL4        2       1       2       3       6       5
CSHELL4        3       1       4       5       8       7
CSHELL4        4       1       5       6       9       8
PSHELL         1       1      1.       1
MAT1           1   10.92             0.3
PLOAD2         1    -0.1       1    THRU       4
ENDDATA
```

Title : Biaxial loading of multilayered membrane

Reference: J.E. Ashton, J.M. Whitney
"Theory Of Laminated Plates"
Technomic, 1970

Description: This example considers a square multilayered membrane which is subjected to biaxial loading. There are total of four plies : two 0 and two 45 degrees. One load case is considered:

Nx = 40. Ny = 8.

Structure is modelled using mesh of four CSHELL4 elements. Geometry, properties, and loading are shown at Figure 1.

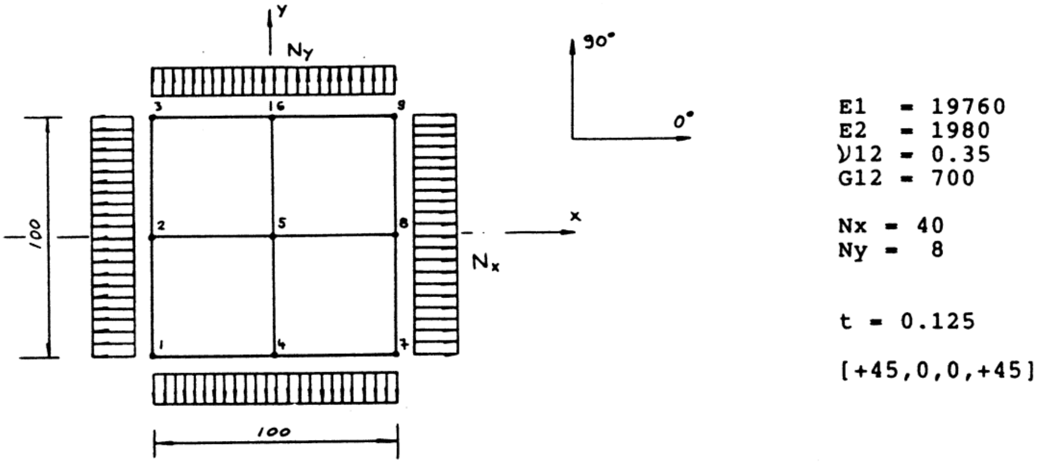


Figure 1. Bi-axially Loaded Laminate

Solution:

Using equations 1.63 and 2.27 from reference, coefficients Aij of A matrix can be defined as

A_{11} = 6639.7 A_{16} = 1125.1
A_{12} = 1463.7 A_{26} = 1125.1
A_{22} = 2139.5 A_{66} = 1462.9

so, it is possible to calculate strains ε_X, ε_Y, γ_{XY} by solving

$$\begin{Bmatrix} N_X \\ N_Y \\ N_{XY} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix} \cdot \begin{Bmatrix} \varepsilon_X \\ \varepsilon_Y \\ \gamma_{XY} \end{Bmatrix}$$

which results in:

$\varepsilon_X = 6.59758\times10^{-3}$, $\varepsilon_Y = 3.18005\times10^{-3}$, $\gamma_{XY} = -7.51987\times10^{-3}$

In-plane stains for 0^0 layers are identical to the above values. Strains for 45^0 layers are calculated using transformation matrix [T], which yields in

$\varepsilon_1 = 1.12888\times10^{-3}$, $\varepsilon_2 = 8.64875\times10^{-3}$, $\gamma_{12} = -3.41753\times10^{-3}$

Using expression 1.63 for stress-strain relation, stresses for 0^0 and 45^0 layers can be calculated. Analytical and finite element values are shown in the following table:

	0^0 layer		45^0 layer	
	Analytical	UNA	Analytical	UNA
σ_X	134.22	134.22	28.65	28.65
σ_Y	11.00	11.00	18.13	18.13
τ_{XY}	-5.26	-5.26	-2.39	-2.39

Table 1. Ply Stresses in Material System

UNA input file:

```
TITLE = "TV007 - Biaxial loading of multilayered membrane"
$
SOL = 1
VER = 7.2
$
PARAM PLATE {1} = 2
PARAM PLATE {2} = 8
$
SUBCASE = 1
$
FORCE = ALL
$
BEGIN BULK
GRID           1       0    -50.    -50.      0.       0   13456
GRID           2       0    -50.      0.      0.       0  123456
GRID           3       0    -50.     50.      0.       0   13456
GRID           4       0      0.    -50.      0.       0    3456
GRID           5       0      0.      0.      0.       0    3456
GRID           6       0      0.     50.      0.       0    3456
GRID           7       0     50.    -50.      0.       0    3456
GRID           8       0     50.      0.      0.       0    3456
GRID           9       0     50.     50.      0.       0    3456
CSHELL4        1       1       1       4       5       2
CSHELL4        2       1       2       5       6       3
CSHELL4        3       1       5       8       9       6
CSHELL4        4       1       4       7       8       5
$
LAYERS         1
+              1    .125    +45.
+              1    .125      0.
+              1    .125      0.
+              1    .125    +45.
$
MAT2           1  19760.   1980.    0.35    700.
$
FORCE          1       1       0      1.  -1000.   -200.      0.
FORCE          1       2       0      1.  -2000.      0.      0.
FORCE          1       3       0      1.  -1000.    200.      0.
FORCE          1       4       0      1.      0.   -400.      0.
FORCE          1       6       0      1.      0.    400.      0.
FORCE          1       7       0      1.   1000.   -200.      0.
FORCE          1       8       0      1.   2000.      0.      0.
FORCE          1       9       0      1.   1000.    200.      0.
ENDDATA
```

Title : Buckling of multilayered simply supported plate

Reference: J.E. Ashton, J.M. Whitney
"Theory Of Laminated Plates"
Technomic, 1970

Description: This example considers buckling of a square multilayered plate. Material is multilayered composite made from 12 plies. Plate is symmetric and specially orthotropic (D_{16} and D_{26} are equal to zero). Two load cases are considered:

1. Nx = -1.

2. Nx = -1. Ny = -0.2

Due to symmetry of structure and boundary conditions, only upper right part is modelled utilizing a mesh of four CSHELL4 elements. Geometry and properties are shown at Figure 1.

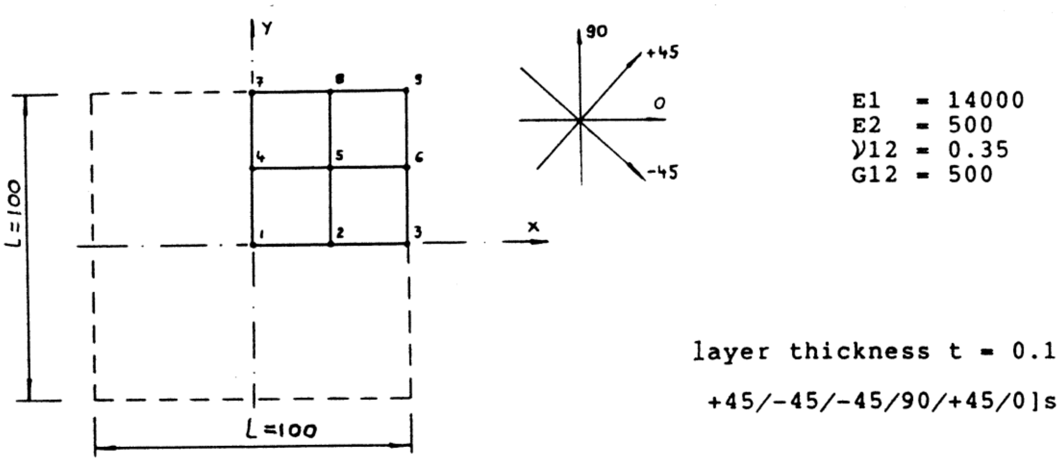


Figure 1. Multilayered Plate

Solution:

Critical buckling load is defined by expression 4-53

$$N_X^C = -\frac{\pi^2\left[D_{11}\left(\frac{m}{a}\right)^2 + 2\left(D_{12} + 2D_{66}\right)\cdot\left(\frac{n}{b}\right)^2 + D_{22}\left(\frac{n}{b}\right)^4\left(\frac{a}{m}\right)^2\right]}{1+\alpha\left(\frac{a}{b}\frac{n}{m}\right)^2}$$

where : a = plate length
: b = plate width
: α = Ny/Nx

Using equation 2.27 from the reference, it is possible to calculate D_{11} - D_{66} stiffness as follows :

D_{11} = 568.3 D_{22} = 731.0
D_{12} = 424.2 D_{66} = 470.9

Finite element buckling load is given by

$$N_X^C = \lambda \min \cdot N_X$$

Analytical and finite element values are shown in the following table

Load Case	UNA		Analytical	Difference
	λmin	N_X^C	N_X^C	(%)
1	4.0281	4.0281	3.9787	1.2
2	3.3576	3.3576	3.3156	1.3

Table 1. Critical Buckling Loads

UNA input file:

```
TITLE = "TV008 - Buckling of multilayered simply sup. plate"
$
SOL = 3
VER = 7.2
$
ECHO MAT  = ON
$
PARAM PLATE {2} = 3
$
SUBCASE = 1
SUBCASE = 2
$
FORCE = ALL
$
BEGIN BULK
GRID           1       0      0.      0.      0.       0   12456
GRID           2       0     25.      0.      0.       0     246
GRID           3       0     50.      0.      0.       0    2346
GRID           4       0      0.     25.      0.       0     156
GRID           5       0     25.     25.      0.       0       6
GRID           6       0     50.     25.      0.       0     346
GRID           7       0      0.     50.      0.       0    1356
GRID           8       0     25.     50.      0.       0     356
GRID           9       0     50.     50.      0.       0    3456
CSHELL4        1       1       1       2       5       4
CSHELL4        2       1       2       3       6       5
CSHELL4        3       1       4       5       8       7
CSHELL4        4       1       5       6       9       8
$
LAYERS         1                                                       SYM
+              1      .1     45.
+              1      .1    -45.
+              1      .1    -45.
+              1      .1     90.
+              1      .1     45.
+              1      .1      0.
$
MAT2           1  14000.    500.    0.35    500.
$
FORCE          1       3       0      1.   -12.5
FORCE          1       6       0      1.   -25.0
FORCE          1       9       0      1.   -12.5
FORCE          2       3       0      1.   -12.5
FORCE          2       6       0      1.   -25.0
FORCE          2       9       0      1.   -12.5    -2.5
FORCE          2       8       0      1.     0.0    -5.0
FORCE          2       7       0      1.     0.0    -2.5
$
ENDDATA
```

Title : Cantilevered beam modelled by solid elements

Description: A cantilevered beam with concentrated load at free end is analysed for stresses. Beam is modelled by three solid HEXA elements. Structure, material data, load and mesh are shown on the following figure:

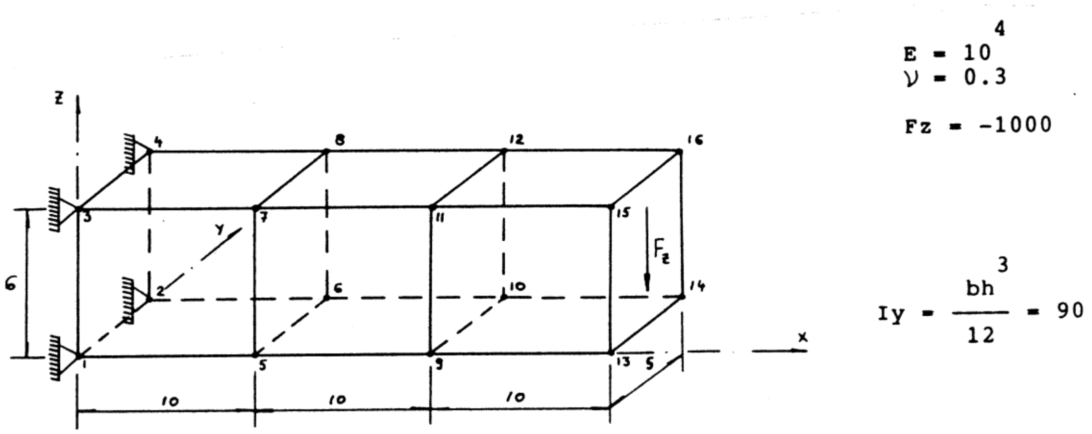


Figure 1. Cantilevered Beam

Solution:

Normal stress due to bending on upper and lower beam surface are given by:

$$\sigma_x = \pm \frac{M_y}{I_y} z$$

The following table compares analytical and finite element stresses:

Element	Analytical	UNA
1	833.33	833.33
2	500.00	500.00
3	166.67	166.67

Table 1. Stress comparison

UNA input file:

```
TITLE = "TV009 Cantilevered beam modelled by solid elements"
$
SOL = 1
VER = 7.2
$
PARAM SOLID {1} = 2
PARAM SOLID {2} = 1
$
SUBCASE = 1
$
FORCE = ALL
$
BEGIN BULK
$
GRID           1       0      0.      0.      0.       0  123456
GRID           2       0      0.      5.      0.       0   13456
GRID           3       0      0.      0.      6.       0    1456
GRID           4       0      0.      5.      6.       0    1456
GRID           5       0     10.      0.      0.       0     456
GRID           6       0     10.      5.      0.       0     456
GRID           7       0     10.      0.      6.       0     456
GRID           8       0     10.      5.      6.       0     456
GRID           9       0     20.      0.      0.       0     456
GRID          10       0     20.      5.      0.       0     456
GRID          11       0     20.      0.      6.       0     456
GRID          12       0     20.      5.      6.       0     456
GRID          13       0     30.      0.      0.       0     456
GRID          14       0     30.      5.      0.       0     456
GRID          15       0     30.      0.      6.       0     456
GRID          16       0     30.      5.      6.       0     456
$
CSOLID         1       1       1       5       6       2       3       7
+              8       4
CSOLID         2       1       5       9      10       6       7      11
+             12       8
CSOLID         3       1       9      13      14      10      11      15
+             16      12
$
FORCE          1      13       0      1.      0.      0.   -250.
FORCE          1      14       0      1.      0.      0.   -250.
FORCE          1      15       0      1.      0.      0.   -250.
FORCE          1      16       0      1.      0.      0.   -250.
$
PSOLID         1       1
$
MAT1           1   1.e+4             .3
$
ENDDATA
```

Title : Simply supported sandwich beam

Reference: E.H. Bruhn
"Analysis and Design of Flight Vehicle Structures"
Tri-State Offset Company, 1973

Description: The simply supported sandwich beam is to be analysed for deflections due to load applied midspan. Sandwich material is modelled with six orthotropic solid elements (CSOLID) for core idealization, and twelve isotropic membrane (CMEMB) elements for skins idealisation. Geometry and loads are given on the following figure:

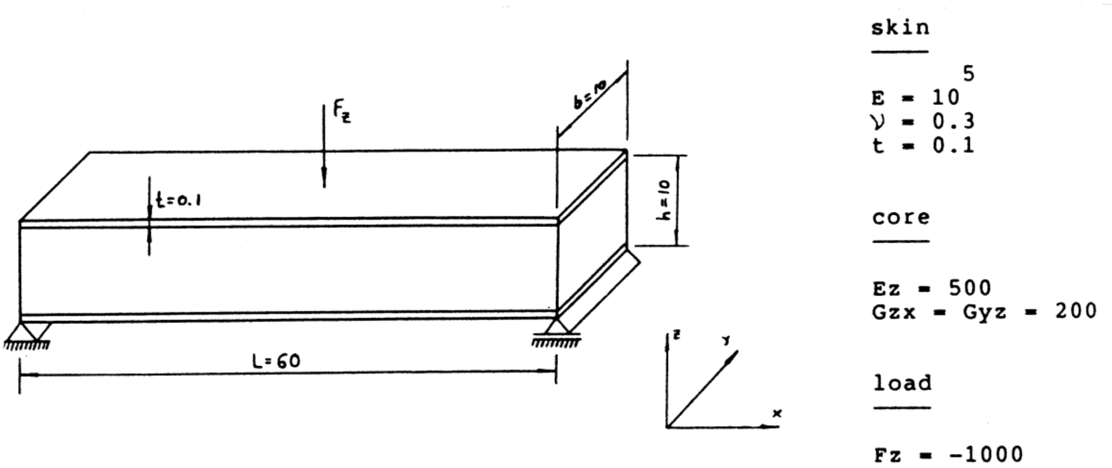


Figure 1. Simply Supported Beam

Solution:

Given reference defines sandwich bending and shear stiffness as

$$D = \frac{Eth^2}{2(1-\nu^2)} \qquad U = hG \qquad \text{(eq. C12.4.1a and C12.4.3)}$$

where G = Gzx = Gyz. Analytical expressions for deflection due to bending and shear at midspan are :

$$U_B = F_Z \frac{L^3(1-\nu^2)}{24Eth^2b} = 0.82 \qquad U_S = \frac{F_Z L}{4bhG} \cdot \frac{\tau_{max}}{\tau_{min}} = 0.75$$

The following table compares analytical and finite element total deflections

Analytical	UNA	Diff. (%)
1.57	1.66	5.7

Table 1.Deflection comparison

Title : Cantilevered box beam

Reference: E.H. Bruhn
"Analysis and Design of Flight Vehicle Structures"
Tri-State Offset Company, 1973
Example problem 15, chapter A 8.10

Description: The doubly symmetric four flange idealized box beam is to be analysed for stresses due to load applied at the point 1 and 2. Flanges areas taper linearly from root to tip, while sheet thickness are constant in each panel. The beam is mounted rigidly at the root, providing full restraint against warping of the root cross section due to any torsional loading.

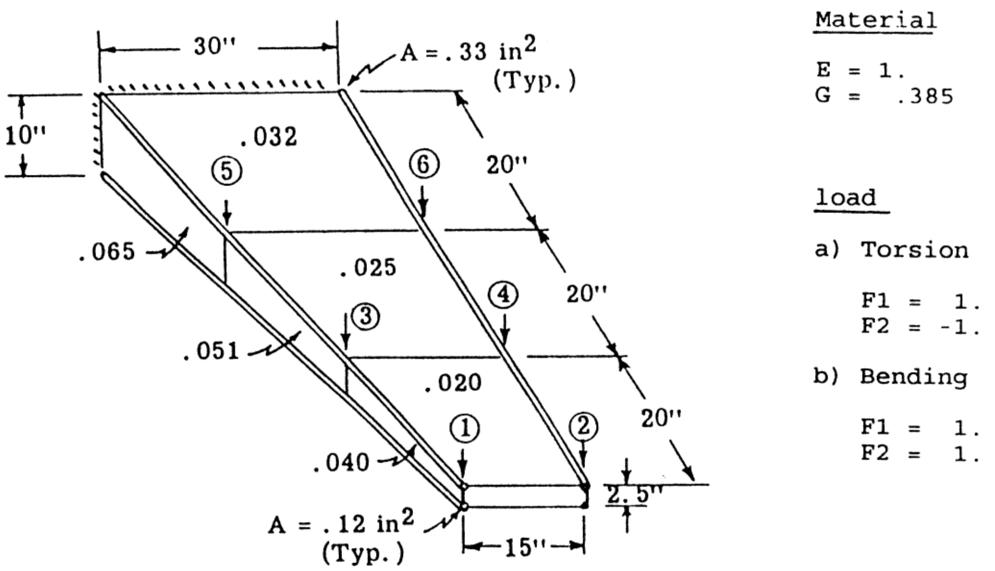


Figure 1. Cantilevered Box Beam

Solution:

Structure is modelled by CROD and CSHEAR elements. Idealization is presented on the following picture. Lateral ribs are omitted for clarity reasons.

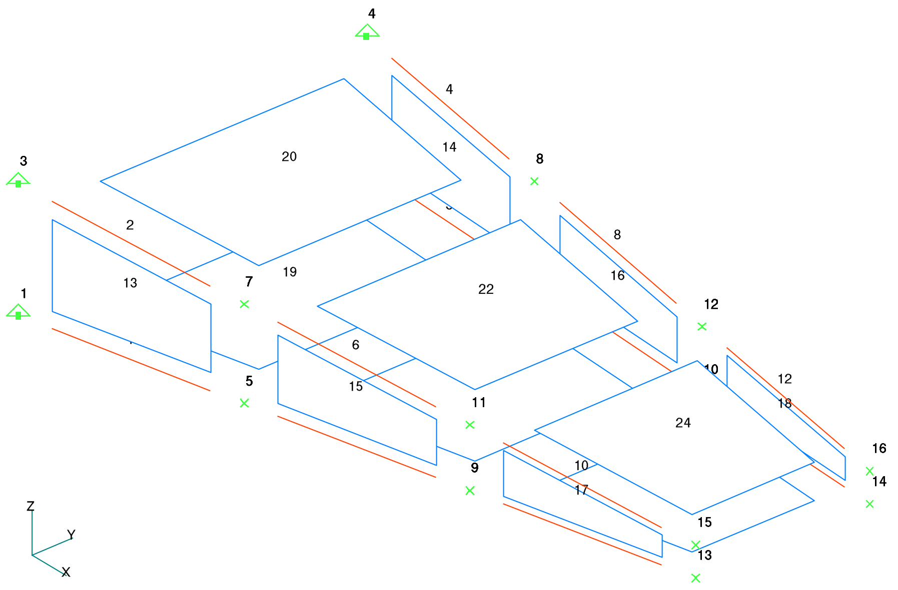


Figure 2. Structure idealization

Analytical and finite element shear flows for the selected panels are enclosed on the following table

Load Case	Shear Panel	Edge Nodes	Analytical	UNA	Diff. (%)
Torsion	13	5-7	0.0464	0.0466	0.5
Torsion	20	7-8	0.0314	0.0319	1.6
Bending	13	5-7	0.0444	0.0444	0.0
Bending	20	7-8	0.	0.	0.0

Table 1. Shear Flows (lbf/in)

Title : Random vibration of a frame structure

Reference: Ray W. Clough, Joseph Penzien
"Dynamics of Structures"
McGraw-Hill Book Co., 1993
Example E23-1, chapter 23-3

Description: A uniform inverted L-shaped member of unit mass per unit length and flexural stiffness EI in its plane, is discretized as shown in Figure 1. The model is subjected to simultaneous stationary random base accelerations $A_x(t)$ and $A_y(t)$ having power spectral and cross-spectral densities given by :

$$\phi_{xx}(\Omega)=S_o \qquad \phi_{yy}(\Omega) = \frac{1}{2} S_o \qquad \phi_{xy}(\Omega) = \phi_{yx}(\Omega = C \cdot S_o$$

Structure is to be analysed for r.m.s. value of base moment M(t).

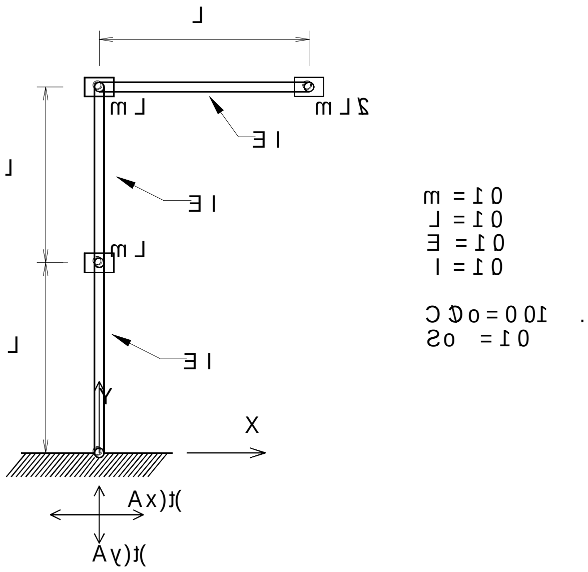


Figure 1. Discrete model of uniform inverted L-shaped member

Solution:

Structure is modelled by CBAR and CMASS elements as shown on the above figure.

In order to apply a harmonic acceleration at base, a large mass is added at structure root. Structure is analysed for natural frequencies and rigid body modes in the plane. Applied harmonic load and large mass produce a unit harmonic accelerations in x and y directions. The following are frequencies in [Hz] as calculated by UNA.

Mode	UNA [Hz]	Analytical [Hz]
1	0.	0.
2	0.	0.
3	0.07072	0.07064
4	0.25502	0.25495
5	0.74395	0.74409

The solution for the root bending moment r.m.s. value is given as:

$$M_{r.m.s.} = \left[\frac{m^2 L^2}{\xi} \sqrt{\frac{EI}{L}} (9.3 + 3.2C) S_o \right]^{1/2}$$

If excitations $A_x(t)$ and $A_y(t)$ are statistically independent, then C=0.0. For a fully statistically correlated excitations C = +/- 0.707107. The following table compares UNA and Analytical r.m.s. responses for all three cases, damping C/Co = 0.01, input power spectral density So = 1.0.

C	UNA $M_{R.M.S.}$	Analytical $M_{R.M.S.}$
0.	30.85	30.50
0.7071	34.46	34.06
-0.7071	26.76	26.46

UNA input file:

```
TITLE  = "TV012 - Random vibration of a frame structure"
$
SOL = 8
MEM = 2
VER = 7.2
$
LAMBDA =  5
SHIFT  = -0.1
FZERO  =  0.001
$
$ Modal damping, frequency set, dynamic load set, random PSD set
$ --------------------------------------------------------------
$
SDAMPING CRIT = 0.01
$
FREQ   = 1
DLOAD  = 1
RANDOM = 1
$
$ Random vibration PSD output
$ ---------------------------
$
PSD ACCE = RANGE   4   4  COMP  1   2
PSD BAR  = RANGE 103 103  COMP 11  11
$
BEGIN BULK
$
$ Grid points
$
GRID           1       0      1.      2.      0.       0     345
GRID           2       0      0.      2.      0.       0     345
GRID           3       0      0.      1.      0.       0     345
GRID           4       0      0.      0.      0.       0    3456
$
$ Elements
$
CBAR         101     100       1       2      0.      1.      0.
CBAR         102     100       2       3     -1.      0.      0.
CBAR         103     100       3       4     -1.      0.      0.
$
CMASS          1       1       0     0.5
CMASS          2       2       0      1.
CMASS          3       3       0      1.
CMASS          4       4       0   1.e+6
```

```
$
$ Properties and materials
$
PBAR         100       1    1.e5      1.      1.      1.
MAT1           1      1.      1.     0.3
$
$ Load sources Ps
$
FORCE        101       4       0   1.e+6      1.      0.      0.
FORCE        102       4       0   1.e+6      0.     -1.      0.
$
$ Dynamic oscillatory load P = Ps x cos (wt).
$
RLOAD          1     101
RLOAD          1     102
$
$ Frequencies to be used in frequency response analysis.
$
FREQ1          1   .0010   .0010    1000
FREQ1          1   .0699   .0001      30
$
$ PSD definition for random analysis. Factor 4 x Pi to convert PSD.
$
$            SID       J       K       X       Y     TID
$
RANDPS         1     101     101 12.5664      0.       1
RANDPS         1     102     102 12.5664      0.       2
RANDPS         1     101     102 12.5664      0.       3
$
TABRND1        1
+             0.      1.      1.      1.
TABRND1        2
+             0.      .5      1.      .5
TABRND1        3
+             0.   .7071      1.   .7071
$
ENDDATA
```