

# Supplemental Manual

## RELEASE 6 USER'S MANUAL

ENGINEERING DYNAMICS, INC.  
2113 38TH STREET  
KENNER, LOUISIANA 70065  
U.S.A.

No part of this document may be  
reproduced in any form, in an  
electronic retrieval system or  
otherwise, without the prior  
written permission of the publisher.

Copyright © 2006 by  
ENGINEERING DYNAMICS, INC.  
Printed in U.S.A.



---

**TABLE OF CONTENTS**

<b>1.0 INTRODUCTION</b>	<b>1-1</b>
<b>2.0 FATIGUE</b>	<b>2-3</b>
<b>2.1 ANALYSIS OPTIONS</b>	<b>2-3</b>
2.1.1 General Options	2-3
2.1.2 In-line Connection Option	2-5
2.1.3 Report Options	2-6
2.1.4 Tubular Connection Redesign Options	2-7
2.1.5 Wave Spreading Options	2-7
<b>2.2 STRESS CONCENTRATION FACTOR</b>	<b>2-8</b>
2.2.1 Program Calculated SCF Values	2-8
2.2.2 User Defined SCFs	2-12
<b>2.3 S-N FAILURE CURVES</b>	<b>2-14</b>
2.3.1 Default S-N Curve	2-14
2.3.2 Overriding the Default S-N curve	2-16
2.3.3 API Thickness Dependent Curve Parameters	2-17
2.3.4 Norwegian Standards S-N Curve Parameters	2-17
2.3.5 British Standards S-N Curve Parameters	2-17
2.3.6 API RP2A WSD Errata and Supplement 2	2-18
<b>2.4 Additional FATIGUE input lines</b>	<b>2-19</b>
<b>3.0 JOINT CAN</b>	<b>3-2</b>
<b>3.1 BASIC OPTIONS</b>	<b>3-2</b>
3.1.1 Overlapping Brace Check	3-2
3.1.2 Effective Thickness of Grouted Elements	3-2
<b>3.2 ANALYSIS TYPE AND CODE</b>	<b>3-4</b>
3.2.1 API Punching Shear Check	3-4
3.2.2 European Punching Shear Checks	3-5
3.2.3 Simplified Fatigue Check	3-5
3.2.4 Earthquake Joint Check	3-6
3.2.5 Simplified and MSL Ultimate Strength Check	3-6
3.2.6 Low Level Earthquake Analysis	3-7
<b>3.3 OUTPUT REPORT SELECTIONS</b>	<b>3-8</b>
3.3.1 Punching Check Report	3-8
3.3.2 Strength Check Report	3-8
3.3.3 Load Path Report	3-8
3.3.4 SCF Report	3-8
3.3.5 Chord Load Transfer Report	3-8
<b>3.4 REDESIGN PARAMETERS</b>	<b>3-8</b>
<b>3.5 OVERRIDING YIELD STRESS</b>	<b>3-9</b>
3.5.1 Specifying a Default Yield Stress	3-9
3.5.2 Changing a Global Yield Stress	3-9
3.5.3 Changing Member Group Yield Stress	3-9
3.5.4 Changing Joint Yield Stress	3-10
<b>3.6 Brace Chord OverRides</b>	<b>3-10</b>

---

---

<b>3.7 Additional JOINT CAN input lines</b>	<b>3-11</b>
<b>4.0 SACS IV</b>	<b>4-3</b>
<b>4.1 ANALYSIS OPTIONS</b>	<b>4-3</b>
<b>4.2 POST PROCESSOR OPTIONS</b>	<b>4-4</b>
4.2.1 Member Check Code	4-4
4.2.2 Member Check Locations	4-4
4.2.3 Output Reports	4-4
4.2.4 Redesign Parameters	4-5
4.2.5 Hydrostatic Collapse Parameters	4-5
4.2.6 Grouping Elements by Unity Check Ratio	4-5
4.2.7 Allowable Stress/Material Factor	4-5
4.2.8 Resistance Factors	4-5
4.2.9 Euro Code Check Options	4-6
4.2.10 Span designation	4-6
<b>4.3 ELEMENT DATA</b>	<b>4-7</b>
4.3.1 Members or Beam Elements	4-7
4.3.2 Plate Elements	4-10
4.3.3 Shell Elements	4-18
4.3.4 Solid Elements	4-20
<b>4.4 Additional SACS IV input lines</b>	<b>4-22</b>



SECTION 1  
***INTRODUCTION***



## 1.0 INTRODUCTION

This manual is intended to supplement the Fatigue, Joint Can and the SACS IV manuals. The manual addresses new features in these modules for the SACS 5.2 Service Pack 2 Release.

Only the main sections that effected by the new features are included in the supplemental manual.



SECTION 2  
***Fatigue***



Fatigue is a post-processing program and requires one or more SACS common solution files as part of its input data. For a discussion on creating the appropriate SACS common solution files, see the previous section. In addition, certain other information is required and the user must select from various options available to execute a fatigue analysis. The following sections detail the information required and options available for executing Spectral, direct deterministic or interpolated deterministic fatigue analysis.

Various fatigue analysis options specified on the FTOPT and FTOPT2 input lines are available.

General and overall options are designated on the FTOPT input line. The design life of the structure (i.e. number of years the structure is to survive) and the life safety factor are specified in columns 8-14 and 22-28, respectively. The life safety factors can be overridden by using the JNTOVR input line to allow the user to define different safety factors at different locations as specified in Table 5.2.5-1 of Errata And Supplement 2 API RP2A WSD 21<sup>st</sup> Edition. For deterministic analysis, the time period over which the specified number of cycles occur is designated in columns 15-21.

For example the following designates a design life of 20 years with a safety factor of 2. Non-tubular members are to be omitted from the analysis.

#### 2.1.1.1 Designating Additional Postfiles

Release 6: Revision 1

The following indicates that 3 additional solution files are to be used for a total of 4 solution files.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT 3 20.0	2.0		SK				

Note: Designate more than 9 additional solution files (10 total) using the letters 'A' for 10 additional (11 total), 'B' for 11 additional (12 total), 'C' for 12 additional (13 total), 'D' for 13 additional (14 total), 'E' for 14 additional (15 total) and 'F' for 15 additional (16 total).

### 2.1.1.2 Checking Grouted Piles

Enter 'G' in column 6 of the FTOPT line if an effective thickness is to be used when calculating fatigue damage for grouted legs. The effective thickness can be determined by specifying '1', '2', '3', '4' or '5' in column 70 on the FTOPT2 line as follows:

Option 1, selected by inputting '1', the effective thickness is based on the moment of inertia of the cross section of the element as follows:

$$t_{eff} = \frac{D_{Leg} - (D_{Leg}^4 - I_{comp} \frac{64}{\pi})^{1/4}}{2}$$

where:  $D_{Leg}$  is the outside diameter of the larger tube (leg)

$I_{comp}$  is the moment of inertia of the composite section

$$I_{comp} = \frac{\pi}{64} [(D_{Leg}^4 + D_{Pile}^4) - (d_{Leg}^4 + d_{Pile}^4)]$$

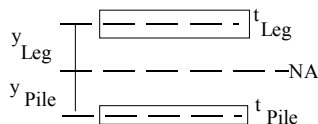
where:  $d_{Leg}$  and  $d_{Pile}$  are the inside diameter of the leg and pile, respectively.

When using option '2', the program uses an effective thickness based on the moment of inertias of the outer tube and the inner tube walls as follows:

$$t_{eff} = (12 \times I_{eff})^{1/3}$$

$$I_{eff} = \frac{1}{12} (t_{Leg}^3 + t_{Pile}^3) + (t_{Leg} \times y_{Leg}^2 + t_{Pile} \times y_{Pile}^2)$$

where  $t$  and  $y$  are defined in the figure below:



Option 3, selected by inputting '3', the effective thickness is based on recommendations in accordance to the Norsok Standards.

Option 4, selected by inputting '4', the effective thickness is based on the RMS average of leg thickness and pile thickness as follows:

$$t_{eff} = \sqrt{t_{Leg}^2 + t_{Pile}^2}$$

where:  $t_{Leg}$  and  $t_{Pile}$  are wall thickness of leg and pile, respectively.

By default, the chord effective thickness limit expressed as a factor of the actual chord thickness is 1.75. Enter the limit in columns 71-74 on the FTOPT2 line.

Option 5 selected by inputting '5', the effective thickness on recommendations in accordance to the Errata and Supplement 2, API RP2A- WSD 21<sup>st</sup> Edition.

### 2.1.1.3 Norsok Axial Stress Linear Variation

Enter 'L' in column 75 of the FTOPT2 line if the axial stress at the points between the crown and saddle are to be determined using a linear variation as stipulated by Norsok Standards.

## 2.1.2 In-line Connection Option

By default, in-line connections, or joints with no braces, are checked as part of the standard fatigue check.

Fatigue damage can be calculated at points of cross-section changes on segmented tubular members (i.e. where no joint exists) by entering 'TI' in columns 68-69 on the FTOPT2 line.

*Note: The Fatigue program calculates SCF's using API cone SCF criteria for cone/cylinder intersections when they are defined as internal segments (not end segments). If the cone is located at either end of the member, a default SCF of 5.0 is used.*

The sample below stipulates that change in cross sections for segmented tubular members are to be checked as in-line connections.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0	SK				
FTOPT2						TI	

### 2.1.3 Report Options

Basic report options are designated on the FTOPT line of the input file. The reporting level; summary, intermediate or full is designated by 'SM', 'IN' or 'FL', respectively, in columns 31-32. Enter 'NE' in columns 46-47 if the input data is not to be printed in the output listing file.

Additional report options may be specified on the FTOPT2 input line. Member summary reports may be generated in either joint order, where members are listed in ascending joint order, or life order, where members are listed in ascending fatigue life order, by specifying 'PT' in columns 10-11 or 12-13, respectively.

A member detail report, containing actual damage may be obtained by inputting 'PT' or percent damage by inputting 'PC' in columns 8-9. Damage at all points around the circumference of the connection on both the brace and chord side are reported.

*Note:* The member detail report can be quite voluminous. Also, the member detail report option overrides the report options input on the FTOPT input line. See Section 3.1.2.1 for member report override features.

A report containing the ranges that certain SCF parameters are valid may be output by inputting 'VC' in columns 14-15 on the FTOPT2 input line.

Plate summary reports may be generated in either plate order, where plate names are listed in ascending order, life order, where members are listed in ascending fatigue life order, or by both plate order and life order, by specifying 'PO', 'LO' or 'PL' in columns 18-19, respectively.

The following requests summary reports with connections listed in damage order. The SCF validity report is also requested.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0	SM	SK			
FTOPT2	PT	VC					

### 2.1.3.1 Overriding Joint Report Options

The report print options for individual joints or groups of joints may be overridden using the JNTSEL input line. The print level, 'SM', 'FL', 'PC' or 'DG' is designated in columns 11-12. If a range of joints is specified, 'R' is entered in column 15. The individual joint names or joint ranges to be overridden are stipulated in columns 16-75.

If the joints specified are to be removed from the analysis report, 'RM' is entered in columns 8-9. Any report option specified in columns 11-12 is ignored.

The report level for a joint may also be overridden using the JNTOVR line. Enter either 'SM', 'FL', 'PC' or 'DG' in columns 29-30 for a summary, full print with stress and damage, full print with damage and percent of total damage, or diagnostic print, respectively.

### 2.1.4 Tubular Connection Redesign Options

The program has the ability to redesign tubular connections based on fatigue damage.

Redesign options are designated on the FTOPT input line. 'EX' is entered in columns 49-50 if tubular chords and/or braces that do not satisfy the design life requirements are to be redesigned. The chord and brace may be redesigned based on the option specified in column 29 and 30, respectively. The following options are available for both the chord and brace:

'O'	Constant outside diameter	'M'	Constant mean diameter
'I'	Constant inside diameter	'T'	Constant thickness

The following additional brace options are available to allow increasing diameter when the brace D/t minimum has been reached:

'A' Constant outside diameter  
 'B' Constant mean diameter  
 'C' Constant inside diameter

The maximum number of thickness changes to be attempted, the thickness increment and the minimum D/t ratio are designated in columns 51-53, 54-59 and 60-65, respectively.

The following designates that redesign is to be performed. The number of thickness steps is 50 and the thickness increment is 0.125.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0	SK	EX	50	.125	

A redesign progress report may be output by entering 'RD' in columns 16-17 on the FTOPT2 input line. If a new SACS input file is to be created reflecting the redesigned member groups, enter 'UP' in columns 24-25 on the FTOPT2 input line.

### 2.1.5 Wave Spreading Options

Wave spreading options for spectral analysis are stipulated on the FTOPT2 input line.

Enter 'WS' in columns 26-27 to invoke wave spreading or 'AP' to invoke wave spreading with response function averaging (API energy approach).

If the structure is symmetrical about the XZ plane and transfer function data was developed for 0 through 180 degree approach directions only, enter 'XZ' in columns 28-29. Likewise, if the structure is symmetrical about the YZ plane, and the transfer function data was developed for 90 through -90 degree approach directions only, enter 'YZ' in columns 30-31.

The wave spreading is assumed to be defined by the distribution  $\cos(\theta)^n$ . The power of the cosine distribution,  $n$ , is designated in columns 32-36.

*Note: When using wave spreading, transfer functions must be developed using the same waves (i.e. Height and period) for each direction.*

## 2.2 STRESS CONCENTRATION FACTOR

Local stresses at connection discontinuities "hot spot stresses" can be up to 20 times higher than the calculated nominal stresses. Several options are available in Fatigue to account for this local increase of stresses. "Hot spot" stresses are calculated as the product of nominal stresses and suitably determined stress concentration factors (SCF's). For tubular connections, the program can calculate SCFs automatically or can use user supplied values. User supplied values are required for non-tubular connections.

### 2.2.1 Program Calculated SCF Values

The program has the ability to calculate SCFs for tubular connections using various techniques and methods. The following sections detail features available to use program calculated values.

#### 2.2.1.1 Specifying the Default SCF Method

The Fatigue program can calculate stress concentration factors for tubular connections using numerous methods including the most generally accepted methods used in the offshore industry. The default method to be used for all connection types is designated in columns 78-80 on the FTOPT input line. For tubular joints the following SCF options are available:

1. 'PSH' - Punching shear SCF's do not distinguish among joint types (K, T, KT, Y, X), but are applied uniformly independent of joint classification. For this option all brace SCF's are taken as 6.0 and chord SCF's are given by:

$$SCF = \frac{Y \sin \theta}{K_a Q_\beta} (4.0 + 0.67\gamma)$$



where:

$$Q_{\beta} = \begin{cases} 1.0 & \text{for } \beta \leq 0.6 \\ \frac{0.3}{\beta(1-0.833\beta)} & \text{for } \beta > 0.6 \end{cases}$$

$K_a$ ,  $\theta$ ,  $Y$ ,  $\beta$  and  $\gamma$  are as defined in API-RP2A.

2. 'PS2' - A variation of the punching shear SCF's discussed above with brace SCF's equal to 5.0 instead of 6.0.
3. 'KAW' - SCF's suggested by KUANG, et al for K and KT joints and those suggested by Wordsworth and Smedley for T, Y and X joints.
4. 'DNV' - SCF's suggested by Kuang, et al for all joint types except X joints, for which Wordsworth and Smedley SCF's are used. Brace SCF's are reduced by a technique suggested by Marshall (Marshall reduction factor) except that the reduction factor must be greater than or equal to 0.8. Furthermore a minimum SCF of 2.5 is imposed on all members.
5. 'MSH' - SCF's suggested by Marshall.
6. 'UEG' - SCF's suggested by UEG.
7. 'EFT' - SCF's suggested by Efthymiou (Model C options).
8. 'EFR' - SCF's suggested by MSL for single-sided weld connections.
9. 'SAF' - SCF's suggested by Smedley and Fischer.
10. 'COJ' - SCF's calculated using COJAC methodology including joint classification.
11. 'ALK' - uses Alpha Kellog SCF's.
12. 'USR' - user defined SCF's. Requires SCF and/or SCF2 input line.
13. 'CA1' - alpha will be computed with the factor before gamma in the equation of Figure C4.3.1-2(a) being 0.6 and the maximum alpha value limited to 2.4 according to Table C5.1-1 of API-RP2A 21st edition.
14. 'CA2' - alpha will be computed with the factor before gamma in the equation of Figure C4.3.1-2(a) being 1.6 and the maximum alpha value limited to 3.8 according to Table C5.1-1 of API-RP2A 21st edition.

*Note: When using the effective thickness for grouted connections, the effective thickness is used when calculating  $\gamma$  and the actual chord thickness is used when calculating  $\tau$ .*

For in-line connections or segmented tubular section changes, the default SCF method is designated in columns 51-53 on the FTOPT2 line. The following in-line SCF options are available:

1. 'AWS' - SCF's suggested by American Welding Society.
2. 'DNV' - SCF's suggested by Det Norske Veritas.
3. 'BS' - SCF's suggested by British Standards.
4. 'DE' - SCF's suggested by Department of Energy.
5. 'NS1' - SCF's suggested by Norsok Standard Equation C.2.7.
6. 'NS2' - SCF's suggested by Norsok Standard Equation C.2.10.

7. 'USR' - user defined SCF's. Requires SCF and/or SCF2 input line.  
 For Norsok in-line SCF methods, the percent out of roundness and maximum out of roundness are specified on the FTOPT3 line in columns 8-12 and 13-17, respectively. The options for the 'EFT' SCF calculations can be assigned on the EFTOPT input line when the validity ranged do not meet the requirements as discussed in API RP2A WSD Supplement 2. This line should be defined after the FTOPT2 line in the fatigue input file. To calculate the SCF's based upon the actual geometric parameters 'ACT' should be entered in columns 8-10 of the EFTOPT input line. To use the limiting geometric parameters 'LIN' should be entered. To use the maximum SCF's from the 'ACT' and 'LIN' options then 'MAX' should be entered in columns 8-10 of the EFTOPT line. The short can reduction may be turned off by entering 'NSC' on columns 11-13 of the EFTOPT input line.  
 The following designates that Efthymiou Model C options is the default method to calculate SCFs for tubular connections. In-line connection SCFs will be determined using American Welding Society recommendations.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0	SK			EFT	
FTOPT2				AWS		TI	

### 2.2.1.2 Overriding Default Method

The default method may be overridden for connection types or for particular joints. The hierarchy for SCF method overrides is as follows:

1. Program default method is overridden by method specified for the joint (JNTSCF).
2. Joint method is overridden by method specified by connection type (SCFSEL).

#### 2.2.1.2.1 Overriding by Joint Name

The default method used to calculate SCFs may also be designated for a particular joint using the JNTOVR line. Specify the joint name in columns 8-11 and the method to be used in columns 25-27.

The following stipulates that the UEG SCFs are to be used for joint 105.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
JNTOVR	105	UEG					

The method to be used to calculate the default stress concentration factors may be designated for each connection type using the SCFSEL input line. Any of the available SCF options may be designated for 'T&Y', 'K', 'KT' and/or 'X' type connections individually.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
SCFSEL	ALK						

By default, SCFs are determined based on the joint geometry. If SCF's are to be determined based on the load distribution through the connection, enter 'LP' in columns 76-77 on the FTOPT line.

The minimum gap to be used for SCF calculation is specified in columns 66-71 of the FTOPT line.

By default, the average chord length,  $(L_1 + L_2)/2$ , is used as the chord length for SCF calculations, where  $L_1$  and  $L_2$  are the lengths of the chord members on each side of the joint. The total length,  $L_1 + L_2$ , is used when 'T' is specified in column 75 of the FTOPT line. Enter 'W' in column 75 if the chord length to be used is to be taken as  $4(L_1 \times L_2)/(L_1 + L_2)$ .

When having the program calculate SCFs automatically, the user may input maximum and/or minimum values for SCF's using the SCFLM input line. When specifying SCF limits, the SCF limit flags, 'MX' in columns 40-41 and/or 'MN' in columns 42-43 on the FTOPT input line, must be designated.

The following specifies that Alpha-Kellog SCFs are to be used for tubular connections by default. The values calculated by the program shall not exceed 5. nor be less than 2.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0	MXMNSK				ALK
SCFLM	5.0	2.0					

*Note:* The SCF limit features should not be used when the 'DNV' SCF option is selected.

### 2.2.1.7 Plates, Shells and Non-tubular Members

For non-tubular members, plates and shells, the program default values are 5.0 for beam elements and 1.0 for plate and shell elements. The user may specify SCF's to be used by the program for non-tubular beam, plate and shell elements. See user defined SCFs in the ensuing sections.

## 2.2.2 User Defined SCFs

The user may optionally input the stress concentration factors to be used to determine hot spot stresses.

### 2.2.2.1 Specifying User Defined Default Values

If no default SCF method is applicable, the user may define default SCF values using the SCF and SCF2 input lines.

*Note:* When using user defined SCF's designated on the SCF and SCF2 input lines, the 'USR' SCF option must be specified in columns 78-80 on the FTOPT input line.

The overall SCF for tubular members may be specified in columns 6-10. If an overall value is specified, any values entered in columns 36-75 are overridden. The SCF to be used for wide flanges, compact wide flanges, prismatic sections, boxes and/or plate elements are designated in columns 11-15, 16-20, 21-25, 26-30 and 31-35, respectively. The SCF's to be used for angle, tee and channel type sections may be stipulated on the SCF2 input line in columns 6-10, 11-15 and 16-20, respectively.

For tubular elements, detailed stress concentration factors may be specified for axial and bending loads and for the brace or chord. The brace side SCF for axial loads in the crown and saddle may be designated independently in columns 36-40 and 41-45, respectively. The brace SCF's for in-plane and out-of-plane bending are specified in columns 46-50 and 51-55, respectively. The chord side SCF for axial loads in the crown and saddle may also be designated independently in columns 56-60 and 61-65, respectively. The chord SCF's for in-plane and out-of-plane bending may be specified in columns 66-70 and 71-

75, respectively.

The following designates that user defined default SCF values are to be used. 4.0 is to be used for all wide flange, box, tee, angle and channel sections. For tubular sections, the values for both brace and chord axial load crown and saddle are 3.0 and 3.5. The defaults (5.0) are applicable for bending.

	1	2	3	4	5	6	7	8
	123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0		MXMNSK				USR
SCF	4.0	4.0	4.0	4.0	3.0	3.5	3.0	3.5
SCF2	4.0	4.0	4.0					

### 2.2.2.2 Overriding Beam Element Default Values

The user can specify an SCF value used to override any value either calculated by the program or specified as a default value using the SCF and SCF2 lines.

Beam element values can be overridden for specific joints, member groups, individual members or specific brace-chord connections. The hierarchy of SCF overrides is as follows:

1. Program calculated or user input values are overridden by SCF overrides specified for the joint (JNTSCF).
2. Joint SCF overrides are overridden by applicable group SCF overrides (GRPSCF).
3. Group SCF overrides are overridden by SCF overrides designated for the member (MEMSCF).
4. Member SCF overrides are overridden by SCF overrides designated for the connection (CONSCF or CONSWF).

*Note: In all cases, the SCF's are limited by the limiting values specified on the SCFLM input line.*

#### 2.2.2.2.1 Overriding by Joint

The SCF for an individual joint or range of joints may be overridden using the JNTSCF input line. The SCF is designated in columns 7-14. If a range of joints is specified, 'R' is entered in column 15. The individual joint names or joint ranges to be overridden are stipulated in columns 16-75.

#### 2.2.2.2.2 Overriding by Group

The SCF for all members assigned to a particular member group may be overridden using the GRPSCF input line. The SCF is designated in columns 7-14. The member groups to which the SCF is to be applied are specified in columns 16-74.

#### 2.2.2.2.3 Overriding by Member

The SCF for a particular member may be overridden using the MEMSCF input line. The SCF is designated in columns 7-14. The members to which the SCF is to be applied are specified in columns 15-78.

#### 2.2.2.2.4 Overriding by Connection

Values may be overridden for a particular connection using the CONSCF line for tubular connections and CONSWF line for wide flange and plate girder connections.

#### 2.2.2.3 Overriding Plate Element Default Values

The SCF for any plate or plate group may be overridden.

##### 2.2.2.3.1 Overriding by Group

The SCF for all plates assigned to a particular plate group may be overridden using the PGRPOV input line. The SCF is designated in columns 9-13. The groups to which the SCF is to be applied are specified in columns 20-74.

##### 2.2.2.3.2 Overriding by Plate

The SCF for a particular plate may be overridden using the PLTOVR input line. The SCF is designated in columns 19-25 and the plate name is specified in columns 8-11.

## 2.3 S-N FAILURE CURVES

For a particular stress range, there exist a theoretical number of cycles  $N_F(s)$ , at which fatigue failure may occur. The relationship between this number of allowable cycles and the stress range is usually expressed as an S-N curve.

*Note: Because published S-N curves are for virgin material, fatigue life is determined assuming that there are no inclusions or surface cracks exceeding the allowable based on fracture mechanics analysis. The existence of cracks greater than this value will reduce the predicted life of the connection.*

The user may specify S-N data or select a curve included in the program.

### 2.3.1 Default S-N Curve

Numerous S-N curves including ones recommended by AWS, Norwegian Standards, British Standards and API are built into the program. The user may also define the relationship between allowable cycles and stress range by specifying a user defined curve.

### 2.3.1.1 Intrinsic Curves

Several S-N curves are intrinsic or built into the program. The default S-N curve option is designated in columns 33-35 on the FTOPT input line. The following S-N options are available:

1. 'AWS' - American Welding Society X-X Curve
2. 'API' - API X Curve
3. 'APP' - API X' (X Prime) Curve
4. 'AXX' - API thickness dependent X Curve
5. 'AXP' - API thickness dependent X' (X Prime) Curve
6. 'NS3' - Norwegian Standards (NS3472E) including material thickness variations (requires SNT1 input line)
7. 'Baa' - British Standard (BS6235) curves where aa is as follows:  
     'C ' for C curve   'D ' for D curve   'E ' for E curve   'F ' for F curve  
     'F2' for F2 curve   'G ' for G curve   'W ' for W curve
8. 'HTP' - HSE T Prime curve with thickness correction only
9. 'HPP' - HSE P curve with thickness correction only
10. 'TTP' - HSE T Prime for tubular and P curve for other sections (with thickness correction only)
11. 'USR' - User defined S-N curve using S-N input line
12. 'xxx' - Advanced user defined curve using SN-USR line where xxx is the label supplied on the SN-USR line.
13. 'Nbb' - Norsok standards where bb is as follows:  
     'B1' for B1 curve   'B2' for B2 curve   'C ' for C curve   'C1' for C1  
     'C2' for C2 curve   'D ' for D curve   'E ' for E curve   'F ' for F curve  
     'F1' for F1 curve   'F3' for F3 curve   'G ' for G curve   'W1' for W1  
     'W2' for W2 curve   'T ' for T curve
14. API-RP2A WSD Supplement 2 as follows:  
     'WJT' for welded joint with no weld improvement.  
     'WJ1' for welded joint with weld profile as described in section 11.1.3.d of API- RP2A WSD 21<sup>st</sup> Edition.  
     'WJ2' for welded joint with weld toe burr grind.  
     'WJ3' for welded joint with hammer peening  
     'CJT' for cast joint

Enter the mudline elevation and water depth in columns (54-67) of the FTOPT2 line for the HSE T prime, P and API RP2A 21 supplement 2 'WJ\_' S-N curves.

*Note: Typically, brace, chord and in-line connections are adjusted when selecting thickness dependent S-N criteria. However, for APP or APX options, thickness dependency is considered for braces only (i.e. chord and in-line connections are not adjusted for thickness).*

The following designates that the API X Prime curve is to be used as the default S-N curve.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
FTOPT	20.0	2.0	APP	SK			EFT

### 2.3.1.2 User Defined S-N Curves

The user may define an S-N curve to be used by designating 'USR' in columns 33-35 on the FTOPT input line. The points defining the S-N curve are specified on the S-N input line. The curve is defined by inputting stress range and number of allowable cycles for up to twelve points. If the last two points have the same stress range level, this level is assumed to be the endurance limit.

Advanced user defined curves may be defined using the SN-USR line. The points defining the S-N curve are specified in columns 25-80. The minimum thickness and thickness correction power are specified in columns 13-18 and 19-24, respectively. Enter 'B' in column 11 if the thickness correction applied to both the brace and chord is to be determined solely from the brace thickness.

If the last value defines the endurance limit, input 'E' in column 12. The name or ID of the curve is entered in columns 8-10. To make an advanced user defined curve the default, enter the curve ID in columns 33-35 on the FTOPT line.

*Note: User defined S-N curve data must be input in order of increasing number of allowable cycles.*

## 2.3.2 Overriding the Default S-N curve

The default S-N curve may be overridden for beam and plate elements.

### 2.3.2.1 Beam Elements

The default S-N curve may be overridden for beam elements by specifying S-N override for particular joints or connection. The hierarchy for S-N overrides is as follows:

1. Program default curve is overridden by method specified for the joint (JNTOVR).
2. Joint curve overridden by method specified for a connection (CONSCF or CONSWF).

#### 2.3.2.1.1 Overriding by Joint Name

The default curve used to calculate allowable cycles also be designated for a particular joint using the JNTOVR line. Specify the joint name in columns 8-11 and the curve to be used in columns 13-15.

The following stipulates that the API X curve is to be used for joint 105.



1	2	3	4	5	6	7	8
123456789012345678901234567890123456789012345678901234567890123456789012345678901234567890							
JNTOVR	105	API	UEG				

#### 2.3.2.1.2 Overriding by Connection

The curve to use may be overridden for a particular connection using the CONSCF line for tubular connections and CONSWF line for wide flange and plate girder connections.

#### 2.3.2.2 Plate Elements

The S-N curve to use for any plate or plate group may be overridden.

##### 2.3.2.2.1 Overriding by Group

The S-N curve for all plates assigned to a particular plate group may be overridden using the PGRPOV input line. The S-N curve is designated in columns 14-16. The groups to which the overrides are to be applied are specified in columns 20-74.

##### 2.3.2.2.2 Overriding by Plate

The S-N curve to use for a particular plate may be overridden using the PLTOVR input line. The curve is designated in columns 16-18 and the plate name is specified in columns 8-11.

### 2.3.3 API Thickness Dependent Curve Parameters

When selecting either the API X or X' thickness dependent S-N curve, the elevations defining the lower level and upper level of the splash zone are designated on the FTOPT2 input line in columns 37-43 and 44-50, respectively. The splash zone area has no endurance limit defined.

### 2.3.4 Norwegian Standards S-N Curve Parameters

The Norwegian Standard (NS3472) S-N 'T' curve is designated by 'NS3' in columns 33-35 on the FTOPT input line.

In addition, the SNT1 input line is required. The waterline elevation must be specified in columns 7-14. If the 'T' curve thickness dependent parameters default values are to be used, no other input is required. The thickness dependent parameters may be overridden in columns 15-62. Additional curve data may be specified using up to twelve SNT2 input lines.

### 2.3.5 British Standards S-N Curve Parameters

When using the BS6235 curves, the dividing elevation between the air and sea water endurance limits is designated in columns 44-50 on the FTOPT2 input line.

---

**2.3.6 API RP2A WSD Errata and Supplement 2**

The API Supplement 2 S-N curves can be used by designating the following options in columns 33-35 on the FTOPT input line:

WJT : to use the S-N curve without any weld improvement

WJ1 : to use the S-N curve with weld profile conforming to section 11.1.3.d in.

API- RP2A WSD 21<sup>st</sup> edition code of practice.

WJ2: to use the S-N curve conforming to a weld profile with weld toe burr grind.

WJ3: to use the S-N curve for a welded joint with hammer peening

CJT: to use the S-N curve designated for cast joints.

## 2.4 ADDITIONAL FATIGUE INPUT LINES

## FATIGUE OPTION LINE PART 1

COLUMNS	COMMENTARY	COLUMNS	COMMENTARY
GENERAL	THE FATIGUE PROGRAM PERFORMS AN EVALUATION OF THE TOTAL ENVIRONMENTAL FATIGUE DAMAGE FOR THE LIFE OF A STRUCTURE USING THE S-N PALMGREN-MINER ACCUMULATED DAMAGE METHOD OR THE PARIS EQUATION CRACK GROWTH METHOD. AN APPROXIMATE ANALYSIS BASED ON PUNCHING SHEAR (AS SHOWN IN THE API-RP2A) CAN BE PERFORMED USING THE "JOINT CAN" PROGRAM.	(31-32)	ENTER 'SM' FOR A SUMMARY, 'IN' FOR INTERMEDIATE, OR 'FL' FOR A FULL REPORT.
( 1- 5 )	ENTER 'FTOPT'.	(33-35)	ENTER THE SOURCE OF THE S-N (STRESS RANGE VERSUS CYCLES) CURVE:  'AWS' - AMERICAN WELDING SOCIETY X-X CURVE. 'API' - AMERICAN PETROLEUM INSTITUTE X CURVE. 'APP' - AMERICAN PETROLEUM INSTITUTE X PRIME CURVE. 'USR' - SUPPLIED BY USER ON 'S-N' LINE. 'NS3' - NS3472 THICKNESS DEPENDENT CURVE - 1984. 'AXX' - API X CURVE WITH THICKNESS CORRECTION. 'AXP' - API X PRIME CURVE WITH THICKNESS CORRECTION. 'BC', 'BD', 'BE', 'BF', 'BF2', 'BG' OR 'BW' FOR THE BRITISH STANDARD (BS 6235) C, D, E, F, F2, G OR W CURVES. 'HTP' - HSE T PRIME WITH THICKNESS CORRECTION ONLY. 'HPP' - HSE P CURVE WITH THICKNESS CORRECTION ONLY. 'TPP' - HSE T PRIME AND P CURVE WITH THICKNESS CORRECTION. T PRIME FOR TUBULAR INTERSECTIONS, P FOR ALL OTHERS. 'XXX' - USER-SPECIFIED LABEL FROM THE 'SN-USR' INPUT DATA. 'NB1', 'NB2', 'NC ', 'NC1', 'NC2', 'ND ', 'NE ', 'NF ', 'NF1', 'NF3', 'NG ', 'NW1', 'NW2', 'NW3', 'NT ' FOR THE NORSOK S-N CURVES B1, B2, C, C1, C2, D, E, F, F1, F3, G, W1, W2, W3, AND T RESPECTIVELY IN AIR AND IN SEAWATER WITH CATHODIC PROTECTION. 'WJT' - API SUPP. 2 WELDED JOINT WITH NO WELD IMPROVEMENT 'WJ1' - API SUPP. 2 WELDED JOINT WITH PROFILE PER 11.1.3D 'WJ2' - API SUPP. 2 WELDED JOINT WITH WELD TOE BURR GRIND 'WJ3' - API SUPP. 2 WELDED JOINT WITH HAMMER PEENING 'CJT' - API SUPP. 2 CAST JOINT ENTER 'CRG' TO USE THE PARIS EQUATION CRACK GROWTH DAMAGE CALCULATION METHOD (S-N DATA IS NOT REQUIRED).
( 6 )	ENTER 'G' IF THE PILE IS GROUTED, AND THE PILE DIAMETER AND WALL THICKNESS ARE CONTAINED ON THE SACS IV 'SECT' LINE.	(36-37)	ENTER 'SK' TO SUPPRESS FATIGUE ANALYSIS OF MEMBERS AND CANS.
( 7 )	ENTER THE ADDITIONAL NUMBER OF POSTFILES THAT ARE TO BE INCLUDED IN THIS ANALYSIS. THE LIMIT IS 3000 LOAD CASES.	(38-39)	ENTER 'SK' TO SUPPRESS FATIGUE ANALYSIS OF PLATE ELEMENTS.
( 8-14)	ENTER THE NUMBER OF YEARS THE STRUCTURE IS TO SURVIVE.		
(15-21)	FOR A DETERMINISTIC ANALYSIS ENTER THE TIME PERIOD OVER WHICH THE NUMBER OF CYCLES ON THE 'FTCASE' LINES ACCUMULATES. FOR A SPECTRAL ANALYSIS THIS FIELD IS LEFT BLANK.		
(22-28)	ENTER THE DESIRED FACTOR OF SAFETY ON THE LIFE OF THE STRUCTURE.		
(29-30)	IF REDESIGN IS TO BE DONE, SELECT FROM THE FOLLOWING FOR REDESIGNING THE CHORD AND BRACE RESPECTIVELY: 'O' - CONSTANT OUTSIDE DIAMETER. 'M' - CONSTANT MEAN DIAMETER. 'I' - CONSTANT INSIDE DIAMETER. 'T' - CONSTANT WALL THICKNESS.  FOR THE BRACE ONLY, THE FOLLOWING ADDITIONAL OPTIONS ARE AVAILABLE THAT ALLOW THE BRACE DIAMETER TO BE INCREASED WHEN THE BRACE D/T MINIMUM HAS BEEN REACHED. 'A' - CONSTANT OUTSIDE DIAMETER. 'B' - CONSTANT MEAN DIAMETER. 'C' - CONSTANT INSIDE DIAMETER.		

LINE LABEL	GROUT PILE	POST FILES	OVERALL PARAMETERS			REDESIGN		GENERAL OPTIONS				SEE FTOPT LINE PART 2
			DESIGN LIFE	E TIME PERIOD	LIFE SAFETY FACTOR	CHORD OPTION	BRACE OPTION	REPORT OPTIONS	S-N OR CRACK	FATIGUE ANALYSIS		
<b>FTOPT</b>										MEMBER	PLATE	
1-5	6	7	8-14	15-21	22-28	29	30	31-32	33-35	36-37	38-39	40-----80
DEFAULT				1.0	1.0			'SM'	'USR'			
ENGLISH			YEARS	YEARS								
METRIC			YEARS	YEARS								

## FATIGUE OPTION LINE PART 2

COLUMNS	COMMENTARY	COLUMNS	COMMENTARY
GENERAL	THE FATIGUE PROGRAM PERFORMS AN EVALUATION OF THE TOTAL ENVIRONMENTAL FATIGUE DAMAGE FOR THE LIFE OF A STRUCTURE USING THE S-N PALMGREN-MINER ACCUMULATED DAMAGE METHOD OR THE PARIS EQUATION CRACK GROWTH METHOD. AN APPROXIMATE ANALYSIS BASED ON PUNCHING SHEAR (AS SHOWN IN THE API-RP2A) CAN BE PERFORMED USING THE "JOINT CAN" PROGRAM.	(60-65)	ENTER THE MINIMUM D/T RATIO PERMITTED IN THE REDESIGN.
( 1-39)	SEE DESCRIPTION ON PRECEDING PAGE.	(66-71)	ENTER THE MINIMUM GAP PERMITTED BETWEEN BRACES AT THE FACE OF THE CHORD (A NEGATIVE VALUE MEANS AN OVERLAP OF THAT AMOUNT). THIS ONLY AFFECTS STRESS CONCENTRATION CALCULATIONS.
(40-41)	ENTER 'MX' IF AN UPPER LIMIT IS PRESCRIBED ON SCF'S ('SCFLM' LINE). IF 'DNV' IS ENTERED IN COLUMNS 78-80 THIS SHOULD BE LEFT BLANK.	( 74 )	ENTER 'K' IF THE EFFECTIVE LENGTH FACTORS (K FACTORS) ARE TO BE USED TO CALCULATE THE CHORD LENGTHS FOR SCF CALCULATIONS.
(42-43)	ENTER 'MN' IF A LOWER LIMIT IS PRESCRIBED ON SCF'S ('SCFLM' LINE). IF 'DNV' IS ENTERED IN COLUMNS 78-80 THIS SHOULD BE LEFT BLANK.	( 75 )	ENTER 'T' IF THE TOTAL LENGTH (JOINT TO JOINT) IS TO BE USED FOR SCF CALCULATIONS. THE DEFAULT IS THE AVERAGE LENGTH. ENTER 'W' FOR $L=4*L1*L2/(L1+L2)$ WHERE L1 AND L2 ARE THE CHORD LENGTHS ON EITHER SIDE OF THE JOINT.
(44-45)	ENTER 'SK' TO SUPPRESS FATIGUE ANALYSIS OF NON-TUBULAR MEMBERS.	(76-77)	ENTER 'LP' TO HAVE SCF'S DETERMINED BY LOAD PATH.
(46-47)	ENTER 'NE' TO SUPPRESS PRINTING OF THE INPUT ECHO.	(78-80)	ENTER THE BASIS ON WHICH STRESS CONCENTRATION FACTORS ARE CALCULATED FROM THE FOLLOWING OPTIONS. 'ALK' - ALPHA-KELLOGG SCF'S. 'COJ' - COJAC METHODOLOGY INCLUDING JOINT CLASSIFICATION. 'DNV' - DET NORSKE VERITAS CRITERION WITH KUANG SCF'S AND MODIFIED MARSHALL REDUCTION FACTORS. 'EFR' - EFTHYMIOU SCF CORRECTED FOR SINGLE-SIDED WELDS. 'EFT' - EFTHYMIOU (MODEL C OPTIONS) - CORRECTED. 'KAW' - SCF'S FROM WORDSWORTH ET AL. FOR T, Y, AND X JOINTS, SCF'S FROM KUANG ET AL. FOR K AND KT JOINTS. 'MSH' - MARSHALL SCF'S. 'PSH' - PUNCHING SHEAR ANALYSIS WITH BRACE SCF = 6.0. 'PS2' - PUNCHING SHEAR ANALYSIS WITH BRACE SCF = 5.0. 'SAF' - SMEDLEY AND FISHER SCF'S. 'UEG' - UNDERWATER ENGINEERING GROUP SCF'S. 'USR' - SCF'S AS INPUT BY THE USER. 'CA1' - COMPUTED ALPHA METHOD 1 'CA2' - COMPUTED ALPHA METHOD 2
(49-50)	ENTER 'EX' FOR CHORD AND/OR BRACE THICKNESSES TO BE REDESIGNED IF THEY DO NOT SATISFY THE DESIGN LIFE REQUIREMENTS.		
(51-53)	ENTER THE MAXIMUM NUMBER OF THICKNESS CHANGES TO BE PERMITTED IN THE REDESIGN PROCESS.		
(54-59)	ENTER THE THICKNESS INCREMENT (DECREMENT) TO BE APPLIED AT EACH REDESIGN.		

LINE LABEL	SEE FTOPT LINE PART 1	GENERAL OPTIONS				TUBULAR REDESIGN PARAMETERS				SCF PARAMETERS				
		SCF LIMITS		NON-TUBE SKIP OPTION	NO INPUT ECHO	REDESIGN OPTION	THICK-NESS STEPS ALLOWED	THICK-NESS STEP	MINIMUM>D T>RATIO	MINIMUM GAP	K FACTOR OPTION	SCF LENGTH OPTION	LOAD PATH SCF OPTION	SCF OPTION
		MAX	MIN											
<b>FTOPT</b>														
1-5	6 -----39	40-41	42-43	44-45	46-47	49-50	51-53	54-59	60-65	66-71	74	75	76-77	78-80
DEFAULT							100	.125IN	20.0	-1000.IN				'PSH'
ENGLISH								IN		IN				
METRIC								CM		CM				

## FATIGUE OPTION LINE 2 PART 1

COLUMNS	COMMENTARY	COLUMNS	COMMENTARY
GENERAL	THE FATIGUE PROGRAM PERFORMS AN EVALUATION OF THE PERFORMANCE OF A STRUCTURE WITH RESPECT TO FATIGUE FAILURE BASED ON THE PALMGREN-MINER ACCUMULATED DAMAGE HYPOTHESIS.	(22-23)	ENTER 'BB' IF A BROAD-BAND ANALYSIS APPROACH IS DESIRED. THIS IS DESIRABLE IF THE WAVE SPECTRUM IS NOT NARROW-BANDED, AS IS THE CASE FOR OCHI-HUBBLE OR DOUBLE JONSWAP SPECTRA. DEFAULT IS NARROW BAND.
( 1- 6)	ENTER 'FTOPT2'.	(24-25)	THE FATIGUE PROGRAM HAS THE OPTION OF UPDATING THE SACS IV GEOMETRY FILE REFLECTING THE FATIGUE REDESIGN. TO ACTIVATE THIS OPTION, ENTER 'UP' IN THESE COLUMNS.
( 8- 9)	MEMBER DETAIL REPORT OPTION. THIS REPORT CONTAINS THE DAMAGES AT ALL POINTS AROUND THE CIRCUMFERENCE FOR ALL BRACE/CHORD INTERSECTIONS. THE USER CAN SELECT SPECIFIC JOINTS FOR THIS REPORT (SEE JNTSEL): 'PC' - DAMAGES EXPRESSED AS A PERCENT OF THE TOTAL DAMAGE. 'PT' - ACTUAL DAMAGES ARE TO BE LISTED. NOTE: AN ENTRY INTO THIS FIELD WILL OVERRIDE THE REPORT OPTION ON THE 'FTOPT' RECORD.	(26-27)	SELECT ONE OF THE FOLLOWING TO INCLUDE WAVE SPREADING: 'WS' FOR TRANSFER FUNCTION AVERAGING. 'AP' FOR RESPONSE FUNCTION AVERAGING (API ENERGY APPROACH). NOTE: THIS OPTION IS ONLY AVAILABLE FOR SPECTRAL ANALYSIS.
(10-11)	ENTER 'PT' FOR A MEMBER SUMMARY REPORT IN JOINT ORDER. THE FATIGUE DAMAGES ARE REPORTED IN ASCENDING JOINT ORDER.	(28-29)	IF THE STRUCTURE IS SYMMETRICAL, AN APPROXIMATION USING SYMMETRY CAN BE SELECTED FROM THE FOLLOWING: (NOTE: ANGLES ARE MEASURED FROM X TOWARD Y) 'XZ' - STRUCTURE IS SYMMETRICAL ABOUT THE X-Z PLANE AND TRANSFER FUNCTIONS ARE INPUT ONLY FROM ZERO TO 180.0 DEGREES. 'YZ' - STRUCTURE IS SYMMETRICAL ABOUT THE Y-Z PLANE AND TRANSFER FUNCTIONS ARE INPUT ONLY FROM -90.0 TO +90.0 DEGREES. 'XY' - STRUCTURE IS SYMMETRICAL ABOUT THE X-Z AND X-Y PLANES AND TRANSFER FUNCTIONS ARE ONLY INPUT FROM ZERO TO 90.0 DEGREES.
(12-13)	ENTER 'PT' FOR A MEMBER SUMMARY REPORT IN LIFE ORDER. THE FATIGUE DAMAGES ARE REPORTED IN ASCENDING LIFE ORDER.	(30-31)	SELECT THE OPTION USED TO SELECT THE UNDEFINED TRANSFER FUNCTIONS: 'MI' - MIRRORED ABOUT THE PLANE(S) OF SYMMETRY (DEFAULT). 'OP' - USE THE OPPOSING TRANSFER FUNCTION (180 DEGREES).
(14-15)	ENTER 'VC' IF THE WARNING MESSAGES ABOUT THE VALID RANGES OF SCF PARAMETERS ARE TO BE LISTED.	(32-36)	ENTER THE POWER OF THE COSINE DISTRIBUTION FUNCTION. THE WAVE SPREADING IS A COS(THETA)**N DISTRIBUTION.
(16-17)	ENTER 'RD' IF THE JOINT REDESIGN PROCEDURE IS TO BE INCLUDED IN THE OUTPUT LISTING. THE CHORD AND BRACE DIAMETERS AND THICKNESSES ALONG WITH THE DAMAGES ARE LISTED FOR EACH STEP DURING THE REDESIGN PROCESS.	(37-80)	SEE FATIGUE OPTION LINE 2 PART 2.
(18-19)	PLATE SUMMARY REPORT OPTIONS: 'PO' - SUMMARY REPORT IN PLATE ORDER. 'LO' - SUMMARY REPORT IN LIFE ORDER. 'PL' - SUMMARY REPORT IN PLATE ORDER AND LIFE ORDER.		
(20-21)	ENTER 'PV' FOR POSTVUE FATIGUE LIFE PLOTS. POSTVUE DATABASE MUST BE CREATED FIRST.		

LINE LABEL	REPORT OPTIONS							SPECTRUM ANALYSIS OPTION	UPDATE SACS FILE	WAVE SPREADING				SEE FTOPT2 LINE PART 2
	MEMBER DETAIL REPORT	BER Y REPORT JOINT ORDER	MEMBER SUMMARY REPORT LIFE ORDER	SCF VALIDITY RANGE CHECK	JOINT REDESIGN PROGRESS	PLATE SUMMARY REPORT OPTION	PLOT RESULTS			WAVE SPREADING OPTION	SYMMETRY PLANE OPTION	SYMMETRY LOADING OPTION	COSINE POWER	
<b>FTOPT2</b>														
1-6	8-9	10-11	12-13	14-15	16-17	18-19	20-21	22-23	24-25	26-27	28-29	30-31	32-36	37 ----- 80
DEFAULT						'PO'						'MI'		
ENGLISH														
METRIC														

## FATIGUE OPTION LINE 2 PART 2

COLUMNS	COMMENTARY
GENERAL	THE FATIGUE PROGRAM PERFORMS AN EVALUATION OF THE PERFORMANCE OF A STRUCTURE WITH RESPECT TO FATIGUE FAILURE BASED ON THE PALMGREN-MINER ACCUMULATED DAMAGE HYPOTHESIS.
( 8-36)	SEE FATIGUE OPTION LINE 2 PART 1.
(37-43)	ENTER THE ELEVATION FOR THE LOWER LIMIT OF THE ZONE FOR THE 'AXX' AND 'AXP' S-N DATA OPTIONS. SPLASH ZONE DEFINES THE REGION WHERE NO ENDURANCE LIMIT IS ALLOWED.
(44-50)	ENTER THE ELEVATION DEFINING THE UPPER LIMIT OF THE SPLASH ZONE. THIS IS ALSO USED AS THE DIVIDING ELEVATION BETWEEN THE AIR AND SEA WATER ENDURANCE LIMITS FOR THE BS F2 S-N DATA.
(51-53)	SELECT FROM THE FOLLOWING SCF OPTION FOR TUBULAR INLINE CONNECTIONS: 'AWS' - AMERICAN WELDING SOCIETY. 'DNV' - DET NORSKE VERITAS. 'BS ' - BRITISH STANDARDS. 'DE ' - DEPARTMENT OF ENERGY. 'NS1' - NORSOK EQ. C.2.7. 'NS2' - NORSOK EQ. C.2.10. 'USR' - USER-DEFINED VALUE.

COLUMNS	COMMENTARY
(54-67)	ENTER THE MUDLINE ELEVATION AND WATER DEPTH FOR THE HSE T PRIME, P AND API RP2A 21 SUPPLEMENT 2 'WJ_' CURVES.
(68-69)	ENTER 'TI' IF FATIGUE ANALYSIS IS TO BE PERFORMED AT SECTION CHANGES ALONG TUBULAR MEMBERS.
( 70 )	USE EFFECTIVE THICKNESS FOR GROUT. ENTER '1' FOR EFFECTIVE THICKNESS BASED ON THE COMPOSITE SECTION MOMENT OF INERTIA, '2' FOR EFFECTIVE THICKNESS BASED ON MOMENT OF INERTIAS OF THE TWO WALLS OR '4' FOR EFFECTIVE THICKNESS BASED ON RMS AVERAGE THICKNESS. ENTER '3' FOR EFFECTIVE THICKNESS BASED ON NORSOK STANDARDS. ENTER '5' FOR EFFECTIVE THICKNESS CHORD THICKNESS CALCULATION BASED UPON API RP2A WSD SUPPLEMENT 2 (NOTE: 'G' NEEDS TO BE DEFINED IN COLUMN 6 OF FTOPT LINE). THE EFFECTIVE THICKNESS RATIO IN COLUMNS 71-74 WILL BE IGNORED FOR THIS OPTION. THE SCF'S OF GROUTED CHORD JOINT WILL BE SET TO 1.5
(71-74)	ENTER THE EFFECTIVE THICKNESS RATIO LIMIT FOR OPTIONS '1' OR '2'. THE DEFAULT RATIO LIMIT IS 1.75 IF COLUMN 70 IS '2' OR BLANK. FOR OPTION '1', A VALUE MUST BE INPUT.
( 75 )	ENTER 'L' IF THE AXIAL STRESS AT THE POINTS BETWEEN THE CROWN AND SADDLE IS TO BE CALCULATED AS A LINEAR VARIATION AS REQUIRED BY THE NORSOK STANDARDS.
( 76 )	ENTER 'I' TO INPUT A DAMAGE FILE FROM A PREVIOUS RUN TO BE ADDED TO THE DAMAGE CALCULATED, 'O' TO OUTPUT A DAMAGE FILE OR 'B' TO INPUT DAMAGE FROM A PREVIOUS RUN AND OUT A NEW DAMAGE FILE.

LINE LABEL	SEE FTOPT2 LINE PART 1	SPLASH ZONE		INLINE TUB. SCF	WATER LEVEL		TUB INLINE CHECK	EFF. THICK		AXIAL STRESS OPTION	ACCUMULATE DAMAGE
		LOWER LEVEL	UPPER LEVEL		MUDLINE ELEV.	WATER DEPTH		OPTION	RATIO LIMIT		
<b>FTOPT2</b>											
1 - 6	8 -----36	37 - 43	44 - 50	51 - 53	54 - 60	61 - 67	68 - 69	70	71 - 74	75	76
DEFAULT								2			
ENGLISH		FT	FT		FT	FT					
METRIC		M	M		M	M					

## EFTHYMIU SCF OPTIONS

COLUMNS	COMMENTARY
GENERAL	THIS DATA ENABLES THE USER TO SELECT OPTIONS TO BE USED FOR THE EFTHYMIU SCF CALCULATIONS. ENTER 'EFTOPT'.
( 8-10)	SELECT THE OPTION TO BE USED WHEN THE SCF PARAMETER(S) ARE OUTSIDE THE VALID RANGE. 'ACT' - USE THE ACTUAL PARAMETER VALUES 'LIM' - USE THE LIMITING PARAMETER VALUES 'MAX' - USE THE MAXIMUM SCF FROM EITHER THE ACTUAL OR LIMITING
(11-13)	ENTER 'NSC' IF THE SHORT CHORD CORRECTION IS NOT TO BE USED.

LINE LABEL	EXCEED VALID. CHECK OPTION	SHORT CHORD CORRECT. OPTION	LEAVE BLANK
<b>EFTOPT</b>			
1 - 6	8 - 10	11 - 13	14 ----- 80
DEFAULT	ACT		



## JOINT OVERRIDE LINES

COLUMNS	COMMENTARY	COLUMNS	COMMENTARY
GENERAL	THIS DATA ENABLES THE USER TO OVERRIDE FATIGUE PARAMETERS ON A JOINT BY JOINT BASIS. ANY ITEM LEFT BLANK WILL DEFAULT TO THE OVERALL FATIGUE INPUT PARAMETERS OR THE 'SCFSEL' INPUT. THESE RECORDS MAY BE ENTERED IN ANY ORDER.	(25-27)	ENTER THE BASIS ON WHICH STRESS CONCENTRATION FACTORS ARE CALCULATED FROM THE FOLLOWING OPTIONS: 'ALK' - ALPHA-KELLOGG SCF'S. 'COJ' - COJAC METHODOLOGY INCLUDING JOINT CLASSIFICATION. 'DNV' - DET NORSKE VERITAS CRITERION WITH KUANG SCF'S AND MODIFIED MARSHALL REDUCTION FACTORS. 'EFR' - EFTHYMIU SCF CORRECTED FOR SINGLE-SIDED WELDS. 'EFT' - EFTHYMIU (MODEL C OPTIONS) - CORRECTED. 'KAW' - SCF'S FROM WORDSWORTH ET AL. FOR T, Y, AND X JOINTS, SCF'S FROM KUANG ET AL. FOR K AND KT JOINTS. 'MSH' - MARSHALL SCF'S. 'PSH' - PUNCHING SHEAR ANALYSIS WITH BRACE SCF = 6.0. 'PS2' - PUNCHING SHEAR ANALYSIS WITH BRACE SCF = 5.0. 'SAF' - SMEDLEY AND FISHER SCF'S. 'UEG' - UNDERWATER ENGINEERING GROUP SCF'S. 'USR' - SCF'S AS INPUT BY THE USER.
( 1- 6 )	ENTER 'JNTOVR'.		
( 8-11 )	ENTER THE JOINT NAME FOR THIS OVERRIDE.		
(13-15)	SELECT FROM THE FOLLOWING S-N DATA OPTIONS: 'AWS' - AMERICAN WELDING SOCIETY X-X CURVE. 'API' - AMERICAN PETROLEUM INSTITUTE X CURVE. 'APP' - AMERICAN PETROLEUM INSTITUTE X PRIME CURVE. 'USR' - SUPPLIED BY USER ON USER-DEFINED S-N DATA. 'NS3' - NS3472 THICKNESS DEPENDENT CURVE - 1984. 'AXX' - API X CURVE WITH THICKNESS CORRECTION. 'AXP' - API X PRIME CURVE WITH THICKNESS CORRECTION. 'BC', 'BD', 'BE', 'BF', 'BF2', 'BG' OR 'BW' FOR THE BRITISH STANDARD (BS 6235) C, D, E, F, F2, G OR W CURVES. 'HTP' - HSE T PRIME WITH THICKNESS CORRECTION. 'HPP' - HSE P CURVE WITH THICKNESS CORRECTION ONLY. 'TPP' - HSE T PRIME AND P CURVE WITH THICKNESS CORRECTION. T PRIME FOR TUBULAR INTERSECTIONS, P FOR ALL OTHERS.	(29-30)	SELECT FROM THE FOLLOWING PRINT LEVELS FOR THIS JOINT: 'SM' - SUMMARY PRINT. 'FL' - FULL PRINT WITH STRESS AND DAMAGE. 'PC' - FULL PRINT WITH DAMAGE AND PERCENT DAMAGE. 'DG' - DIAGNOSTIC PRINT.
( 17 )	ENTER THE EFFECTIVE CHORD LENGTH CALCULATION OPTION: 'A' - AVERAGE LENGTHS FROM BOTH SIDES. 'T' - TOTAL LENGTHS FROM BOTH SIDES. 'W' - WEIGHTED AVERAGE FROM BOTH SIDES (4*L1*L2/(L1+L2)). 'N' - NO CHORD CONSIDERED. TUBES CHECK AT ENDS ONLY.	(32-37)	ENTER THE CHORD CAN THICKNESS OVERRIDE. THIS WILL OVERRIDE THE THICKNESS OF THE CHORD JOINT CAN FOR FATIGUE CALCULATIONS.
(18-23)	ENTER THE EFFECTIVE CHORD LENGTH. IF ENTERED, THIS WILL OVERRIDE THE CALCULATED EFFECTIVE CHORD LENGTH.	(38-43)	ENTER THE SAFETY FACTOR OVERRIDE

LINE LABEL	JOINT NAME	S-N DATA OPTION	EFFECTIVE CHORD LENGTH CALCULATION OPTION	EFFECTIVE CHORD LENGTH	SCF OPTION	JOINT PRINT OPTION	CHORD THICKNESS OVERRIDE	SAFETY FACTOR OVERRIDE	LEAVE BLANK
JNTOVR									
1 - 6	8 - 11	13 - 15	17	18 - 23	25 - 27	29 - 30	32 - 37	38 - 43	44 ----- 80
DEFAULT									
ENGLISH				FT			IN		
METRIC				M			CM		

Section 3  
***Joint Can***

### 3.0 JOINT CAN

The Joint Can program requires a SACS common solution file containing member internal loads and a Joint Can input file for punching shear, effective strength, simplified fatigue analysis, earthquake punching check and ultimate strength check. The Joint Can input file allows the user to specify basic analysis options, designate the analysis type and code to use and override various properties.

### 3.1 BASIC OPTIONS

Basic Joint Can options are specified on the JCNOPT line.

Enter the units in columns 12-13. Enter the minimum and maximum gap to be used for 'K' joints in columns 20-25 and 26-31.

*Note: Negative value for minimum or maximum gap indicates an overlapped joint.*

#### 3.1.1 Overlapping Brace Check

Enter 'B' in column 32 if overlapping braces are to be checked to ensure that the axial load may be transferred directly through one brace to another via their common weld.

*Note: Overlapping braces are members with a negative gap.*

##### 3.1.1.1 Weld Allowable Stress

By default, the allowable stress for weld material is assumed to be the same as the connection steel. The weld allowable stress used for brace on brace check may be specified using the WELD line.

Specify the allowable stress in columns 7-14. The following specifies an allowable of 70.0 ksi.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
JCNOPT	API	EN				B	
WELD	70.0						

#### 3.1.2 Effective Thickness of Grouted Elements

By default, the thickness of the outside tubular (leg) is used as the chord thickness when analyzing the capacity of a grouted connection. The effective thickness of grouted elements may be determined based on the properties of both the outer and inner tubular members and used for the analysis and redesign of grouted connections. Enter one of the following effective thickness options in column 33:

Option 1, selected by inputting '1', the effective thickness is based on the moment of

$$t_{eff} = \frac{D_{leg} - (D_{leg}^4 - I_{comp} \frac{64}{\pi})^{1/4}}{2}$$

inertia of the cross section of the element as follows:

where:  $D_{leg}$  is the outside diameter of the larger tube (leg)

$I_{comp}$  is the moment of inertia of the composite section

$$I_{comp} = \frac{\pi}{64} [(D_{leg}^4 + D_{pile}^4) - (d_{leg}^4 + d_{pile}^4)]$$

where:  $d_{leg}$  and  $d_{pile}$  are the inside diameter of the leg and pile, respectively

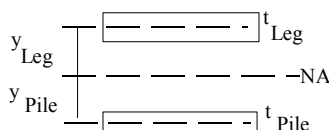
Option 2 uses the moment of inertias of the walls instead of the composite section

$$t_{eff} = (12 \times I_{eff})^{1/3}$$

$$I_{eff} = \frac{1}{12} (t_{leg}^3 + t_{pile}^3) + (t_{leg} \times y_{leg}^2 + t_{pile} \times y_{pile}^2)$$

moment of inertia and is selected by specifying '2' in column 33.

where  $t$  and  $y$  are defined in the figure below:



Option 3 uses the sum of the square root of the squares of the leg and pile thickness and is selected by specifying '3' in column 33.

### 3.1.2.1 Effective Thickness Limit

A chord effective thickness limit expressed as a factor of the actual chord thickness may be specified in columns 76-79 on the JCNOPT input line. The default limit is 1.75.

The following designates that option 1 is to be used for grouted elements and that the effective thickness limit is 2.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
JCNOPT	LRFDEN		1				2.0

### 3.1.2.2 Allowable Punching Shear Stress Limit

By default, the allowable punching shear stress for API codes is limited to the allowable shear stress in the chord. Enter 'N' in column 51 if the allowable punching shear stress is not to be limited.

By default, when calculating the allowable punching stress factor (equation 6.56) for Norsok codes, L is set to the larger of D/4 or 30cm, enter 'L' if the actual modeled length from the crown to the end of the can is to be used.

## 3.2 ANALYSIS TYPE AND CODE

The Joint Can analysis option is designated in columns 8-11 on the JCNOPT line. Various types of analyses are available by designating the appropriate option.

### 3.2.1 API Punching Shear Check

For standard Working Stress Design punching shear check per API, select one of the following options:

1. 'AP21' - Errata and Supplement 2 API RP2A WSD 21<sup>st</sup> Edition
2. 'API' - API 20th Edition
3. 'AP91' - API 19th Edition
4. 'AP84' - API 15th Edition
5. 'AP83' - API 13th Edition Supplement
6. 'AP80' - API 13th Edition
7. 'LG' - Linear global analysis based on API 21st Edition Section 17 criteria

For Ultimate Strength punching check per API, select:

1. 'LRFD' - API LRFD 1st Edition

#### 3.2.1.1 Overriding LRFD Resistance Factors

The default resistance factors used in the API LRFD punching check may be overridden by the user using the RSFAC line. The following overrides the resistance factor for T&Y joints.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
JCNOPT LRFDEN							
RSFAC 0.85 0.90 0.90 0.90							

### 3.2.2 European Punching Shear Checks

The program supports various other punching shear analyses and code check options as follows:

1. 'NPD' - NPD 1977 Edition
2. 'NP94' - NPD 1994 Edition
3. 'NP90' - NPD 1990 Edition
4. 'DNV' - DNV 1977 Edition
5. 'DN83' - DNV 1983 Edition
6. 'DOC' - Danish 1984 Edition
7. 'NS' - Norsok Standard 1998

### 3.2.3 Simplified Fatigue Check

The API Simplified Fatigue analysis is invoked by specifying one of the following in columns 8-11.

1. 'FTG' - API 20th Edition
2. 'FT91' - API 19th Edition
3. 'FT84' - API 16th Edition
4. 'FT82' - API 13th Edition

The appropriate load cases containing the reference level wave should be specified on the LCSEL input line.

The load path dependant SCF's are calculated automatically based on the option input into columns 37-39 on the FATIGUE line. The water depth, water line member elevation, fatigue life and weld classification should be specified in columns 9-16, 17-24, 27-30 and 33-36, respectively, on the FATIGUE input line.

The following shows the input for simplified fatigue using API 20th Edition. Load cases 'SF00', 'SF45' and 'SF90' contain reference level waves used to calculate fatigue stress. The water depth is 150.0 feet, the water line elevation is -20 and design life is 15 years.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
JCNOPT FTG EN							
LCSEL		SF00 SF45 SF90					
FATIGUE 150.		-20.		15		API	

### 3.2.4 Earthquake Joint Check

The program can check joint can capacity due to combined earthquake and static stresses per API guidelines. Specifying 'EQK' for Working Stress, 'EQ21' for Errata and Supplement 2 API RP2A WSD 21<sup>st</sup> Edition or 'EQLR' for LRFD code.

Joint Can is executed after the earthquake and static stresses are combined using the STCMB option in Dynamic Response or the Combine program. Only load cases created specifically for joint check by using the 'PRSC' or 'PRST' option should be specified on the LCSEL line of the Joint Can input file.

For example, the following designates that an API LRFD earthquake check is to be performed for load cases 3 and 4.

1	2	3	4	5	6	7	8
1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890
JCNOPT EQLREN							
LCSEL		3	4				

### 3.2.5 Simplified and MSL Ultimate Strength Check

Simplified ultimate strength check and MSL ultimate strength check analysis may be performed by specifying 'SUS' or 'MSL', respectively, in columns 8-11 of the JCNOPT line.

For MSL check, additional input including the Qu option, ultimate tension value and reassessment values option must be designated on the JCNOPT line. Enter 'C' or 'M' in column 36 for characteristic Qu factor or mean strength Qu factor, respectively. Enter 'U' in column 36 for ultimate tension values and/or 'R' in column 37 for reassessment values.

#### 3.2.5.1 Overriding MSL Assessment Factors

The default assessment factors used in the MSL ultimate strength check may be overridden using the GMFAC line. The following overrides the gamma factors for axial and in-plane bending.

1	2	3	4	5	6	7	8
1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890
JCNOPT MSL EN				CUR			
GMFAC 0.95 0.95							

### 3.2.5.2 Selecting Members

By default all members are considered unless members are specified on the MSLC line. When using the MSLC line, only those members specified are considered for the ultimate strength analysis.

### 3.2.5.3 Designating Initial Load Cases

The first load case in each direction can be specified using the INITLC line.

*Note: The INITLC line is not required if the analysis contains only one wave direction.*

## 3.2.6 Low Level Earthquake Analysis

For low level earthquake loads, analysis may use API WSD (working stress design) or API LRFD (load and resistance factor design). API WSD is specified by putting 'LLEW' in columns 8-11 of the JCNOPT line; API LRFD design is specified by putting 'LLEL' in columns 8-11 of the JCNOPT line. For low level earthquake analysis per API, the user must input rare intense earthquake data in the dynamic response input file. The resulting data must be combined so that load cases 1 and 2 are the rare intense seismic loads and load case 3 contains the dead loads. The dead load case, 3, used in the low level earthquake analysis is specified using the 'DLOAD' line, where '3' is entered in columns 7-10. The following input specifies low level earthquake analysis with API WSD is to be used, with load case 1 and 2 having a 70% increase in allowable stress.

	1	2	3	4	5	6	7	8
	1234567890123456789012345678901234567890123456789012345678901234567890							
	JCNOPT LLEWEN							
LCSEL		1	2					
AMOD								
AMOD	1	1.7	2	1.7				
DLOAD	3							
END								



### 3.3 OUTPUT REPORT SELECTIONS

Output reports are designated in columns 56-69 on the JCNOPT line.

#### 3.3.1 Punching Check Report

Enter one of the following report levels in columns 56-57 for reporting punching check results:

- 'FL' Print results for all load cases for each joint
- 'UC' Print only joints with UC greater than UC limit specified
- 'MX' Print only results for critical load case for each joint
- 'RD' Print results for all load cases for each joint including redesign iterations

*Note: If 'UC' is selected, enter the UC limit in columns 58-61.*

#### 3.3.2 Strength Check Report

Enter 'PT' in columns 62-63 to receive a strength check report. This reports the strength of the connection using 50% of the effective member strength.

#### 3.3.3 Load Path Report

The load path report details the connection classification for each load case and is activated by entering 'PT' in columns 64-65.

#### 3.3.4 SCF Report

The SCFs used for simplified fatigue analysis may be printed by specifying 'PT' in columns 66-67.

#### 3.3.5 Chord Load Transfer Report

The Joint Can program can check to ensure that chords resist general collapse per API specifications when load is transferred across. Enter 'PT' in columns 68-69 to receive the Chord Load Transfer Report.

### 3.4 REDESIGN PARAMETERS

Redesign parameters are designated on the JCNOPT line in columns 38-50.

By default redesign performed by the Joint Can program. Specify 'N' in column 38 to eliminate redesign or 'A' to allow only thickness increases during redesign.

Specify the chord redesign option in columns 39-40. Enter 'OD' if chord outside diameter is to be changed (ie. constant ID), 'ID' if chord inner diameter is to be changed (ie. constant OD) or 'TC' if the thickness is to remain constant when diameter is changed. Designate the thickness and diameter increments in columns 41-45 and 46-50, respectively.

The following sample stipulates that redesign is to be performed allowing only thickness increases using 0.125 increments. The inside diameter is to vary .

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
JCNOPT	API	EN					
				AID	0.125		

### 3.5 OVERRIDING YIELD STRESS

By default, the yield stress specified in the model is used for punching analyses. The yield stress used for joint punching analysis purposes may be modified in several ways in the Joint Can input file.

#### 3.5.1 Specifying a Default Yield Stress

A default yield stress may be specified in columns 14-19 on the JCNOPT line. This value overrides any values in the SACS model.

#### 3.5.2 Changing a Global Yield Stress

Any yield stress specified in the SACS model can be changed for the punching analysis with the UMOD input line. For example, for high strength steel, the design joint strength can be changed to 2/3 of the tensile strength on the UMOD input line. In the following, 50ksi is changed to 46.67ksi for the purposes of punching check.

1	2	3	4	5	6	7	8
123456789012345678901234567890123456789012345678901234567890							
JCNOPT	API	EN		T			
				AID	0.125		
UMOD	50.0	46.67					

*Note: Enter 'T' in column 34 on the JCNOPT line if all yield stress overrides are to be applied only to the chord for the purposes of strength check.*

#### 3.5.3 Changing Member Group Yield Stress

The yield for an entire member group can be modified for the purpose of checking joint capacity, by using the GMOD input line. Overrides specified on the GMOD input line take precedence over those specified on the UMOD input line.

The following changes the yield stress for groups 'TTT' and 'SSS' to 50.0 for punching analysis purposes.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
JCNOPT	API	EN		T	AID	0.125	
GMOD	50.0	TTT	SSS				

### 3.5.4 Changing Joint Yield Stress

The yield stress for specific joints can be modified by using the JMOD input line. Overrides specified on the JMOD input line take precedence over all other yield stress overrides.

## 3.6 BRACE CHORD OVERRIDES

The BRCOVER line can be used to override the effective chord length, chord can thickness and the chord tubular thickness. These overrides only affect the thickened can reduction factor as outlined in Errata and Supplement 2 API RP2A WSD 21<sup>st</sup> Edition. This line should be entered after the LCSEL line in the joint can input file.

### 3.7 ADDITIONAL JOINT CAN INPUT LINES

## JOINT CAN OPTION LINE (PART 1 OF 2)

COLUMNS	COMMENTARY
GENERAL	THIS LINE IS USED TO SPECIFY THE TYPE OF ANALYSIS, CODE CHECK, AND REDESIGN PARAMETERS TO BE USED.
( 8-11)	ENTER THE DESIRED CODE. OPTIONS ARE: 'API ' - WSD API-RP2A 21 <sup>ST</sup> EDITION CODE. 'AP21' - WSD API-RP2A 21 <sup>ST</sup> EDITION SUPPLEMENT 2. 'AP91' - WSD API-RP2A 19 <sup>TH</sup> EDITION CODE. 'AP84' - WSD API-RP2A 15 <sup>TH</sup> EDITION CODE. 'AP83' - WSD API-RP2A 13 <sup>TH</sup> EDITION SUPPLEMENT CODE. 'AP80' - WSD API-RP2A 13 <sup>TH</sup> EDITION CODE. 'LRFD' - LRFD API 1 <sup>ST</sup> EDITION CODE. 'LG ' - LINEAR GLOBAL ANALYSIS-API 21 <sup>ST</sup> EDITION SECT. 17. 'NS ' - NORSOK STANDARD 2004 N-004. 'DOC ' - DANISH 1984 EDITION CODE. 'FTG ' - API-RP2A 21 <sup>ST</sup> EDITION FATIGUE ANALYSIS. 'EQ21' - ULTIMATE EARTHQUAKE ANALYSIS - API-RP2A 21 <sup>ST</sup> ED. SUPPLEMENT 2. 'EQK ' - ULTIMATE EARTHQUAKE ANALYSIS - API-RP2A 19 <sup>TH</sup> ED. 'EQLR' - ULTIMATE EARTHQUAKE ANALYSIS - API-RP2A LRFD. 'SUS ' - SIMPLIFIED ULTIMATE STRENGTH ANALYSIS. 'MSL ' - MSL ULTIMATE STRENGTH. 'CAN ' - CANADIAN CODE. 'LLEW' - LOW LEVEL EARTHQUAKE USING API WS DESIGN. 'LLEL' - LOW LEVEL EARTHQUAKE USING API LRF DESIGN.
(12-13)	ENTER 'EN' FOR ENGLISH UNITS, 'MN' FOR METRIC WITH KILONEWTON FORCE UNITS, OR 'ME' FOR METRIC WITH KILOGRAM FORCE.
(14-19)	ENTER THE JOINT CAN YIELD STRESS IF DIFFERENT FROM THAT SPECIFIED ON THE SACS IV 'GRUP' LINES.

COLUMNS	COMMENTARY
(20-31)	ENTER THE MINIMUM AND MAXIMUM GAP ALLOWED FOR PUNCHING SHEAR ANALYSIS OF 'K' JOINTS (A NEGATIVE GAP INDICATES AN OVERLAP).
( 32 )	ENTER 'B' FOR 'BRACE-ON-BRACE' PUNCHING SHEAR ANALYSIS. THIS OPTION IS REQUIRED IF OVERLAPPED BRACE CHECK IS DESIRED.
( 33 )	USE EFFECTIVE THICKNESS FOR GROUT. ENTER '1' FOR EFF THICK BASED ON THE COMPOSITE SECTION MOMENT OF INERTIA, '2' IF BASED ON MOMENT OF INERTIAS OF THE TWO WALLS OR '3' IF BASED ON THE SRSS OF THE TWO THICKNESSES.
( 34 )	ENTER 'T' IF SY OVERRIDES ARE TWO-THIRDS THE TENSILE STRENGTH AND ALL OVERRIDES ARE APPLIED TO CHORD YIELD STRENGTH ONLY.
( 35 )	ENTER 'M' FOR MEAN STRENGTH FACTOR QU. DEFAULT IS 'C' FOR CHARACTERISTIC QU FACTOR.
( 36 )	ENTER 'U' FOR QU ULTIMATE TENSION VALUES.
( 37 )	ENTER 'R' FOR QF REASSESSMENT VALUES.
( 38 )	ENTER 'A' IF REDESIGN IS TO INCREASE THICKNESSES ONLY. ENTER 'N' IF NO REDESIGN IS TO BE PERFORMED.
(39-40)	ENTER THE DIMENSION TO BE VARIED DURING REDESIGN: 'OD' - THE OUTER DIAMETER IS VARIED. 'ID' - THE INNER DIAMETER IS VARIED. THIS IS THE DEFAULT. 'TC' - THE DIAMETER IS VARIED DURING REDESIGN.

\*NOTE: ALL DEFAULT LENGTH VALUES ARE IN ENGLISH UNITS (INCHES)  
THESE VALUES WILL BE CONVERTED TO CMS WHEN USING METRIC UNITS

LINE LABEL	JOINT CHECK OPTION	UNITS	ANALYSIS PARAMETERS						MSL OPTIONS			REDESIGN PARAMETERS				OUTPUT OPTIONS						EFF THICK LIMIT
			YIELD STRESS	MIN GAP	MAX GAP	BRACE ON BRACE OPTION	EFF. THICK OPTION	SY OVERRIDE OPTION	C OR M	ULT. OPTION	RE-ASSESS OPTION	REDESIGN OPTION	CHORD OPTIONS ID OD TC	THICK INCREM	DIAM INCREM	JOINT CAN	UNITY CHECK LEVEL	STRENGTH ANALYSIS	LOAD PATH REPORT	SCF REPORT	CHORD LOAD TRANS	
JCNOPT																						
1 - 6	8 - 11	12-13	14 - 19	20-25	26-31	32	33	34	35	36	37	38	39 - 40	41-45	46-50	56-57	58-61	62 - 63	64 - 65	66-67	68-69	76-79
DEFAULT*				-100	1000				'C'				'ID'	0.125	0.5	'FL'						1.75
ENGLISH			KSI	IN	IN									IN	IN							
METRIC (KN)			KN/SQ.CM	CM	CM									CM	CM							
METRIC (KG)			KG/SQ.CM	CM	CM									CM	CM							

## JOINT CAN OPTION LINE (PART 2 of 2)

COLUMNS	COMMENTARY
GENERAL	THIS LINE IS USED TO SPECIFY THE TYPE OF ANALYSIS, CODE CHECK, AND REDESIGN PARAMETERS TO BE USED.
(41-45)	ENTER THICKNESS INCREMENT FOR JOINT CAN REDESIGN ANALYSIS. REDESIGN WILL CONTINUE UNTIL THE UNITY CHECK IS LESS THAN 1.0 OR UNTIL THE THICKNESS EQUALS THE RADIUS.
(46-50)	ENTER THE DIAMETER INCREMENT IF THE 'TC' CHORD REDESIGN OPTION IS SELECTED FOR THIS ANALYSIS.
( 51 )	FOR API, ENTER 'N' IF THE ALLOWABLE PUNCHING STRESS IS NOT LIMITED TO THE ALLOWABLE CHORD SHEAR STRESS. FOR NORSOK, ENTER 'L' TO USE MODELED LENGTH FOR THE MODIFICATION FACTOR FOR ALLOWABLES (EQ 6.56).
(54-55)	ENTER ONE OF THE FOLLOWING FOR RESULTS REPORTED IN UC ORDER. 'FL' - ALL JOINTS FOR ALL LOAD CASES ARE PRINTED. 'UC' - JOINTS WITH UC GREATER THAN THE UC LIMIT ENTERED. 'MX' - THE MAXIMUM UC LOAD CASE ONLY FOR EACH JOINT.

COLUMNS	COMMENTARY
(56-57)	ENTER THE OUTPUT DESIRED. REPORT OPTIONS ARE: 'FL' - ALL JOINTS FOR ALL LOAD CASES ARE PRINTED. 'UC' - JOINTS WITH UC GREATER THAN THE UC LIMIT ENTERED. 'MX' - THE MAXIMUM UC LOAD CASE ONLY FOR EACH JOINT. 'RD' - 'FL' PLUS REDESIGN ITERATIONS.
(58-61)	IF A VALUE IS ENTERED HERE AND 'UC' IS ENTERED IN COLUMNS 56-57 THE OUTPUT REPORT WILL INCLUDE ONLY THOSE MEMBERS WITH UN-REDESIGNED UNITY CHECKS GREATER THAN THIS VALUE.
(62-63)	ENTER 'PT' IF THE STRENGTH ANALYSIS REPORT IS TO BE PRINTED.
(64-65)	ENTER 'PT' IF THE LOAD PATH REPORT IS TO BE PRINTED.
(66-67)	ENTER 'PT' IF THE SCF REPORT IS TO BE PRINTED.
(68-69)	ENTER 'PT' TO CHECK FOR LOAD TRANSFER ACROSS CHORD.
(76-79)	ENTER THE EFFECTIVE THICKNESS LIMIT FACTOR FOR THE CHORD FROM GROUTED PILE EFFECTS. THIS FACTOR IS USED TO INCREASE THE WALL THICKNESS OF THE LARGER (OUTSIDE) TUBE.

\*NOTE: ALL DEFAULT LENGTH VALUES ARE IN ENGLISH UNITS (INCHES)  
THESE VALUES WILL BE CONVERTED TO CMS WHEN USING METRIC UNITS

LINE LABEL	JOINT CHECK OPTION	UNITS	ANALYSIS PARAMETERS						MSL OPTIONS			REDESIGN PARAMETERS				ALLOW LIMIT	OUTPUT OPTIONS							EFF THICK LIMIT
			YIELD STRESS	MIN GAP	MAX GAP	BRACE ON BRACE OPTION	EFF. THICK OPTION	SY OVERRIDE OPTION	C OR M	ULT OPT	RE-ASSESS OPTION	REDESIGN OPTION	CHORD OPTION	THICK INCREM	DIAM INCREM		UC ORDER	JOINT CAN	UNITY CHECK LEVEL	STRENGTH ANALYSIS	LOAD PATH	SCF	CHORD LOAD TRANS	
JCNOPT																								
1-6	8 - 11	12-13	14-19	20-25	26-31	32	33	34	35	36	37	38	39-40	41 - 45	46 - 50	51	54-55	56-57	58-61	62 - 63	64-65	66-67	68-69	76-79
DEFAULT*				-100	+1000				C				'ID'	0.125	0.5			'FL'						1.75
ENGLISH			KSI	IN	IN									IN	IN									
METRICKN			KNCM2	CM	CM									CM	CM									
METRICKG			KGCM2	CM	CM									CM	CM									

## BRACE/CHORD OVERRIDE

COLUMNS	COMMENTARY
GENERAL	THIS LINE IS USED TO OVERRIDE THE JOINT CHORD EFFECTIVE LENGTH, CHORD MEMBER THICKNESS, AND THE CAN THICKNESS USED IN THE API 21 <sup>ST</sup> SUPPLEMENT 2 CODE CHECK.
( 8-11)	ENTER THE JOINT ID OF THE BRACE MEMBER CONNECTING TO THE COMMON JOINT.
(13-16)	ENTER THE OTHER END OF THE BRACE MEMBER.
(18-24)	ENTER THE EFFECTIVE CHORD LENGTH (Lc) AS DEFINED IN FIGURE 4.3-2 OF API 21 <sup>ST</sup> SUPPLEMENT 2. LEAVE BLANK IF THE PROGRAM IS TO CALCULATE THE LENGTH.
(25-31)	ENTER THE CHORD MEMBER THICKNESS (Tn IN EQ. 4.3-4). LEAVE BLANK IF THE THICKNESS DEFINED ON THE SECT OR GRUP LINES IS TO BE USED.
(32-38)	ENTER THE JOINT CAN THICKNESS (Tc IN EQ. 4.3-4). LEAVE BLANK IF THE THICKNESS DEFINED ON THE SECT OR GRUP LINES IS TO BE USED.

LINE LABEL	COMMON JOINT	CONNECTING JOINT	EFFECTIVE CHORD LENGTH	CHORD MEMBER THICKNESS	CHORD CAN THICKNESS	LEAVE THIS FIELD BLANK
<b>BRCOVR</b>						
1 - 6	8 - 11	13 - 16	18 ----- 24	25 ----- 31	32 ----- 38	39 ----- 80
DEFAULT						
ENGLISH			FT	IN	IN	
METRIC			M	CM	CM	

SECTION 4

***SACS IV***





## 4.0 SACS IV

The SACS IV model file is the standard input for all types of analyses in the SACS System. The user need generate only one structural model that can be used in any type of analysis.

The model file can be generated by various SACS program modules. Precede, Data Generator or a text editor is used to create the analysis options, model geometry and user-defined loading. Seastate or Wave Response is used to generate environmental loading data resulting from wave, wind, current, dead weight and buoyancy. Launch, Flotation or Tow is used to generate loads induced by a jacket launch, upending sequence or transportation, respectively.

The model file is made up of the following:

1. Analysis Options
2. Post Processor Options
3. Material and Section Property Data
4. Element Data
5. Joint Data
6. Load Data

## 4.1 ANALYSIS OPTIONS

Analysis options may be specified in the model file or may be designated when creating the runfile using the Executive. Options specified in the model file are input on the OPTIONS input line as follows:

1. Units must be specified in columns 14-15
  - a. EN - English
  - b. MN- Metric with kN force
  - c. ME - Metric with kg force
2. Create Super Element (column 10)
3. Import Super Element (column 9)
4. Consider/Ignore member releases (columns 21-22)
5. Include/Exclude shear effects (columns 23-24)
6. Include P- $\delta$  effects in the analysis (columns 17-18)

The following sample input designates English units, a standard analysis (columns 19-20 blank) and include shear effects:

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
OPTIONS	EN	SD					

Two analysis techniques for plate elements are supported, DKT (Discrete Kirchhoff theory) and traditional plate beam-strip theory. By default, traditional plate beam-strip theory is used. Enter 'DK' in columns 36-37 to use the DKT method.

*Note: For some structures, axial force has a significant effect on the lateral stiffness of the elements. The  $P-\delta$  option gives a first order approximation of these effects. Using the  $P-\delta$  option requires specifying  $P-\delta$  load cases (i.e. the load cases used to determine the axial force in the member) using the LCSEL line with the 'PD' option.*

Two analysis techniques for solid elements are supported, traditional constant strain 3 degree-of-freedom solids and isoparametric 6 degree-of-freedom solids. By default, constant strain 3 DOF solids are used. Enter '6' in column 71 to use the isoparametric 6 DOF solids.

Solid joint ordering has two options as well. By default, solids' joints are ordered such that flat planes in solid elements become solid faces. A more robust ordering scheme which allows solid face warpage may be specified with an 'R' in column 72.

## 4.2 POST PROCESSOR OPTIONS

Post processor options may be specified in the SACS model file but are not required. The post processor options specified are used as defaults by the Post and Postvue programs and may be modified in the Post input file.

*Note: A Post input file is not necessary if the post processing options specified in the model file are to be used.*

The following is a brief discussion of the post processing options that may be specified in the model file. The Post User's Manual addresses these features in detail.

### 4.2.1 Member Check Code

The code that member stresses are to be checked with respect to is specified on the OPTIONS line in columns 25-26.

### 4.2.2 Member Check Locations

The locations at which to check non-segmented and segmented members are specified on the OPTIONS line in columns 29-30 and 31-32 respectively.

For non-segmented members, the number of equal length pieces the member is to be divided into should be stipulated. For segmented members, specify the number of pieces each segment of the member is to be divided into. In either case, the member is checked at the beginning and end of each piece.

### 4.2.3 Output Reports

The desired output reports are designated on the OPTIONS input line. For member reports, when 'PT' is entered in the appropriate columns, all members are reported unless 'SK' appears on the individual MEMBER line. When 'SE' is specified for a member detail report, only members with 'RP' on the MEMBER line are reported.

#### 4.2.4 Redesign Parameters

If automatic redesign is desired, the parameters are designated on the 'REDESIGN' input lines.

#### 4.2.5 Hydrostatic Collapse Parameters

Hydrostatic collapse parameters are specified on the HYDRO input line. Full hydrostatic check including actual member stresses due to axial forces, bending and hoop stress can be performed by the Post program.

#### 4.2.6 Grouping Elements by Unity Check Ratio

Elements with unity check ratios that fall within a defined range can be printed together as a report group. Up to three ranges may be defined using the 'UCPART' input line. For example, all elements with unity check ratio greater than 1.00 can be reported in the first report, elements with unity check ratio between 0.8 and 1.0 in the second and elements with unity check ratio between 0.5 and 0.8 in the third report.

#### 4.2.7 Allowable Stress/Material Factor

For API/AISC working stress analysis, the calculated allowable stresses for a load case (or load combination) can be modified by specifying the load case name and the appropriate allowable stress factor on the 'AMOD' line.

For NPD analysis, the material factor used for all load cases is specified using the 'AMOD' line. Only one material factor may be specified and it must be designated for the first load case in the model, although it will be used for all load cases.

For Danish code analysis, the factors  $\gamma_m$  and  $\gamma_E$  selected on the 'GRUP' line can be changed for all members by using the 'AMOD' line. Only one factor may be specified and it must be designated for the first load case in the model, and it will be used for all load cases. This is useful for blast analysis.

#### 4.2.8 Resistance Factors

The resistance factors indicated by API are used by default when selecting LRFD codes.

The user can specify that resistance factors indicated for AISC or API seismic codes are to be used by entering 'C' or 'S' in column 40 on the OPTIONS line.

For example, the following line specifies that resistance factors indicated by AISC are to be used.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
OPTIONS	EN	SDLR	2 2	A			

The user can modify the resistance factors to be used for LFRD analyses using the RFLFRD line. The resistance factors for yield, axial compression, axial tension, bending, shear and hoop capacities for tubular and non-tubular members can be entered. For example, the following line specifies that 1.0 is to be used for axial compression and tension for both tubular and non-tubular members.

	1	2	3	4	5	6	7	8
	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890
	OPTIONS	EN	SDLR	2 2	A			
	RFLRFD	1.0	1.0		1.0	1.0		

#### 4.2.9 Euro Code Check Options

The CODE line can be used to modify the default Euro code check option, shear area option, Gamma M0 value and the Gamma M1 value

The SPAN input line can be used to identify analytical beam elements that make up physical members for serviceability and code check requirements by entering the joints in order of occurrence in the span. Any number of members can be included in a continuous line. Cantilever members can also be analyzed but must be specified by entering 'C' in column 14 of the SPAN line. Moment discontinuities and moment member end releases are allowed along the continuous member, however, force end releases are not allowed.

*Note: The beam element local x axes of all elements defined in the SPAN line are required to be acting in the same direction.*

The SACS system allows the use of beam, plate, shell and/or solid elements in the model.

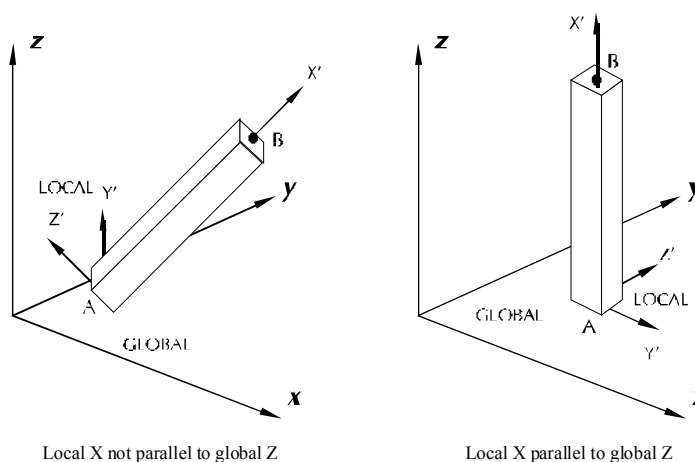
Beam elements are specified on MEMBER lines following the MEMBER header input line. Beam elements are named by the joints to which they are connected. In addition to the connecting joints, the property group label along with some optional property data are specified on the MEMBER line. Member properties specified, such as flood condition, K-factors, average joint thickness and density override data specified on the GRUP line. The following defines member 101- 201 and assigns it to property group GL2.

Note: When an average joint thickness is entered, the member length used for Euler buckling and hydrodynamic load generation is shorted by the average joint thickness. Any existing loads are not affected nor modified when an average joint thickness is specified.

Each member has an associated local coordinate system which loads and stresses may be defined with respect to. The default member local coordinate system is defined as:

The member local X-axis is defined along the member neutral axis from the first connecting joint specified toward the second connecting joint.

For members that are not vertical, i.e. local X-axis is not parallel to global Z, the local Z-axis is defined as perpendicular to local X axis, lying in the plane formed by the global Z and local X axes and having a positive projection along the global Z axis. The right-hand rule is used to determine the local Y-axis. The local Z-axis for vertical members, i.e. members whose local X-axis is parallel to global Z, is parallel to the global Y axis and in the positive Y direction. The local Y-axis is determined by using the right-hand rule. See figure below.



The default orientation of the member local coordinate system can be overridden by specifying a chord (beta) angle and/or a local Z-axis reference joint on the 'MEMBER' line. When a chord angle is input, the default local coordinate system is rotated about the local X-axis by the angle specified following the right-hand rule. The Z-axis reference joint is used with the local X-axis to define the local XZ plane. The local Z-axis is defined such that it is perpendicular to the member and positive toward the reference joint.

#### 4.3.1.1.1 Member Internal Load and Stress Sign Convention

The sign convention used by the Post program module for reporting member internal loads and stresses is dependent on the member local coordinate system as follows:

1. Axial tension is positive at both ends of the member while compression is negative at both ends.
2. Positive bending at both ends of the member causes the center of the member to deflect downward or in the negative direction of the local coordinate system.
3. Positive shear force is in the direction of the positive local member coordinate at the beginning of the member and in the negative local member coordinate at the end of the member.
4. A positive torsion vector is outward at both ends of the member.

By default, the ends of a member are fixed to the connecting joints for all six degrees of freedom. However, any of the six degrees of freedom may be released from the connecting joint by specifying a '1' in the appropriate column on the Member Description line. Degrees of freedom are in the member local coordinate system. For instance, the start of member 101-102 is fixed for axial load and shear. The torsion, moment Y and moment Z degrees of freedom are therefore released by specifying '000111' in columns 23-28. The end of the member is fixed for all degrees of freedom.

	1	2	3	4	5	6	7	8
	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890	1234567890
	MEMBER0	101	201	GL2	000	111		

Member offsets are used to shorten or lengthen the member or to move the member when the neutral axis is not located on the line between its connecting joints. When offsets are specified, the program creates a rigid link between the neutral axis of the member end and the connecting joint.

The offsets describe the length of the rigid link and may be described in local or global rectangular coordinates. The coordinate system used is specified in column 7 on the MEMBER line. Enter '1' for global coordinate system or '2' for local coordinate system. The offsets are defined on the MEMBER OFFSETS line immediately following. The following defines offsets in the global coordinate system for member 203-301.



1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER1 203 301 T01							
MEMBER OFFSETS				-60.73	9.98	79.83	52.50 -8.64-69.01

*Note: Specified member end releases are applied to the connection between the member end and the rigid link.*

#### 4.3.1.4 K-factors/Effective Buckling Length

K-factors or effective buckling length, but not both, may be specified for buckling about the local Y and Z axes. K-factors are specified on the pertinent GRUP line in columns 52-59 but may be overridden on the MEMBER line in columns 52-59.

When K-factors are used, the effective buckling length is calculated as the K-factor multiplied by the actual member length. When effective lengths are specified on the MEMBER line, 'L' must be input in column 47. The effective buckling length is then determined using the K-factor from the GRUP line multiplied buckling length specified. The following defines members 101-201 and 201-301. The effective buckling length for member 101-201 is determined using the K-factors specified for group T01 since no K-factors are specified on the MEMBER line. The effective length for member 201-301 is determined using the buckling length on the MEMBER line and the K-factors specified for group T01.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER 101 201 T01							
MEMBER 201 301 T01				L	5.0	5.0	

#### 4.3.1.5 Unbraced Length of Compression Flange

The distance between bracing against twist or lateral displacement of the compression flange for use in calculating bending allowable stresses for non-tubular members, may be input on the GRUP or MEMBER line in columns 60-64. The default is the member length. The bottom unbraced length for wide flanges or plate girders can be assigned by inputting 'B' in column 46 and the unbraced length in columns 48-51 of the GRUP line. The following designates that the unbraced length of the compression flange for member 101-201 is 5.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER 101 201 T01				2.0	2.0	5.0	

*Note: Values specified on the MEMBER line override values specified on the GRUP line.*

#### 4.3.1.6 Shear Area Factor for Tubular Members

For tubular members, the factor with which to multiply the cross section area for purposes of shear stress calculations, may be input on the GRUP line in columns 65-69 or on the MEMBER line in columns 60-64.

The following specifies a shear area modifier of 0.5 for member 101-501.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER 101 501	T01				2.0 2.0 0.5		

#### 4.3.1.7 Skipping from Output Reports

A member may be eliminated from output reports by inputting 'SK' on the MEMBER line in columns 20-21. If 'SE' was designated as the element detail report option, enter 'RP' to have the stress and unity check results reported for the particular member. All members of a group may be skipped from output reports by specifying '9' in column 47 of the GRUP line.

#### 4.3.1.8 Multiple Members Between Two Joints

A maximum of two members, spanning in opposite direction, are allowed between the same two joints. For example, two members may be modeled between joints 101 and 102, member 101-102 and member 102-101. However, all loading applied to the members will be applied to the first member specified. In general, modeling two members between the same joints is applicable when the second member is a dummy member used only to simulate additional stiffness.

#### 4.3.1.9 Defining Special Element Types

##### 4.3.1.9.1 Cable Element

Cable elements are defined using standard beam elements except that additional member data is specified on the MEMB2 line. The tension used to determine the cable stiffness is input in columns 8-14 on the MEMB2 line.

The following specifies a tension force of 10.0 for cable member 101-501.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER	101	501	AT01		2.0	2.0	0.5
MEMB2	10.0						

*Note: Enter 'A' in column 16 on the MEMBER line if additional member data is specified on the MEMB2 line.*

#### 4.3.1.9.2 Gap Element

Elements can be designated as tension-only, compression-only, no-load or friction elements for Gap analyses. The gap element type may be designated on the member group line in column 30 or on the MEMBER line in column 22 using 'T', 'C', 'N' or 'F', respectively.

*Note: The gap element type is only applicable when running a gap element analysis and is ignored for all other analysis types.*

#### 4.3.1.9.3 X-Brace or K-Brace

By default, the buckling length and K-factors specified on the GRUP and MEMBER lines in the model are used for unity check calculations for each load case.

Members making up an X-brace or chord members of a K-brace not braced out of plane may be designated as such using the MEMB2 line. The MEMB2 line allows designation of the K-factor and/or buckling length to be used for load cases where the member is part of an X-brace or the chord of a K-brace.

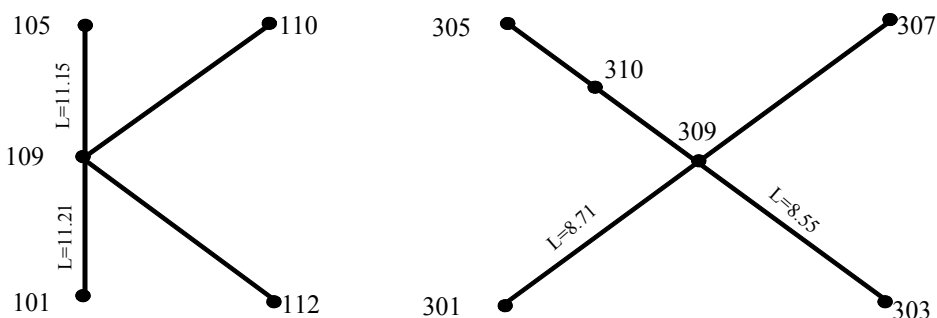
*Note: The X-brace or K-brace parameters are only applied to the axis in the plane of the connection for load cases where the member is in compression and the reference member(s) are in tension.*

The brace type 'X' or 'K' is designated in column 15. The member local axis, 'Y' or 'Z', that lies in the plane of the X-brace or K-brace is entered in column 16. Enter the reference member(s) that will be checked for tension in columns 17-32. The K-factor and/or buckling length to be used for load cases where the member is part of an X-brace or the chord of a K-brace is designated in columns 33-38 and 39-45, respectively.

*Note: K-braces require two reference members while the second reference member is optional for X-braces.*

The following example defines parameters for members 101-109 and 105-109 which are chord members of a K-brace whose local Y-axes lie in the brace plane. The diagonal or K-brace members are 109-110 and 109-112. For load cases where chord members 101-109 and 105-109 are in compression and members 109-110 and 109-112 are in tension, a K-factor of 0.8 and a buckling length of 11.15 is to be used. For other load cases, the K-factor and buckling length specified in the model file are to be used.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER 101 109 A							
MEMB2	KY	109	112	109	110	0.8	11.21
MEMBER 105 109 A							
MEMB2	KY	109	112	109	110	0.8	11.15



This example defines parameters for members 301-309 and 307-309 which are chord members of an X-brace and members 303-309, 305-310 and 310-309 which make up the two brace elements framing into the chord. The members local Y-axes lie in the plane of the brace. For members 301-309 and 307-309, a K-factor of 0.9 and a buckling length of 8.71 is to be used for load cases where the member is in compression and the other pair of members framing into the chord, 303-309 and 310-309, are in tension. For members 303-309, 305-310 and 310-309, a K-factor of 0.9 and a buckling length of 8.55 is to be used for load cases where the member is in compression and members 301-309 and 307-309 are in tension. For other load cases, the K-factor and buckling length specified in the model file are to be used.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
MEMBER 301 309 A							
MEMB2	KY	310 309 303 309	0.9	8.71			
MEMBER 307 309 A							
MEMB2	XY	310 309 303 309	0.9	8.71			
MEMBER 303 309 A							
MEMB2	XY	301 309 307 309	0.9	8.55			
MEMBER 305 310 A							
MEMB2	XY	301 309 307 309	0.9	8.55			
MEMBER 310 309 A							
MEMB2	XY	301 309 307 309	0.9	8.55			

The SACS system contains both triangular and quadrilateral orthotropic flat plate elements. The element is a true 6-degree of freedom linear strain element. The orthotropic nature of the flat plate element allows for the modeling of the following plate types: Isotropic, Membrane, Shear, Stiffened & Corrugated. The appendices contain a detailed discussion of each plate element type.

For isotropic plate elements, the plate name, connecting joints, thickness and material properties may be specified on the appropriate Plate Description line. A plate group is not required. If a plate group is specified, the material properties and thickness are obtained from the plate group unless overridden on the PLATE line.

The following defines plates AAAA and AAAB. The properties of plate AAAA are defined directly on the PLATE line while plate AAAB obtains properties from group P01.

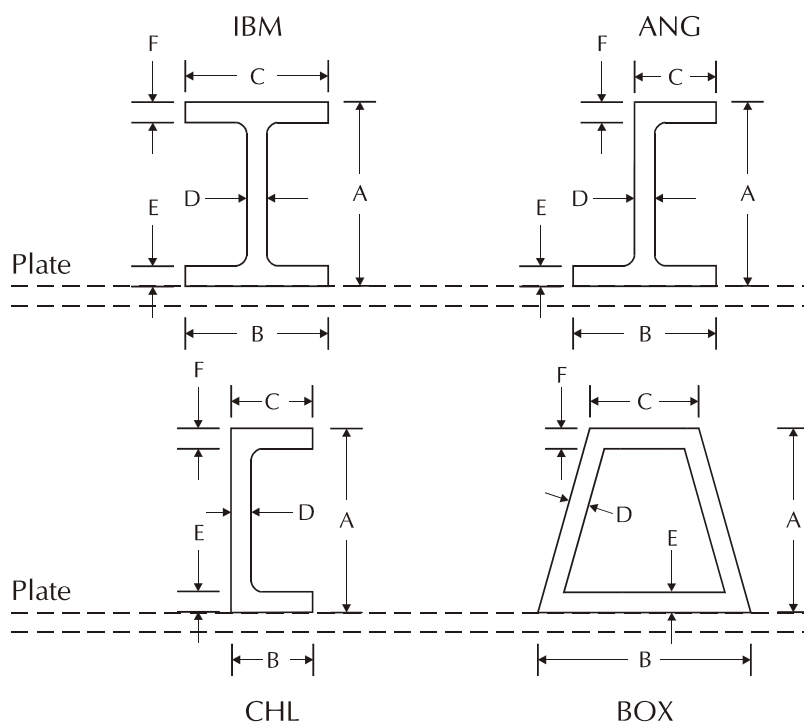
[illegible]

A PLATE line containing the plate name, connecting joints and plate property group name is used to define the plate. The plate type, thickness and material properties are stipulated on the appropriate PGRUP line. Any plate material properties input on the PLATE line override those specified for the plate group.

A PLATE line containing the plate name, connecting joints and plate property group name is used to define a stiffened plate. The plate type, material properties, stiffener section labels, stiffener direction, location (top, bottom or both) and spacing are specified on the appropriate PGRUP input line. Multiple PGRUP lines having the same group label can be used to describe plates with more than two sets of stiffeners. Plate material properties input on the PLATE line override those specified for the plate group. Plate stiffener cross sections may be any shape definable by the SECTION line. Special stiffener cross sections not available on the SECTION line may be defined using the PSTIF line. Sections not found in the section library file must be defined in the model using PSTIF lines. An outline of PSTIF geometry is shown in the diagram following.

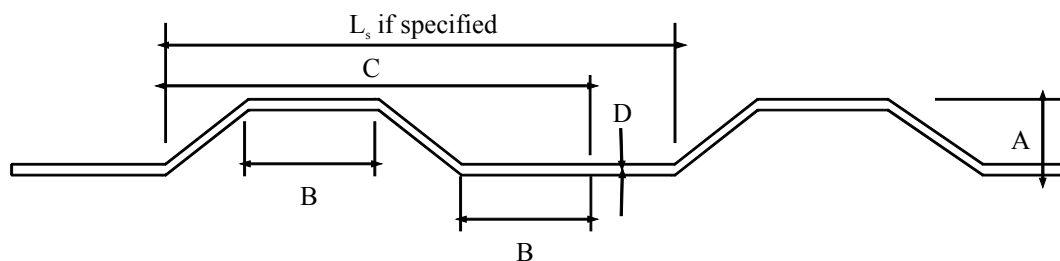
The following sample shows plate AAAA defined by group P01. Group P01 is a stiffened plate group with W12X26 running along the local X axis at 100.0 spacing. W12X26 is a section defined in the section library file.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
PGRUP	P01	1.0000	2039.0	0.2502532.0	W12X26	100.00XB	7.849
PLATE							
PLATE	AAAA	601	614	625	627P01		0



#### 4.3.2.4 Corrugated Plates

Corrugated plates are special plates with a combination of both in-plane and out-of-plane stiffness. Corrugated plates are given directly on the PSTIF line by specifying four parameters A, B, C, and D as shown in the following figure.



The following input defines a corrugated plate 'AAAB' with corrugations running in the local X direction. The thickness of the plate is 0.25 and the spacing C is 12. The A and B dimensions are 3 and 3, respectively. With the stiffener spacing unspecified on the PGRUP line, the stiffener spacing defaults to the C dimension 12. A specification of 'T' or 'B' for top or bottom stiffeners is unnecessary.

1	2	3	4	5	6	7	8
12345678901234567890123456789012345678901234567890123456789012345678901234567890							
PSTIF							
PSTIF	CRG	CORR01	3.0	3.0	12.0	0.25	
PGRUP							
PGRUP	P01	29.0	36.0	CORR01	X	490.0	
PLATE							
PLATE	AAAB	614 615 627 626	P01	0			

*Note: A von Mises check versus an allowable of  $0.6F_y$  is used to check the corrugated plate. Buckling is not included in the plate model or code check. If buckling can occur, the plate thickness may require adjustment to limit the plate capacity. The normal limitations apply such as aspect ratio and grid density as with any FE model. Since the corrugated plate has significant out-of-plane stiffness, adjacent members are assumed to share the load with the corrugated plate.*

#### 4.3.2.5 Plate Local Coordinate System

Like beam elements, each plate element has an associated local coordinate system which loads and stresses may be defined with respect to. The plate local X-axis is defined at the plate center line from the first connecting joint specified to the second connecting joint. The local XY plane is defined by the first three joints with local Y-axis perpendicular to the local X-axis toward the third joint. The right-hand rule is used to define the local Z-axis.

For example, plate 'AAAB' connected to joints 614, 615, 627 and 626 has a local X axis from joint 614 to joint 615. The local Y axis is perpendicular to the local X axis in the direction of joint 627.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
PLATE AAAB	614	615	627	626	P01		0

#### 4.3.2.6 Plate Offsets

Plate offsets may be used when the plate's center plane is not located at the plane formed by the connecting joints or when one of the edges does not correspond to a line between the joints to which it is connected. Plate offsets can also be used to generate the transition between the flat plates and beam elements. See the Commentary for a detailed discussion. When an offset is stipulated, the program creates a rigid link between the plate corner and the connecting joint. The offsets describe the length of the rigid link and may be described in local or global rectangular coordinates. The coordinate system used is specified on the PLATE line.

Local Z offsets may be specified directly on the PGRUP line in columns 36-41. For stiffened plates, the automatic offset option, which calculates the offset such that the center plane of the plate itself lies in the joint plane, may be selected by entering 'Z' in column 10. Any local Z offsets specified are added to the calculated offsets.

The following defines plate groups P01 and P02 containing a local Z offset of 10. Group P02 is a stiffened plate and also has the neutral axis offset option on so that the offset is measured from the plate center instead of the neutral axis.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
PGRUP P01	1.0000	2039.0	0.2502532.0	10.0			7.849
PGRUP P02	Z1.0000	2039.0	0.2502532.0	10.0	W12X26	100.00XB	7.849

Offsets defining the location of the plate edges are designated on the two PLATE OFFSETS lines immediately following the PLATE input line. The first offset line contains the offsets for the first two joints, and the second contains the offsets for the third and fourth (optional) joint(s). The coordinate system that the offsets are defined with respect to is designated in column 43 on the PLATE line. Enter '1' for global coordinates or '2' for local coordinates.

The following defines plate AAAB with global X offset of 10.0 specified at each joint.

1	2	3	4	5	6	7	8
1234567890123456789012345678901234567890123456789012345678901234567890							
PLATE AAAB	614	615	627	626	P01		1
PLATE OFFSETS				10.0		10.0	
PLATE OFFSETS				10.0		10.0	



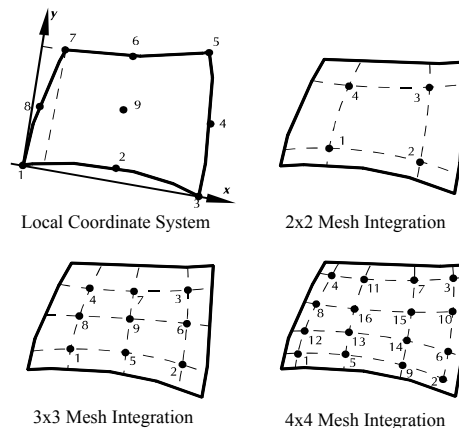
#### 4.3.2.7 Skipping from Output Reports

A plate may be eliminated from output reports by inputting 'SK' in columns 31-32 on the PLATE line. If 'SE' is designated for element detail reports on the OPTIONS line, enter 'RP' in columns 31-32 to have the stress and unity check results reported for the particular plate.

#### 4.3.2.8 Plate Modeling Considerations

Unlike beam elements, flat plate elements are not closed form solutions. Therefore, there are limitations to the geometry and mesh size that are necessary to generate accurate stresses and deflections. The following suggestions are made for the use of flat plates in the SACS system:

1. The aspect ratio (width versus height) for plate elements subjected to out-of-plane bending should be limited to 6 to 1 for three node plates and 3 to 1 for four node plates. If the primary plate load is in the plane of the plate then the aspect ratio can be increased to 10 to 1 for three node plates and 5 to 1 for four node plates.
2. Interior angles within a plate should not exceed 180 degrees.
3. Four node plates are limited to 3 degrees of out-of-plane tolerance between the four nodes such that the angle between the 'normals' to any triangular portions of the four node plate cannot exceed this value.
4. For detailed stresses, a mesh size of four nodes by four nodes will accurately represent a flat plate for both stiffness and stress calculations. A coarser mesh spacing will result in relatively accurate stiffness representation but stress calculations may not represent local stress variations within the plate.
5. Because four node plates are represented internally by 4 three node plates, a 4 node plate is inherently more accurate than a 3 node plate.
6. Plate stresses for traditional "beam-strip theory" plates are only reported at the geometric center of the plate. Plate stresses for DKT plates are reported at the corner joints and the geometric center. Plate stresses reported at the geometric center of plates are theoretically more accurate than those at corner joints.



#### 4.3.3 Shell Elements

The SACS program contains 6 node triangular, and 8 or 9 node rectangular isoparametric shell elements. Shell elements can have constant thickness or thickness may be specified at each node. Rigid link offsets can be modeled at each node to allow for connection eccentricities.

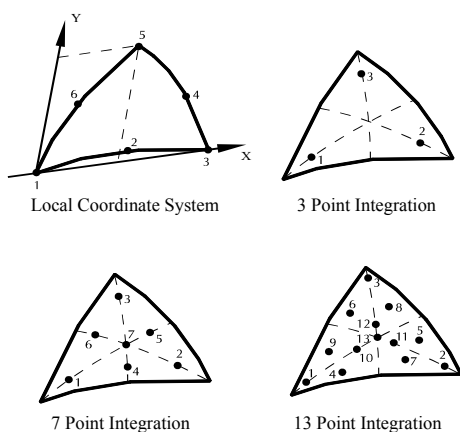
Material properties including modulus of elasticity, Poisson's ratio, yield stress, coefficient of thermal expansion and density are specified either on the SHLGRP line or on the SHELL line itself. Shell thickness, if constant, may be specified either on the SHLGRP line or on the SHELL line. For shells with varying thickness, the thickness at each node is specified on the SHELL THICK line immediately following the SHELL line defining the element.

#### 4.3.3.1 Shell Local Coordinate System

For triangular shell elements, the local X-axis is defined from node one through node three. The local Y-axis is perpendicular to the local X-axis and lies in the plane formed by nodes one, three and five. The right-hand rule is used to determine the local Z-axis. The local X-axis for a rectangular shell is defined by nodes one and three. The local Y-axis is perpendicular to the local X-axis and lies in the plane formed by nodes one, three and seven. The local Z-axis is determined by the right-hand rule. A detailed discussion on shell elements is located in the appendices.

#### 4.3.3.2 Integration Points

The number of Gaussian Integration points along the element surface is specified either on the SHLGRP line or on the SHELL line itself. The user specifies 'Fine', 'Medium' or 'Coarse' integration corresponding to 13 points, 7 points or 3 points respectively for triangular shells, or 4x4, 3x3 or 2x2 mesh respectively for rectangular shells. There are also two integration points through the element thickness for both triangular and rectangular shell elements.



#### 4.3.3.3 Shell Offsets

Shell offsets can be modeled at each node to allow for connection eccentricities. The offsets are specified on the SHELL OFFSET line in global coordinates. Two offset lines are required for 6 node elements and three are required for eight or nine node elements.

#### 4.3.3.4 Shell Element Report

If 'PT' is designated in the element detail report field on the options line, the stress details for a shell element may be skipped by inputting 'S' on the SHLGRP or SHELL line. If 'SE' or ' ' is designated in the element detail report field on the options line, all shell element details will be skipped.

#### 4.3.4 Solid Elements

The SACS program contains 4 node tetrahedron, 5 node pyramid, 6 node wedge and 8 node brick solid finite element shapes. The elements are constant strain elements and do not restrain rotation at the nodes. The solid name, connecting joints and material properties including modulus of elasticity, Poisson's ratio, yield stress, coefficient of thermal expansion and density are stated either on the SLDGRP line or on the SOLID line itself.

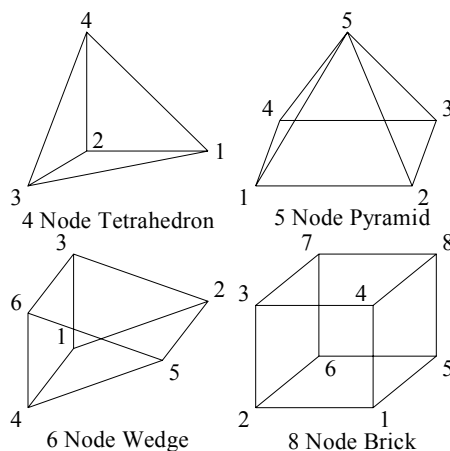
Being as these solid finite elements do not contain inherent rotational stiffness, the rotational degrees of freedom for joints contained within only solid elements will be constrained. SACS automatically generates the constraints of rotational degrees of freedom for joints which are exclusively contained in solids. With the extra constraints on solid joints, there will be extra reaction forces generated in the Post output for these constrained degrees of freedom.

Inherent rotational degrees of freedom in solid elements may be modeled by specifying '6' in column 71 of the OPTIONS line. These elements are a condensation of higher order isoparametric solid elements, with the rotational degrees of freedom being obtained from mid-side node translational degrees of freedom.

Joint ordering in solid elements is free. As such, arbitrary joint order may be input with the program determining solid faces. There are two options for joint ordering: (1) the default method which requires flat solid faces and (2) a more robust scheme allowing solid face warpage. The second scheme, which is specified with an 'R' in column 72 of the options line, has the additional feature of allowing the program to bypass joint ordering for any solid when an 'N' is specified in column 44 of the SOLID line (or column 14 of the SLDGRP line). With the default joint ordering method an 'N' specified in column 44 of the SOLID line (or column 14 of the SLDGRP line) will mean that only 8 node brick solid elements are not reordered. The default joint ordering for solids is shown in the figure.

##### 4.3.4.1 Solid Local Coordinate System

The local X-axis is defined by nodes one and two. The local XY plane is defined by nodes one, two and three. The local Y-axis is perpendicular to the local X-axis,

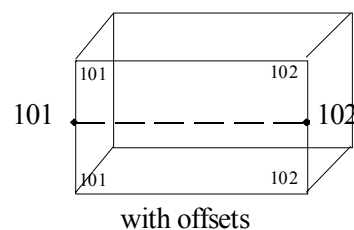
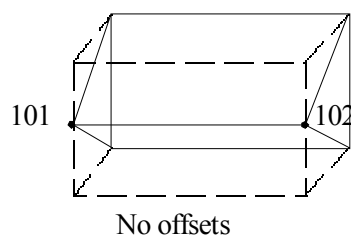


positive in the direction of node three. The right-hand rule is used to determine the local Z-axis.

#### 4.3.4.2 Solid Offsets

Solid offsets can be specified to account for eccentricities or element transitions on the SOLID OFFSET line following the SOLID line defining the element.

Normally offsets are used to locate the element relative to the connecting joints using a rigid link. Offsets can also be used to generate transitions between solid elements and isoparametric shells, flat plates, and members. For example, if a four node face of a solid element is connected to a beam or plate element, the solid face should be described using only two joints lying at the center of the face. Two joints should be specified as the four connecting joints (i.e. 101, 102, 102, 101). Offsets are then specified at each connecting joint to offset the joints to the corners of the element. The resulting offset solid element will form a full 6 degree of freedom transition connection between the elements.



#### **4.4 ADDITIONAL SACS IV INPUT LINES**

## MEMBER GROUP LINE PART 1

COLUMNS	COMMENTARY
GENERAL	THE GRUP LINES DESCRIBE GROUPS OF MEMBERS HAVING IDENTICAL STRUCTURAL, MATERIAL, AND CODE CHECK PARAMETERS. MEMBERS WHOSE CROSS SECTIONS VARY BETWEEN THE END JOINTS (SEGMENTED) CAN BE DESCRIBED WITH UP TO TWENTY (20) DIFFERENT CROSS SECTION TYPES. FOR THIS INPUT, THE MEMBER GROUP LABEL IS REPEATED FOR EACH 'GRUP' ENTRY THAT DEFINES THE CROSS SECTION AND SEGMENT LENGTH FOR EACH SEGMENT. DATA IS ENTERED FROM JOINT A TO JOINT B.
( 1- 4 )	ENTER THE LABEL 'GRUP'. A BLANK 'GRUP' HEADER LINE IS REQUIRED.
( 6- 8 )	ENTER A UNIQUE GROUP LABEL FOR THIS MEMBER GROUP. IF THIS LABEL IS REPEATED THE PROGRAM WILL ASSUME IT HAS MULTIPLE SEGMENTS.
( 9 )	FOR A TAPERED SECTION THE 'B' OR BEGIN TAPER OPTION IS USED TO DESIGNATE THAT THE SECTION SPECIFIED ON THE GROUP IS THE BEGINNING OF A TAPER AND THIS SECTION WILL BE TAPERED TO THE NEXT SECTION DEFINED ON THE PROPERTY GROUP. THE 'E' OR END TAPER OPTION DESIGNATES THAT THE SECTION SPECIFIED IS THE SECTION AT THE END OF THE MEMBER AND THE PREVIOUS SECTION DEFINED ON THE PROPERTY GROUP WILL BE TAPERED TO THIS END SECTION.
(10-16)	ENTER A SECTION LABEL FROM A 'SECT' INPUT LINE OR FROM THE SECTION LIBRARY FILE. LEAVE BLANK IF TUBULAR DATA IS ENTERED.

COLUMNS	COMMENTARY
( 17 )	THE FOLLOWING DESIGNATIONS CAN BE APPLIED TO A GROUP FOR REDESIGN: ' ' - DEFAULT TO 'REDESIGN' LINE. 'D' - DECREASE SIZE ONLY. 'E' - CONSTANT DEPTH (O.D.),DECREASE SIZE ALLOWED. 'F' - CONSTANT I.D., DECREASE SIZE ALLOWED. 'G' - MINIMUM WEIGHT, DECREASE SIZE ALLOWED. 'I' - INCREASE SIZE ONLY. 'J' - CONSTANT DEPTH (O.D.),INCREASE SIZE ONLY. 'K' - CONSTANT I.D., INCREASE SIZE ONLY. 'L' - MINIMUM WEIGHT, INCREASE SIZE ONLY. 'U' - USER-SPECIFIED DESIGN, DECREASE SIZE ALLOWED. 'X' - SKIP THIS GROUP DURING REDESIGN.
(18-23)	THE INPUT IN THIS FIELD DEPENDS ON THE SECTION TYPE AS FOLLOWS: ' ' - (BLANK) ENTER TUBULAR OUTSIDE DIAMETER. 'PLG' - ENTER PLATE GIRDER MAXIMUM DEPTH FOR REDESIGN. 'DTB' - ENTER GROUT ELASTIC MODULUS. OTHERWISE, LEAVE BLANK.
(24-29)	THE INPUT IN THIS FIELD DEPENDS ON THE SECTION TYPE AS FOLLOWS: ' ' - (BLANK) ENTER TUBULAR WALL THICKNESS. 'CON' - ENTER CONE TENSILE STRENGTH. 'PLG' - ENTER PLATE GIRDER WEB YIELD STRESS. 'DTB' - GROUT UNCONFINED CUBIC COMPRESSIVE STRENGTH. OTHERWISE, LEAVE BLANK.
(30-59)	SEE MEMBER GROUP LINE PART 2.
(60-80)	SEE MEMBER GROUP LINE PART 3.

LINE LABEL	GROUP LABEL	TAPER OPTION	SECTION LABEL	REDESIGN CODE	TUBULAR OUTSIDE DIAMETER SEE COMMENTARY	TUBULAR WALL THICKNESS SEE COMMENTARY	SEE GRUP LINE PART 2	SEE GRUP LINE PART 3
GRUP								
1 - 4	6 - 8	9	10 - 16	17	18 - 23	24 - 29	30 ----- 59	60 ----- 80
DEFAULT								
ENGLISH					IN (KSI)	IN (KSI)		
METRIC (KN)					CM (KN/SQ.CM)	CM (KN/SQ.CM)		
METRIC (KG)					CM (KG/SQ.CM)	CM (KG/SQ.CM)		

## MEMBER GROUP LINE PART 2

COLUMNS	COMMENTARY
GENERAL	THE GRUP LINES DESCRIBE GROUPS OF MEMBERS HAVING IDENTICAL STRUCTURAL, MATERIAL, AND CODE CHECK PARAMETERS. MEMBERS WHOSE CROSS SECTIONS VARY BETWEEN THE END JOINTS (SEGMENTED) CAN BE DESCRIBED WITH UP TO TWENTY (20) DIFFERENT CROSS SECTION TYPES. FOR THIS INPUT, THE MEMBER GROUP LABEL IS REPEATED FOR EACH 'GRUP' ENTRY THAT DEFINES THE CROSS SECTION AND SEGMENT LENGTH FOR EACH SEGMENT. DATA IS ENTERED FROM JOINT A TO JOINT B.
( 1- 4 )	ENTER THE LABEL 'GRUP'. A BLANK 'GRUP' HEADER LINE IS REQUIRED.
( 5-29 )	SEE MEMBER GROUP LINE PART 1.
( 30 )	ENTER GAP ELEMENT TYPE USED IN GAP ANALYSES AS 'T', 'C', 'N' OR 'F' FOR TENSION ONLY, COMPRESSION ONLY, NO-LOAD OR FRICTION.
(31-45)	ENTER MATERIAL ELASTIC PROPERTIES AS NOTED. IF LEFT BLANK, DEFAULTS OR LAST VALUES ENTERED WILL BE USED.
( 46 )	IF THE BOTTOM UNBRACED LENGTH FOR WIDE FLANGES OR PLATE GIRDERS IS INPUT IN THE AVERAGE JOINT THICKNESS FIELD, ENTER A 'B' HERE. THIS IS VALID FOR THE EUROCODE ONLY.
( 47 )	ENTER           AISC/API/NORSOK     / CANADIAN / 1984 NPD '1'   CM = 0.85 (PRIMARY)     / MOMENT   / EFF. MOMENT '2'   CM = 0.85 (SECONDARY) / CM = 1.0 / CM = 0.85 '3'   CM = 0.6 - 0.4(M1/M2)(PRI) / CM = .85 / CM = 1.0 '4'   SAME AS '3'            (SEC) / '5'   CM = 1.0 - 0.4(FA/FE)(PRI) / '6'   SAME AS '5'            (SEC) / '7'   CM = 1.0               (PRI) /

COLUMNS	COMMENTARY
	DANISH CODE ENTER '1' THROUGH '8' AS SHOWN: SAFETY MATERIAL DS       GAMMA M   E       PUNCHING SHEAR CLASS    CLASS        CODE    M    E '1' HIGH   STRICT       449   1.21 1.48 1.34 '2' NORMAL STRICT       449   1.09 1.34 1.21 '3' HIGH   NORMAL       412   1.41 1.72 '4' HIGH   STRICT       412   1.34 1.72 '5' NORMAL NORMAL       412   1.28 1.56 '6' NORMAL STRICT       412   1.21 1.56 '7' LOW    NORMAL       412   1.15 1.41 '8' LOW   STRICT       412   1.09 1.41 BRITISH CODE ENTER '1' OR '2' AS SHOWN: '1' - MT BASED ON TABLE 13, SECTION 4, BS5950. '2' - MT = 1.0 FOR MEMBERS IN THIS GRUP. EUROCODE 3 END ROTATION FACTORS (SECTION F.1.1) '1' - NO FIXITY ( K = 1.0) '2' - ONE END FIXED AND OTHER END FREE ( K = 0.7) '3' - FULLY FIXED ( K = 0.5) ENTER '8' TO INCLUDE THIS PREVIOUSLY SKIPPED GRUP IN POST. ENTER '9' TO SKIP STRESS OUTPUT FOR THIS GRUP.
(48-51)	IF MEMBER OFFSETS ARE NOT USED, ENTER AVERAGE JOINT THICKNESS. ONE HALF OF THIS VALUE IS SUBTRACTED FROM BOTH ENDS OF THE MEMBER FOR EULER BUCKLING ALLOWABLES AND HYDRODYNAMIC LOAD CALCULATIONS IN THE SEASTATE PROGRAM.
(52-59)	ENTER K-FACTORS USED FOR SLENDERNESS IN THE LOCAL Y AND Z AXES.
(60-80)	SEE MEMBER GROUP LINE PART 3.

LINE LABEL	SEE GRUP LINE PART 1	GAP ELEMENT TYPE	ELASTIC PROPERTIES			WIDE FLANGE BOTTOM UNBRACED LENGTH OPTION	MEMBER CLASS  SEE COMMENTARY	AVERAGE JOINT THICKNESS OR BOTTOM UNBRACED LENGTH	K-FACTORS		SEE GRUP LINE PART 3
			E	G	SY				KY	KZ	
				1000							
GRUP											
1 - 4	5 ----- 29	30	31 - 35	36 - 40	41 - 45	46	47	48 - 51	52-55	56-59	60 ----- 80
DEFAULT				11.6 ENGL	36.0 ENGL		'1'		1.0	1.0	
ENGLISH				KSI	KSI			FT			
METRIC(KN)				KN/SQ.CM	KN/SQ.CM			M			
METRIC(KG)				KG/SQ.CM	KG/SQ.CM			M			

### MEMBER GROUP LINE PART 3

COLUMNS	COMMENTARY
GENERAL	THE GRUP LINES DESCRIBE GROUPS OF MEMBERS HAVING IDENTICAL STRUCTURAL, MATERIAL, AND CODE CHECK PARAMETERS. MEMBERS WHOSE CROSS SECTIONS VARY BETWEEN THE END JOINTS (SEGMENTED) CAN BE DESCRIBED WITH UP TO TWENTY (20) DIFFERENT CROSS SECTION TYPES. FOR THIS INPUT, THE MEMBER GROUP LABEL IS REPEATED FOR EACH 'GRUP' ENTRY THAT DEFINES THE CROSS SECTION AND SEGMENT LENGTH FOR EACH SEGMENT. DATA IS ENTERED FROM JOINT A TO JOINT B.
( 1- 4)	ENTER THE LABEL 'GRUP'. A BLANK 'GRUP' HEADER LINE IS REQUIRED.
( 5-29)	SEE MEMBER GROUP LINE PART 1.
(30-59)	SEE MEMBER GROUP LINE PART 2.
(60-64)	THE INPUT IN THIS FIELD DEPENDS ON THE SECTION TYPE AS FOLLOWS: ' ' (BLANK) - TUBULAR RING SPACING. 'WF ', 'WFC', 'PLG' - COMPRESSION FLANGE BRACE SPACING. STIFFENED MEMBERS - UNBRACED LENGTH. OTHERWISE, LEAVE BLANK.

COLUMNS	COMMENTARY
(65-69)	THE INPUT IN THIS FIELD DEPENDS ON THE SECTION TYPE AS FOLLOWS: ' ' (BLANK) SHEAR AREA MODIFIER USED TO MODIFY TUBULAR CROSS SECTION AREA IN CALCULATION OF MEMBER SHEAR STRESS. USE 0.5 FOR PEAK STRESS (DEFAULT = 1.0). 'PLG' - PLATE GIRDER STIFFENER SPACING (FEET OR METERS). 'DTB' - GROUT DENSITY (LB/FT**3 OR TONNE/M**3). 'BOX' - (STIFFENED) LONGITUDINAL STIFFENER SPACING FOR THE VERTICAL SIDES (INCHES OR CENTIMETERS).
( 70 )	MEMBER FLOODING: ENTER 'N' FOR NON-FLOODED, 'F' FOR FLOODED.
(71-76)	MEMBER DENSITY. BOTH MEMBER FLOODING AND MEMBER DENSITY ARE ALSO USED BY 'SEASTATE' AND 'DYNPAC'.
(77-80)	ENTER THE SEGMENT LENGTH IF THE GROUP HAS MORE THAN ONE SEGMENT. TWO METHODS ARE AVAILABLE: (A) LENGTH SPECIFICATION. EACH SEGMENT LENGTH FROM 1 TO N ARE ENTERED. ANY ONE SEGMENT LENGTH CAN BE LEFT BLANK ALLOWING THE PROGRAM TO CALCULATE ITS LENGTH. (B) FRACTIONAL SPECIFICATION. EACH SEGMENT LENGTH FROM 1 TO N IS ENTERED AS A FRACTION OF THE TOTAL MEMBER LENGTH. ALL SEGMENT LENGTHS MUST BE ENTERED AND SUM TO EXACTLY 1.0.

LINE LABEL	SEE GRUP LINE PART 1	SEE GRUP LINE PART 2	TUBULAR RING SPACING	TUBULAR SHEAR AREA MODIFIER	LOAD DATA		SEGMENT LENGTH
			SEE COMMENTARY	SEE COMMENTARY	FLOODING N-BUOYANT F-FLOODED	WEIGHT DENSITY	
<b>GRUP</b>							
1 - 4	5 ----- 29	30 ----- 59	60 ----- 64	65 ----- 69	70	71 --- 76	77 - 80
DEFAULT						490.0 ENGL	
ENGLISH			FT	ABOVE		LB/CU.FT	FT
METRIC(KN)			M	ABOVE		TONNE/CU.M	M
METRIC(KG)			M	ABOVE		TONNE/CU.M	M



## EUROCODE OPTION LINE

COLUMNS	COMMENTARY
GENERAL	THIS LINE IS A USED TO MODIFY THE DEFAULT EUROCODE OPTIONS. THIS LINE WILL BE IGNORED FOR ALL OTHER CODE CHECKS. THIS LINE SHOULD FOLLOW THE 'OPTION' LINE.
( 7- 8 )	ENTER THE CODE CHECK OPTION 'EC'.
(10-11)	ENTER THE SHEAR AREA CALCULATION OPTION FROM THE FOLLOWING: ' ' - LEAVE BLANK FOR STANDARD 'ST' - USE FOR STANDARD STATIC 'E3' - TO USE SECTION 5.5.6 OF THE EUROCODE 3
(21-26)	ENTER GAMMA M0 VALUE USED FOR THE EUROCODE CHECK.
(27-32)	ENTER GAMMA M1 VALUE USED FOR THE EUROCODE CHECK.

LINE LABEL	CODE CHECK OPTION	SHEAR AREA OPTION	GAMMA M0 VALUE	GAMMA M1 VALUE	LEAVE BLANK
<b>CODE</b>					
1 - 5	7 -- 8	10 -- 11	21 --- 26	27 ---- 32	33 ----- 80
DEFAULT	EC	ST	1.1	1.1	

## POST-PROCESSING SPAN DESIGNATION

COLUMNS	COMMENTARY
GENERAL	<p>THIS LINE IS USED TO DESIGNATE THE MEMBERS CONSIDERED AS A SPAN FOR SERVICEABILITY CHECK REPORT. THIS LINE CAN BE REPEATED AS OFTEN AS NECESSARY TO SELECT AS MANY SPANS AS REQUIRED. FEATURES AND LIMITATIONS ARE:</p> <p>1) ANY NUMBER OF MEMBERS CAN BE INCLUDED IN A CONTINUOUS LINE.</p> <p>2) CANTILEVER MEMBERS CAN BE ANALYZED BUT MUST BE SPECIFIED BY THE USER.</p> <p>3) MOMENT DISCONTINUITIES ARE ALLOWED ALONG THE CONTINUOUS MEMBER.</p> <p>4) MOMENT RELEASES (SIMPLE SUPPORTS) ARE ALLOWED AT THE ENDS OF THE CONTINUOUS MEMBER BUT FORCE RELEASES ARE NOT ALLOWED.</p> <p>( 6-13) ENTER THE SPAN IDENTIFICATION. THIS IS USED ONLY FOR REPORTING PURPOSES. IF MORE THAN TWELVE JOINTS ARE TO BE USED, CONTINUE ON THE NEXT LINE WITH THE 'SPAN' IDENTIFIER LEFT BLANK.</p> <p>( 14 ) ENTER 'C' IF THIS SPAN IS CONSIDERED A CANTILEVER.</p> <p>(17-75) ENTER THE JOINTS IN ORDER OF OCCURRENCE IN THE SPAN.</p>

LINE LABEL	SPAN ID	CANTILEVER OPTION	SPAN JOINTS											
			1ST	2ND	3RD	4TH	5TH	6TH	7TH	8TH	9TH	10TH	11TH	12TH
<b>SPAN</b>														
1 - 4	6 ---- 13	14	17 - 20	22 - 25	27 - 30	32 - 35	37 - 40	42 - 45	47 - 50	52 - 55	57 - 60	62 - 65	67 - 70	72 - 75

