# FS2000

## *FS-OffFat_Spec Offshore Fatigue Analysis*

***Advanced Structural Analysis***
***for Windows***


***Email:support@aes-uk.com***

# Table of Contents

## 1.0 Introduction

OffFat-Spec is a suite of program modules that interface with FS2000 for the spectral fatigue damage analysis of dynamically sensitive framed structures under cyclic hydrodynamic wave loading.

FS-Wave is used to generate load cases corresponding to a wave being stepped through a structure. This produces element loading for each phase of the passing wave. The dynamic response of the structure to wave loading is obtained using the normal modal response method. The static response may also be used.

The response cases are post-processed by a fatigue modules that evaluates peak stress ranges at stress points within the structure as the wave passes through the structure. A number of waves (up to 100) can be processed so that a relationship between wave period and peak stress at all stress points is established for each wave direction.

For tubular joints, the geometric joint classification and associated SCFs are evaluated and used to generate the peak stress range data. This data is then used to evaluate peak stresses at all brace joints on both the chord and brace side of the joint at up to 12 circumferential points.

For non- tubular joints the peak stress data is evaluated on the nominal stress ranges at the element nodes. Individual SFCs may be defined for each node and stress point.

If finite elements i.e non beam elements are included in the model the fatigue stress ranges will be evaluated and fatigue lives evaluated accordingly.

This stress data is then combined with wave occurrence data in the form of Wave Scatter Diagrams and an appropriate SN curve to establish damage ratios based on Miners Rule.

The fatigue analysis modules are controlled by command line instructions that initiate the program modules and identify the control files to be used. These commands can become rather complex. This manual describes in detail the how the modules and their commands interact and how a batch control file should be set up. It is however, not essential to fully understand how control files are set up since a utility program is provided that will create all necessary control files and definition files to undertake a fatigue run. The use of the utility is described in Section 5.

-o-

## 2.0 Basic Approach to Fatigue Analysis

The following program modules are employed to perform the spectral fatigue analysis

- FATSPEC1
- FATIG1(FATIG1S)
- FATIG2
- FATSPEC2
- FATSPEC3
- FATIG4

The modules are activated using command line operation within a batch control file. The function and operation of each of these modules are described in detail in Section 4.

There are two basic stages in the fatigue analysis, the Analysis Phase (FATSPEC1, FATIG1, FATIG2 & FATSPEC3) and the Assessment Phase (FATSPEC3 & FATIG4). The output from the analysis phase are the Stress Point Transfer Functions relationships, this is the time consuming stage. The Assessment phases uses the Stress Point Transfer Functions, the wave spectrum, the scatter diagrams and the SN curves to evaluate the fatigue damage.

The Analysis Phase need only be carried out once on a given model configuration. The Assessment Phase is undertake for all sections of the model where different fatigue parameters are to be used eg SN curve and SCF.

The following section provides an overview of what these modules do and how they interact.

### 2.1 Dynamic Response - FATIGSPEC1

FATIGSPEC1 is a program that links FS-Wave to the Dynamic Response model to obtain a frequency domain response as a wave passes through a structure.

FS-Wave is used to evaluate the element loading as a wave is passed through the structure. Typically 12 load steps would be used to represent the wave passage. From the these loads the drag and inertia components of load are processed separately to obtain equivalent harmonic nodal loading. The amplitude and phase of these nodal harmonic loads reflect both the time lag of the wave motion and the spatial offset for each node within the structure. FATIGSPEC1 generates these harmonic nodal loads.

The dynamic response of the structure to the harmonic nodal loads is undertaken using the FS2000's Dynamic Response Module which uses the Normal Mode method to evaluate the dynamic response of linear structures. The use of the Dynamic Response Module require that a prior modal frequency analysis be undertaken to obtain the frequencies and mode shapes. This is done using the FS2000's Frequency Module.

The dynamic displacement response of the structure is evaluated using a static solver using the equivalent elastic loads (inertia and damping loads) obtained from the Dynamic Response Module. This produces result cases for each load(wave) step as the wave passes through the structure.

### 2.2 Peak Stresses - FATIG 1 & FATIG1S

From the previous result cases the peak stresses at all stress points are evaluated for each wave step. For tubular joints this process is undertaken by FATIG1. This module evaluates the SCF's and peak stresses at up to 12 locations at each brace end for both brace and chord side. For non-tubular joints this process is undertaken by FATIG1S

### 2.3 Peak Stress Ranges - FATIG 2

Once the peak stresses are evaluated at all points for all load cases it is necessary to establish for each point the maximum peak stress range as the wave passes through the structure. This is done by FATIG2.

## 2.5 Stress Point Transfer Functions - FATSPEC2

The peak stress ranges evaluated from FATIG2 are stored by direction for each wave stepped through the structure and the relationship between stress range and wave frequency i.e the Stress Transfer Function is established for every stress point in the structure. This process is undertaken by FATSPEC2.

## 2.4 Modal Static Improvement - FATSPEC2

The accuracy of a modal response solution is dependent upon the number of frequency mode shapes used. From a practical aspect only the first few modes are used as they are the most important in terms of dynamic amplification. However, when higher modes are omitted there will be a loss accuracy due to the missing mode shapes. To reduce this error a technique called 'Modal Static Improvement' is employed.

When the dynamic modal response analysis is undertaken as described above, the response at two frequencies is carried out. One at the wave frequency, and one at a near zero frequency. The latter is effectively a static solution, but limited in accuracy by the number of modes used. In addition to these two solutions a purely static solution is undertaken using the loads generated directly from FS-Wave.

The final solution is evaluated by; $S = S_{ww} - S_{wo} + S_{st}$

In cases where the dynamic response is not significant $S_{ww} = S_{wo}$ and the solution is the static solution.

In case where the dynamic response is highly dominant $S_{wo} = S_{st}$ and the solution is the dynamic solution.

This static improvement of the stress ranges $S = S_{ww} - S_{wo} + S_{st}$ is undertaken by FATSPEC2.

## 2.5 Stress Response Spectra - FATSPEC3

Offat can use one of the following wave amplitude spectrums to describe a random seastate;

- 1 ISSC Spectrum
- 2 Pierson-Moskowtz PM (ISSC)
- 3 JONSWAP

Using one the above spectra and the Hot Spot Stress Transfer Function the Stress Response Spectra can be established for each stress point.

## 2.6 Fatigue Life Evaluation - FATIG 3

From the Stress Response Spectra the significant stress range can be evaluated from the spectral moments ($S_{sig} = 4\sqrt{m0}$).

Using an upper bound value of $1.8S_{sig}$ and lower value of zero, 40 stress blocks are formed.

Using a Rayleigh probability distribution the number of stress cycles for each block can be established. The total number of stress cycles is calculated by dividing the average value for Tz by the storm durations from the scatter diagram.

For each stress block the allowable number of cycles can be obtained from the S-N curve and hence the damage ratio.

## 2.7 Total Accumulative Damage Ratio - FATIG 4

At all stress points the damage ratio is required to be summed for the different wave directions considered.

This process is accomplished by FATIG 4, which also produces the final output data.

-o-

# 3.0 Creating Models in FS2000

## 3.1 Units

OffFat reads the inputs and result files of models created and analysed by FS2000. To ensure units compatibility it is essential that the model be created in fundamental**S.I Units** or **US-Units**.

The stress and stress range data obtained from FATIG1 and FATIG2 will be in N/mm2 or Lbs/in2 depending upon the unit system employed.

**S-N Curves**

The S-N curves MUST be defined in the S.I. system (Stress in MPa). If US-Units models are used the S-N curves will be converted by the program. This enables the FS2000 supplied S-N curve data (ISO curves) to be used without the requirement to convert.

**SI System**

| | |
|---|---|
| Wave Heights and Water Depths | m |
| Material Thickness | m |
| Joint Gap | m |

**US-Units**

| | |
|---|---|
| Wave Heights and Water Depths | ft |
| Material Thickness | ins |
| Joint Gap | ins |

-o-

## 3.2 Joint Type Recognition/Classification

All joints between tubular member are examined to identify chord and brace members. The member with the largest diameter is taken to be the chord member. All other members are regarded as braces.

It is not necessary for the chord to be continuous across the joint for the member to be identified as a chord. However, the chord continuation member will be recognised as such providing the member alignment is within about 15. The chord continuation member need not be the same diameter as the chord.

In joints in which chord and brace members are the same diameter it may be necessary to fractionally increase a member OD to ensure that the member is identified as the chord.

Individual brace joint classification is evaluated according to the loading pattern in the joint in the same plane as the respective brace or the geometric configuration of the brace. For braces which carry part of their load under different classifications their classification is based on interpolation of the respective portions of each in total.

By default all braces will be automatically classified but the user may re-classify any individual brace. See Sect 3.4 for brace classification methods.

They are geometrically classified as TY, K1, K2, KT or XT joints. If a brace has a member on the opposite side of the joint it is first classified as an X joint. If the brace has an adjacent brace on the same side it is re-classified as a K1 joint. If there are more than two braces on the same side the joint is re-classified as a KT joint, with the central brace (largest angle) being classified as a K2 joint.

Gaps for K type joints are assumed to be 50 mm. For a KT joint, the gap is based on the diameter of the K2 brace + 100 mm. If <.15 then the gap is proportion to make = .15. See Sect 3.4 & 3.5 for methods of explicit gap definition or using the program calculated gaps.

-o-

## 3.3 Stress Concentration Factors (SCF)

Program evaluated tubular joint SCF's are evaluated in accordance with the following methods. Type 4 is the default method.

- Design Recommended Combination Type 1
- UEG Recommended Parametric Equations Type 2
- HSE Recommended Type 3
- Ethymiou Parametric Equations DNV/API/EC3/ISO 1434 Type 4

Thes are described more fully in Section 4.3

The increased joint wall thickness due to cans and stubs are accounted for using the same parameter file (.ECI) as used by the FS-TubeJoint. The user may re-define the SCF if the calculated SFC is not considered appropriate for the joint. The module FATIG1 evaluates the SCF data and the peak stresses. The following section describes how these values may be re-defined by the user.

The minimum SCF for a brace joint is:

1.5 Simple Joint (1.6 for Efthymiou eqations)

2.5 Ring Stiffened Joint

SCF evaluation for overlapped joints and ring stiffened joints are the same as simple joints. If joints are defined as ring stiffened the 2.5 min SCF is applied.

Member Joints

This module evaluates nominal stresses at elements nodes (no SCFs applied) i.e member joints using geometric property stress points. SCF factors may be included at the stress points by using the user defined pipework SIF factors (see Model Definition :Element Menu:Pipework).

The nominal stresses evaluated and associated lives are based on either the direct normal stresses (axial + bending) or the Von-Mises stress - See Sect 3.7

-o-

## 3.4 User Defined SCF Data & SCFs

The FATIG1 module creates the **.EF1** file which shows how the joint was classified and the SCFs evaluated and used for the peak stress evaluation.

The file format for the EF1 file is shown on Sheet D1 Appendix E. This file is re-created for the first load case each time the module is run.

The user may re-define the SCF or certain parameters that define the SCF for any brace if it is considered that a more appropriate SCF is applicable e.g. for complex joints where parametric equations are not applicable or program interpretation is inappropriate.

A utility for the evalution of SFCs is described in Appendic D. This utility can also be used to output the parameters evaluated and used by the program.

Braces that are required to be modified are identified and defined in an **.UJClass** file. This is a user created file. The file format is given below (commas or spaces can be used a field delimiters ). The entry is repeated for each brace. All data fields must be defined. Zero values will be ignored

**JBC**, *Node, Element , Gap, Classification, Multi,* $\alpha$*Value, ACS, ACC, ABS, ABC, IPCC, IPBC, OPCS, OPBS*

*Multi* is not used by this module

**Brace Gap** Any non zero value will be used by the program

$\alpha$**Value** Default value is 35. Any non zero value will be used by the program

**Classification** It is not possible to re-classify a brace to a classification with more braces than the number of braces established by the program. The following are possible

| | | |
|---|---|---|
| KT | > | TY, XT, K1 or KT |
| KT | > | TY, XT, K1 |
| K1 | > | TY, XT, K1 |
| TY | <> | XT |

Ring stiffened joints are identified by adding an S to the classification i.e.XTS

**SCF Values** Any non zero value will be used by the program

ACs = Axial Chord Saddle

ACc = Axial Chord Crown

ABs = Axial Brace Saddle

ABc = Axial Brace Crown

IPCc = In Plane Chord Crown

IPBc = In Plane Brace Crown

OPCs = Out Plane Chord Saddle

OPBs = Out Plane Brace Saddle

-o-

## 3.5 Gap Codes & Individual Gap Definition (as FS-TubeJoint)

The .ECJ file contains gap codes. The gap codes may be used to define how brace gaps are to be defined. The following code are used

- 0 Use default values ( 50 mm) or $\xi = .15$ if $\xi < .15$
- 1 Use GAP1 evaluated by the program
- 2 Use GAP2 evaluated by the program

Gap definition may also be specified by definition in the .UJClass (Joint classification file). Definition here will override the above codes. This file is also used by the static tubular joint codechecker.

-o-

## 3.6 Calculated Joint Gaps File (.EJG) (as TJointCheck)

This file shows the brace gaps evaluated by the program. This file is not used by the program, its purpose to indicate how the program interprets the joint. This file is read by the Design Parameters - Graphics Module to show joint classification.

Overlapped joints may be identified by negative values.

The file format is;

Node Number, Element Number, Geometric Classification , GAP1, GAP2, GAP, e, e/d

Geometric Classifications are those established by the program.

GAP1 is the gap in a K brace

GAP2 is the gap, between central brace and the outer brace in a KT joint. Equal gaps at each side based on the default gap is assumed.

GAP is the gap used by the program

-o-

This evaluation of nominal stresses at element nodes i.e. non-tubular joints uses the geometric property code stress points. Two stress range options are available, the nominal direct stress or the equivalent stress (Von-Mises) as described below.

Stresses and associated fatigues in tubular joints are identified as either chord side or brace side. For non-tubular joints the chord side corresponds to the nominal direct stress and the brace side to the equivalent stress.

The nominal direct stress ranges evaluation is based on the direct normal stresses (axial stress + bending stress) at the stress point.

Local SCF factors may be included at the stress points by using defined pipework SIF factors but they are more often applied in assessments using the Selective Asssessments (see Sect 4.9). If SIFs are specified the mean of the SIOF and SIIF is applied to the axial stress. Note that SIFs are applied to both pipe and beam elements.

The Von-Mises stress option can be used to include the effects of shear and hoop. It does this by evaluating the stress range based on the following equivalent stress. The direction of the equivalent stress is based on the direction of the direct stress.

For beam elements

SEQ = ((SL**2 + 3St**2)**0.5 * SL * ABS(SL)

Where:

SL = SLP + SLA

SLA = F/A

SLB = ( (io.Mo)**2 + (ii.Mi)**2 ) / Z

St = SQRT((Mtt / 2Z)**2+ (2Fy/  A)**2 + (2Fz/  A)**2)

For pipe elements

SEQ = ((SH**2 + SL**2 - SH.SL+ 3St**2)**0.5 * SL * ABS(SL)

Where:

SL = SLA + SLB

SLA = p.Do/4t  +  4F/Pi.(Do**2  - d**2)

SLB = ( (io.Mo)**2 + (ii.Mi)**2 ) / Z

SH = p.Do / 2t

St = SQRT((Mtt / 2Z)**2+ (2Fy/  A)**2 + (2Fz/  A)**2)

-o-

## 3.8 Stresses In Finite Elements

The component stresses in finite elements are evaluated in an identical manner to that done when post-processing a standard result case in FS2000.

The stresses ranges used for fatigue life prediction are based on two types of stress:

- Principle Stress
- Von-Mises Equivalent Stress

**Principle Stress Ranges**

To evaluate the stress range at a given stress point the range the following procedure is adopted:

1. The maximum absolute principle stress for each case is identified from the 3 principle stresses.
2. The sign of this maximum principle stress is maintained.
3. The stress range is based on the largest difference between the stresses from the cases being considered.

**Von-Mises Equivalent Stress**

1. The Von-Mises stress at a stress point is evaluated. Note: A VM stress is a scalar quantity with only one value at a stress point.
2. This VM stress is signed to correspond to the sign of the maximum principle stress.
3. The stress range is based on the largest difference between the stresses from the cases being considered.

Stresses and associated fatigues in tubular joints are identified as either chord side or brace side. For finite elements joints the chord side corresponds to the **principle stress** and the brace side to the **VM equivalent stress**.

**Shell Element Stress Points**

For shell element each element node will have 3 stress points corresponding to:

Pt 1 Mid
Pt 2 Top
Pt 3 Bottom

-o-

## 4.0 Operation of the Fatigue Modules

This section of the manual describes the command line operation of the fatigue modules and the file formats. The setting up of a fatigue analysis is can be somewhat tedious and error prone therefore a utility program is provided that not only sets up the fatigue analysis but also creates the batch control files to generate the basic WaveLoader load cases.

The use of the OffFat Batch Control File Utility is described in Section 5 . If this utility is used, it not essential to initially fully understand the operation of the fatigue modules described in this section since the utility sets up the commands for their operation and for the generation of all wave load data.

The remainder of the section describes the operation of the fatigue modules

The fatigue analysis batch file comprises of two basic sections.

- Analysis Phase
- Assessment Phase

The first section, the **Analysis Phase** produces the stress range data. This where all the load generation and analysis is undertaken, this a lengthy analysis process. The fatigue commands included in this section are FATSPEC1, FATIG1, FATIG2, FATSPEC2 and FATSPEC3 and also the general FS2000 analysis modules.

The second section, is **Assessment Phase** where the fatigue assessments are undertaken. The fatigue commands included in this section are FATIG3 and FATIG4. The final output from the fatigue analysis is produced in FATIG 4 in the form of text data. On-geometry fatigue life plots can also be created. The assessment phase can be a global assessment where global all stress ranges are assessed using global parameters or a selective assessment where specific parameters are assigned. In selective assessments group attribute restricts the output and the output is assigned to its own case number.

All input and output data from the fatigue analysis modules is contained in text files. This enables the user to verify all aspects of the fatigue analysis. The various file formats are described in Appendix E.

-o-

## 4.2 The Fatigue Solution Process

FS2000's post-processor creates the standard results file that are used by the Fatigue modules. FATIG1 or FATIG1S are the fatigue modules that read the elements forces contained in the result files created by the post processor.

The emphasis in this documentation is towards offshore wave fatigue analysis where the loading is generated using FS-Wave. A simplified flow for this type of analysis is:

FS-Wave - Generate Wave Loads

FATIGSPEC1 - Convert wave loads to modal frequency loading

Solution - Model Frequency Response

Post-Process

FATIG1 or FATIG1S - Peak Stresses

FATIG2- Stress Ranges

FATSPEC2 - Stress point Stress Range Transfer Functions

Post Stress Evaluation Assessments

FATSPEC3 - Damage /Scatter Diagram/WaveSpectrum/SN Curve

FATIG4 - Damage Summation

-o-

### 4.2.1 Generating Loads Cases from FS-Wave

The wave load files from the FSWave could be generated interactively in the WaveLoader but this would not be practicable. It is necessary to use the WaveLoader in Batch operation.

Note the Batch File Creation Utility described in Section 5 creates all the controls files and batch entries to do this. The remainder of this section gives some additional background

When setting up the WaveLoader for batch operation there is requirement to set up environmental files for each wave for each direction. This requires that the WaveLoader batch control files **.BC ,** that create the incremental load cases for each stepped wave will also have to be set up for each direction.

FS-Wave requires that the Hydrodynamic Model Data **.WAV** file be linked to the **.BW** file. This means that multiple WAV files for each direction are required. In the fatigue batch file a primary WAV file is copied to the appropriate direction e.g. **.WAV** is coped to **.WAVF1**. This avoids the requirement to manually create the identical directional WAV files.

The BCWAVF file are created by the Batch File Creation Utility.

***WAVF1***

**BC*WAVF1***

***---***

***---***

***WAVF8***

**BC*WAVF8***

To reduce storage requirements on large models fatigue waves load case files can be re-created for each fatigue wave. A series of load cases representing the wave passing through the structure, say L70 to L81, for 12 steps could be created. The fatigue modules FATIG1 and FATIG2 can then be run to create the peak stress range data files for all joints i.e. ".Yw" files. Only these .Y files require to be saved for subsequent fatigue analysis.

-o-

### 4.2.2 Solution

The following shows the first section of a spectral fatigue solution batch file. Appendix A give a more detailed description.

```
REM ************************ DIRECTION    1   WAVE:  1  WAVE No: 1
MFCOPY "WAV" "WAVF1"
WAVELOAD WAVF1
---------Dynamic Response
FATSPEC1 70/81/90/D/0.05/
DYNRESP1 1/50/90/1/
FATSPEC1 70/81/110/S/0.05/
DYNRESP1 1/50/110/1/
---run W=Wn
LOADA C90/1/1/
OFRAME
POST6 91-102
FATIG1 8/91/102/4/0/
FATIG2 91/102/SW1/1/
FATIG1S 8/91/102/
FATIG2 91/102/SW2/2/
---run W=0
LOADA C110/1/1/
OFRAME
POST6 111-122
FATIG1 8/111/122/4/0/
FATIG2 111/122/S01/1/
FATIG1S 8/111/122/
FATIG2 111/122/S02/2/
---run STATIC
LOADA C70/1/1/
OFRAME
POST6 70-81
FATIG1 8/70/81/4/0/
FATIG2 70/81/ST1/1/
FATIG1S 8/70/81/
FATIG2 70/81/ST2/2/
---Stress ranges using static improvement
FATSPEC2 SW1/S01/ST1/1/1/J/
FATSPEC2 SW2/S02/ST2/1/1/M/
REM ************************ DIRECTION    1   WAVE:  2  WAVE No: 2
MFCOPY "WAV" "WAVF2"
WAVELOAD WAVF2
---------Dynamic Response
FATSPEC1 70/81/90/D/0.05/
DYNRESP1 1/50/90/1/
```

-o-

## 4.3 4.3 FATSPEC1 & DYNRESP1

**FATSPEC1**

This module scans the force files .WSP(n) created by FS-Wave with the object of obtaining the load range for each element as the wave is stepped through the structure.

The (n) numbers correspond to the same load case numbers generated by FS-Wave as the wave passes through the structure. The default (n)'s from FatSpecSET are 80 to 91 i.e. 12 wave steps.

The force variation is assumed to be sinusoidal and therefore enables equivalent nodal harmonic loads to be defined by amplitude and phase angle. Drag loading and inertia loading are considered separately when approximating the load variation to a sinusoidal variation.

The program will generate one additional load case for each wave stepped wave. This load case is a dynamic load case for FS2000's Dynamic Response Module - DYNRESP1. This load case file contains the amplitude and phase for all element loading in the form of **HFORCE** nodal load commands.

**DYNRESP1 - Dynamic Response Analysis**

The FS2000's Dynamic Response module is used to generated the the dynamic response due to the stepped wave using the dynamic load cases generated by FATSPEC1

When this load case is run, the response module it will create two dynamically equivalent load cases corresponding to the stepped wave cases generated in FS-Wave. Because the response is sinusoidal the two cases , one at t=0 and one at t=T/4 are sufficient to obtain the the peak response. If the dynamic case was 90 then cases 91 to 92 would be created.

These modules will be run twice in succession, once for the wave frequency and once for the near zero wave frequency. If the Static Improvement option is not active only the wave frequency case will be run.

An extract from a typical Dynamic Response Case created by FATSEPC1 is shown below. This is for a 15s (0.6666 Hz) period wave. This will produce

```
SCASE,DYNRESP CASE
DAMP,1, .05
FREQ, 6.666667E-02
TIMERESP, 0, 3.75,2
INFORCE,91
HFORCE,1,67664.61,89.999981993361,1
HFORCE,1,15647.75,89.999981993361,2
HFORCE,1,117198.6,89.999981993361,3
HFORCE,1,89251.91,7.37878991600387E-07,1
HFORCE,1,9983.387,7.37878991600387E-07,2
HFORCE,1,154588.9,7.37878991600387E-07,3
HFORCE,2,13550.45,78.4373272123631,1
HFORCE,2,6306.59,78.4373272123631,2
HFORCE,2,33716.67,78.4373272123631,3
.
.
.
.
```

-o-

### 4.3.1 Program Input

The inputs to this program module are:

1) **Command Line Switches**

2) .WSP Files - Wave loading file created by FS-Wave

-o-

### 4.3.2 Program Output

FATSPEC1

The output from this program module is:

1) .L"n" files - Load case files "n"

For each wave a Dynamic Response Case will be generated.

This load case is load case for use with FS-Dynamic, FS2000's modal response module.

DYNRESP1

The output from the Modal Response module will be dynamically equivelent load cases (inertia and damping loads) corresponding to the steped wave cases generated in FS-Wave.

-o-

### 4.3.3 Command Line Switches

FATSPEC1 C1/C2/C3/C4/C5/

C1 Start Load Case Number

C2 Final Load Case Number

C3 Dynamic Response Case Number ( From FatSpecSET Start Load + 20 or 30 if No of Wave Steps > 20)

C4 D or S Response frequency. D(Dynamic) is frequency at the wave frequency and S(Static) is freq >> 0 ie at almost zero

C5 Critical damping for dynamic response analysis ( Typically 0.02 to 0.05)

This command will always follows the above command.

DYNRESP1 C1/C2/C3/C4/

C1 The analysis type – always 1 i.e. always frequency response for spectral fatigue analysis

C2 Mass (Frequency Results ) Case - The mass case used to evaluate the eigen modes and frequencies

C3 Dynamic Response Case Number

C4 Engineers Units 1 or 0 1 - Activates

A typical section of a batch file using these commands is:

```
---------Dynamic Response
FATSPEC1 80/91/100/D/0.05/
DYNRESP1 1/2/100/1/
FATSPEC1 80/91/120/S/0.05/
DYNRESP1 1/2/120/1/
```

-o-

## 4.4 FATIG 1 & FATIG1S

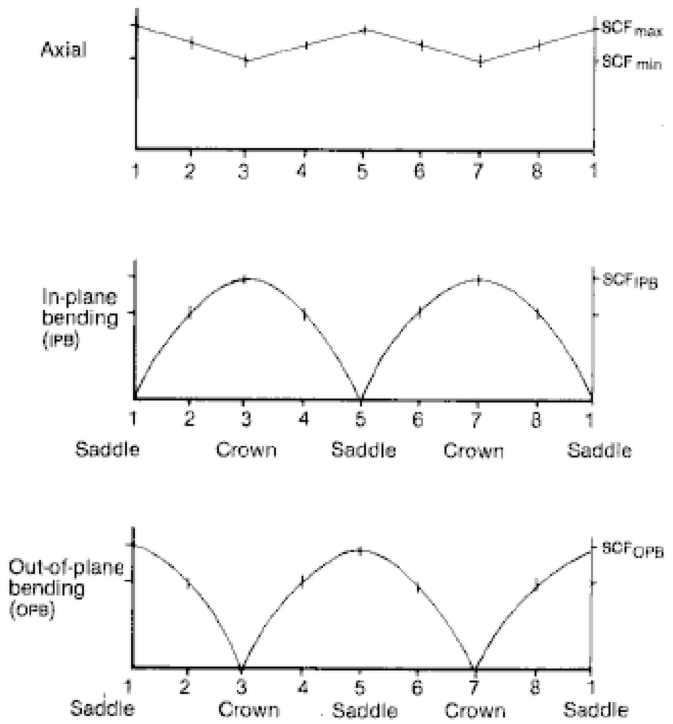
***FATIG 1***

This module evaluates the peak stresses at up to 12 locations (Stress Points) at the brace end of a tubular joint. SCF's and peak stresses at both the chord side and brace side of the weld are evaluated.

The peak stresses are obtained at each stress point using the following expression;

Stress = Axial x SCFa + Bendi x SCFi.Sin(θ) + Bendo x SCFo.Cos(θ)

SCFa is based on an interpolation of the the SCF at the saddle and crown for intermediate points between the saddel and crown.




Joints are classified by their geometric configuration.

The SCFs formulations currently available within the program are :

**1) Design Recommended Combination** **Type 1**

These are commonly used parametric equations that are considered conservative for design (Early 1990s). They are based on;

K & KT Joints (axial & in-plane) - Kuangs Equations

TY, X and K & KT (out-plane) - Wordsworth & Smedley

**2) UEG Recommended Parametric Equations** **Type 2**

**3) HSE Recommended** **Type 3**

**4) Ethymiou Parametric Equations DNV/API/EC3/ISO 1434** **Type4**

A utility for manual SCF evaluation is described in Appendix D . This shows the SCFs evaluated/used by the program. It can also be used to show other correlations and classifications using different joint parameters.

### Validity Ranges

The program does not check for the validity ranges for the various SCF correlations.

The validity ranges for the Efthymiou equations are as follows:

$$0{,}2 \leqslant \beta \leqslant 1{,}0$$
$$0{,}2 \leqslant \tau \leqslant 1{,}0$$
$$8 \leqslant \gamma \leqslant 32$$
$$4 \leqslant \alpha \leqslant 40$$
$$20^\circ \leqslant \theta \leqslant 90^\circ$$
$$\frac{-0{,}6\beta}{\sin\theta} \leqslant \zeta \leqslant 1{,}0$$

The ISO 19902 code recommends the following for cases outside these ranges.

1) evaluate SCFs using the actual values of geometric parameters;
2) evaluate SCFs using the limit values of geometric parameters;
3) use the maximum of 1) or 2) above in the fatigue analysis.

### *FATIG 1S*

This module evaluates nominal stresses at elements nodes (no SCFs applied) ie member joints using geometric property stress points. SCF factors may be included at the stress points by using the user defined pipework SIF factors (see Model Definition :Element Menu:Pipework).

The nominal stresses evaluated and associated lives are based on either the direct normal stresses (axial + bending) or the signed Von-Mises stress - See Section 3.7.

For stress evaluation in finite elements see Section 3.8

-o-

### 4.4.1 Program Input

The inputs to this program modules are:

1) **Command Line Switches**
2) .ECJ Files - Optional user defined SCF definition data (see 3.4)
3) .F"n" files - Element Force Files Created in Post-Processor
4) .ECN files - Can and Stub wall thickness file (same as static joint design )
5) .EJG -Joint Configuration File (see 3.6)

FATIG1S only requires input Item 1

-o-

### 4.4.2 Program Output

The outputs from this program module are:

1) .Z"n" files - Peak stresses at stress points for load case "n"
2) .EF1 file - Program calculated SCF file
3) .ECF - Can and Stub wall thickness file

FATIG1S only outputs Item 1 ie .Z"n" files

-o-

### 4.4.3 Command Line Switches

FATIG1 C1/C2/C3/C4/

C1 Number of Locations (stress points) at Brace End (Up to 12)

C2 First Results Case Number (Element force file)

C3 Final Results Case Number

C4 SCF equations (1 - Design Rec: 2 - UEG Recommended: 3- HSE : 4 Efthymiou )

FATIG1S C1/C2/C3/

C1 Number of Locations (stress points) at Member End

C2 First Load Case Number (Element force file)

C3 Final Load Case Number

Note the C1 in FATIG1S is only applied to tubular elements. Beam elements only have 4 points

-o-

## 4.5 FATIG 2

This module has a very simple function establishes the maximum peak stress range at all stress points from the load cases processed with FATIG1 or FATG1S.

The dynamic frequency response cases are identified using the C5 command switch. When this is active the two response cases (t=0 and t=T/4) are processed to find the peak range.

The peak stress ranges for the static cases are evaluated at each stress point as the wave is passed through the structure by scanning the results of the stepped wave. See Appendix A.

-o-

### 4.5.1 Program Input

The inputs to this program modules are:

1) **Command Line Switches**
2) .Z"n" files - Peak stresses at stress points from FATIG 1
3) .ECF - Can Data

-o-

### 4.5.2 Program Output

The outputs from this program module are:

| | | | |
|---|---|---|---|
| 1) | .Y"ID" files | - | Peak stress ranges at stress points from the range of .Z"n" files defined by C1 and C2. "ID" is the type. |
| 2) | .EC1 or .EC2 | - | Can Data C4 dictates 1 or 2 |

-o-

### 4.5.3 Command Line Switches

FATIG2 C1/C2/C3/C4/C5/

C1 First .Z file

C2 Final .Z file

C3 File ID for the .Y output file e.g. ST1 would produce .YST1

C4 1 for Tubular Joints : 2 for Member Joints

C5 S The S identifies that it is a frequency response case that uses on two response case.

A typical command lines are shown below.

The wave frequency case.

```
FATIG2 91/92/SW1/1/S/
FATIG2 91/92/SW2/2/S/
```

The static case.

```
FATIG2 70/81/ST1/1/
FATIG2 70/81/ST2/2/
```

-o-

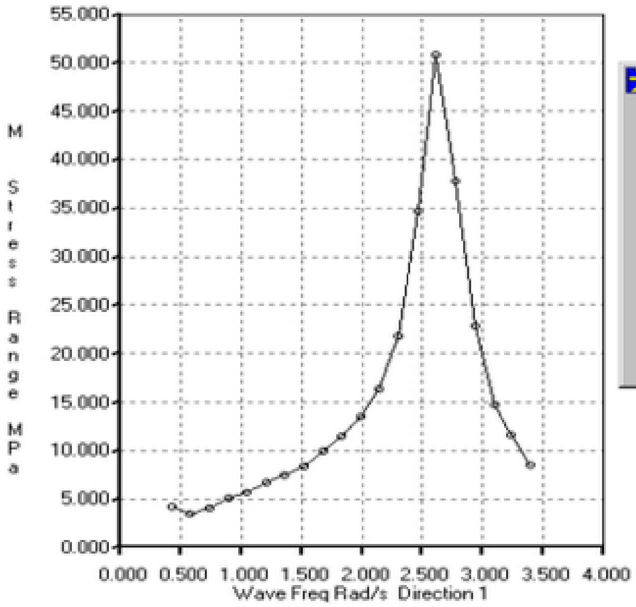
## 4.6 FATSPEC2

This module establishes the relationship between stress range and wave frequency i.e the Hot Spot Stress Transfer Function for every stress point in the structure.

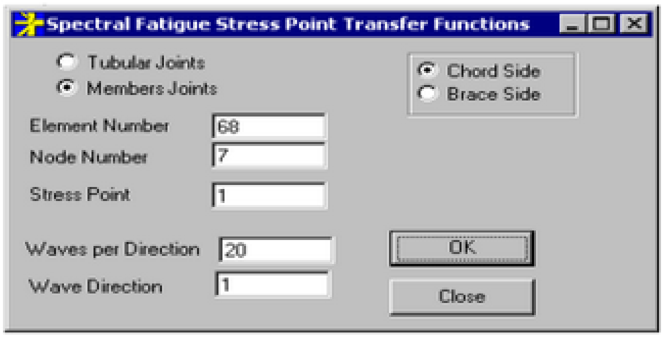
The stress range data read by this module is created by FATIG2. It also incorpates thetechnique call 'Modal Static Improvement' described in Sect 2. It does this by processing the output from FATIG2 for the wave frequenct case, the near frequenct case and the static cases.

The stress ranges are written to a random access file (by frequency).YTRF_J1(or M1) 1 is the direction number. There is a file for each direction and defines stress range as a function of wave period at each stress point.

A utility program called StessTransFunc.EXE can be used to inspect the stress transfer function for any stress point. A typical plot is shown below for a 20 waves between 1.85s 3.4 (Rad/s) period and 15s (0.42 Rad/s) period.




Stress Point Transfer Function E68 N7 StPt 1




-o-

### 4.6.1 Program Input

The inputs to this program module are:

1) **Command Line Switches**
2) .Y"SW1" Files - Stress ranges for wave frequency.
3) .Y"S01" File - Stress ranges for near zero frequency.
4) .Y"ST1" Files - Stress ranges for static case.

-o-

### 4.6.2 Program Output

The outputs from this program module are files that define the Hot Spot Stress Transfer Function every stress point in the structure for each direction.

1) .YRF_J(d) Tubular joint data

2) .YRF_M(d) Member data

(d) is the direction number.

These file are in binary format and cannot be viewed. There is a utility that enables the relationship to be plotted. This described in Sect xxx.

-o-

### 4.6.3 Command Line Switches

FATSPEC2 C1/C2/C3/C4/C5/C5/

C1 is the freq=wavefreq dynamic stress point stress range file ID.

C2 is the freq>>0 dynamic stress point stress range file ID.

C3 is the static stress point stress range file ID

C4 is the direction wave number.

C5 is the direction number.

C6 is the joint type J - Tubular joint M - Member

The following shows a typical command line entry for a tubular joints for Wave No 20 in Direction 1

```
FATSPEC2 SW1/S01/ST1/20/1/J/
```

-o-

## 4.7 FATSPEC3

This module evaluate fatigue damage on a directional basis using the wave scatter diagrams.

This module uses a Wave Surface Elevation Spectrum and the Stress Transfer Function (RAO) to establish the Stress Point Response Spectrum for each stress point.

$$S_\sigma(\omega) = |H(\omega)|^2 . S_S(\omega)$$

Where $S_\sigma(\omega)$ is the Stress Point Response Spectrum

$H(\omega)$ is the Stress Transfer Function (RAO)

$S_S(\omega)$ is the Wave Surface Elevation Spectrum

It uses one of the following Wave Amplitude Spectrums to describe a random seastate;

- 1 ISSC Spectrum
- 2 Pierson-Moskowtz PM (ISSC)
- 3 JONSWAP

The **ISSC** ( International Towing Tank Conference) spectrum used by the program is independent of wave period and is the same as the PM for the case when Tp = 10secs.

The **Pierson-Moskowitz (PM)** spectrum is given by:

$$S_{PM}(\omega) = \frac{5}{16} \cdot H_S^2 \omega_p^4 \cdot \omega^{-5} \exp\left( -\frac{5}{4} \left( \frac{\omega}{\omega_p} \right)^{-4} \right)$$

where $\omega_p = 2\pi/T_p$ is the angular spectral peak frequency.

The **JONSWAP** spectrum is given by:

$$S_J(\omega) = A_\gamma S_{PM}(\omega)\ \gamma^{\exp\left( -0.5 \left( \frac{\omega - \omega_p}{\sigma\ \omega_p} \right)^2 \right)}$$

The spectrum is formulated as a modification of the Pierson-Moskowitz spectrum for a developing sea state in a fetch limited situation ( parameter recommendations from DNV RP C205)

where
SPM($\omega$) = Pierson-Moskowitz spectrum
$\gamma$ = non-dimensional peak shape parameter
s = spectral width parameter
$\sigma$ = sa for $\omega = \omega$p
$\sigma$ = sb for $\omega > \omega$p
A$\gamma$ = 1- 0.287 ln($\gamma$) is a normalizing factor.

The following value for the peak shape parameter $\gamma$ is applied:

Average values for the JONSWAP experiment data are sa = 0.07, sb = 0.09.

$$\gamma = 5 \quad \text{for} \quad T_P / \sqrt{H_S} \le 3.6$$

$$\gamma = \exp(5.75 - 1.15 \frac{T_p}{\sqrt{H_s}}) \quad \text{for} \quad 3.6 < T_P / \sqrt{H_S} < 5$$

$$\gamma = 1 \quad \text{for} \quad 5 \le T_P / \sqrt{H_S}$$

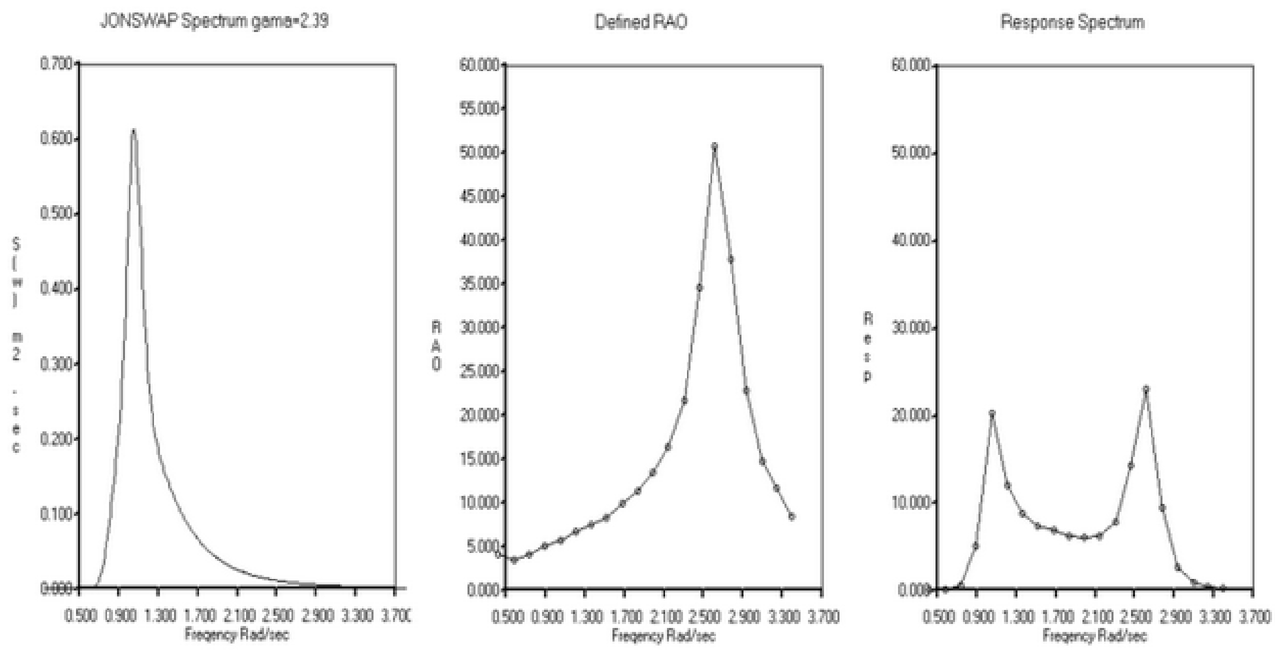
where $T_P$ is in seconds and $H_S$ is in metres.

0.15 Rad/s (41.9s) is the lower frequency cut-off and 5 Rad/s(1.26s) the upper cut-off point for the spectrum for the specific spectrum curve, the curve is divided into 500 equal divisions between the cut-off points.

FATIGUE DAMAGE

1. From the Stress Response Spectrum the significant stress range can be evaluated from the spectral moments ($S_{sig}$ = 4√m0).
2. Using an upper bound value of 1.8S $_{sig}$ and lower value of zero, 40 stress blocks are formed.
3. Using a Rayleigh probability distribution the number of stress cycles for each block are established. The total number of stress cycles is calculated by dividing the average value for Tz by the storm durations from the scatter diagram.
4. For each stress block the allowabale number of cycles can be obtained from the S-N curve and hence the damage ratio.

The FS2000 Seaway utility can be used view the respone spectrum. A typical output is shown below. The RAO plot is the Stress Point Transfer function and the Respones Spectrum is the Stress Range Spectrum. To use the Seaway utility a RAO file for the stress point must be created, this described in Section 4.5.6.




-o-

### 4.7.1 Program Input

The inputs to this program modules are:

| | | |
|---|---|---|
| 1) | Command Line Switches | |
| 2) | .YTRF_J(d) or .YTRF_M(d) | Stress point Transfer Function Files by direction(d) |
| 3) | .EC1 or .EC2 - | Connectivity and wall thickness data files |
| 3) | .DBS(d) | Scatter Diagram by direction (d) see 4.5.5 |
| 4) | SNCURVE.DAT | Must be located in FS2000 user folder |

-o-

### 4.7.2 Program Output

The outputs from this program module is:

1) .DSJ"d" files - Damage ratios at tubular joint stress points by direction

2) .DSM"d" files - Damage ratios at all member joint stress points by direction

-o-

### 4.7.3 Command Line Switches

**FATSPEC3** C1/C2/C3/C4/C5/C6/

C1 Joint type 1-Tubular joint 2-Member

C2 Number of Wave used to define the Stress point Transfer Function

C3 Number of Directions

C4 Additional global SFC

C5 S-N Curve Identification String (see 4.5.7)

C6 Wall thickness - used for non tubular joints (optional -if zero or omitted default is SN curve reference thickness)

C7 Group Set to be read (Faster operation - Use is optional)

C8 Restrict processing to elements assigned to Group C8 in Group SET C7(Faster Operation - Use is optional)

On large models especially when doing selective assessments it is highly recommended to use group restriction.

-o-

### 4.7.4 SN Data

Fatigue life damage based on the following expression

LOG(N) = LOG(K2) - mLOG(s) - mLOG(t/tref) $^{n}$

When t is less than tref, no thickness effects are applied.

The SN curve constants are read from a file named SNCURVE.DAT. This file **MUST EXIST** in the FS2000 directory. This file may be extended to include user-defined curves. Alternatively SN curves can be created in a model related file. If 'modelname'.USNC exists in the model directory it will be read in preference to the above DAT files.

There will be 4 SN curve data files in the FS2000 folder.

SN_ISO.DAT
SN_DNV.DAT
SN-EN.DAT
SNCURVE.DAT

The DNV/API and ISO curves both use a S suffix to identify a seawater curves e.g. in the ISO code F2 or F2S will be present.
The SNCURVE.DAT should be regarded as user defined file that can modified or added to be the user.

The file format for SN curves is given below. If curves do not exist in the distributed file they may be easily added.

The curve is described in two lines. The first line is the string description used for identification. The second line has 8 data fields. These are

$LOG(K2)_1$ $m_1$ StressChange $LOG(K2)_2$ $m_2$ EndLim tref n(tref corr'n exp)

Suffix 1 or 2 permit a two part SN Curve. The change point is defined by StressChange. EndLim is the fatigue endurance limit i.e. stress range below which fatigue damage does not occur.

```
B
15.0055     4     100.3172  17.00688   5     0      .022  .25
C
13.626      3.5   78.18847  16.46571   5     0     .022   .25
D
12.1817     3     53.36218  15.63617   5     0     .022   .25
E
12.0151     3     46.95698  15.3585    5     0     .022   .25
F
11.8004     3     39.82296  15.00067   5     0     .022   .25
F2
11.6342     3     35.05366  14.72367   5     0     .022   .25
G
11.3939     3     29.14965  14.32317   5     0     .022   .25
W
11.197      3     25.0611   13.995     5     0     .022   .25
T
12.1638     3     52.63401  15.60633   5     0     .032   .25
EN160
```

```
12.901        3      117    17.036        5      64      0.025  .25
EN140
12.751        3      104    16.786        5      57      0.025  .25
EN125
12.601        3      93     16.536        5      51      0.025  .25
EN112
12.451        3      83     16.286        5      45      0.025  .25
```

-o-

### 4.7.4 Scatter Diagram Wave Data

The wave occurrence data has to be presented in the form of wave scatter diagram. A scatter diagram has to be craated for eaxh wave direction. The data is contained in a **<model>DBS"n"** file , where n represent sthe wave direction number. The format of this file is presented in Appendix E.

A grid editor can be used to created this data. This is accessed from the **Edit Scatter Diagram Button** in the Batch File Creation Utility.

The entries in the scatter diagram represent the number of 3 hour storm durations in the defined return period. The total number of waves in the Hs and Tp bin in the scatter diagram is calculated by dividing the value by Tz for the storm durations ie 3 hours. For a single entry in the bin the number of waves per year (N) is calculated by the following:

N = NB x 3600 x 3 x DSF / (RT x Tz)

Where

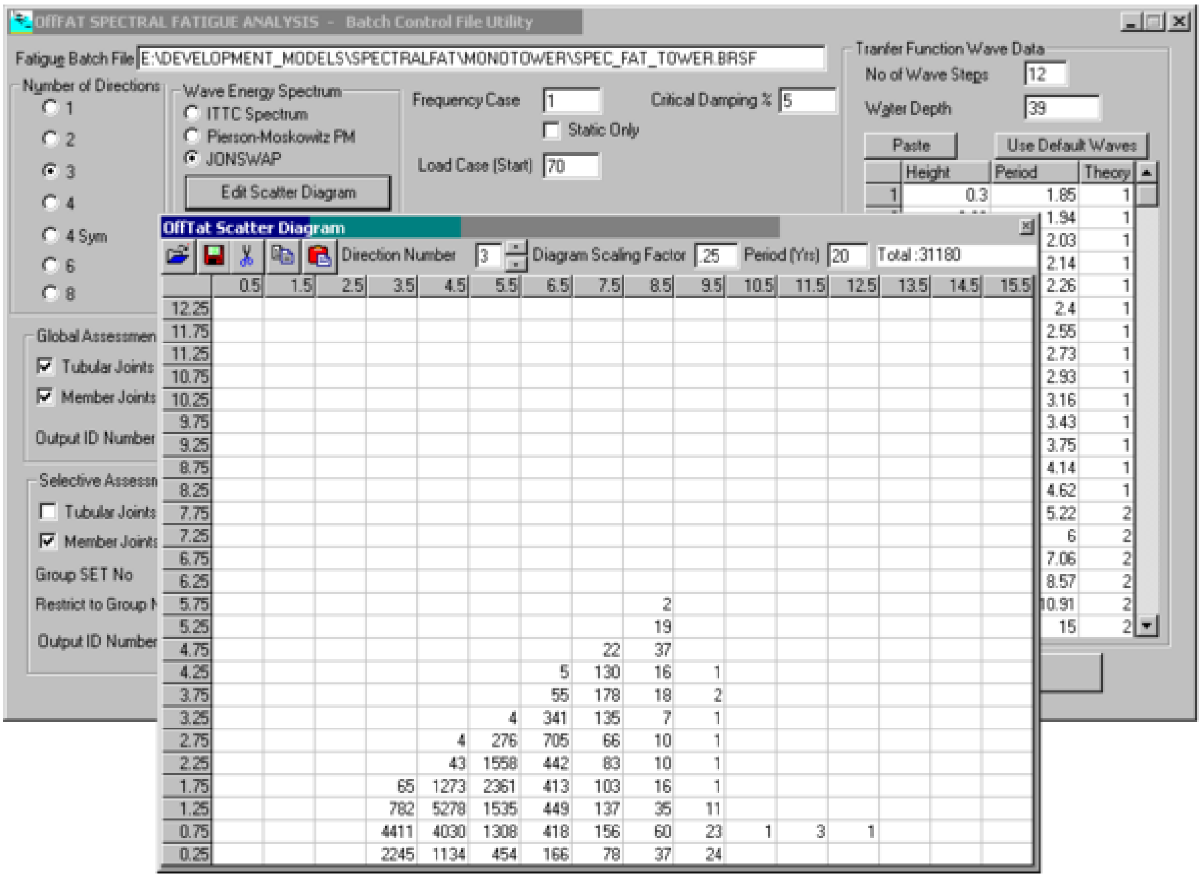
NB = Value in scatter diagram bin for a specifc Tp and Hs
RT = Return period in years
DSF = Diagram Scaling Factor
Tz = Mean zero crossing period = $2\pi$ x SRQT(m0/m2)

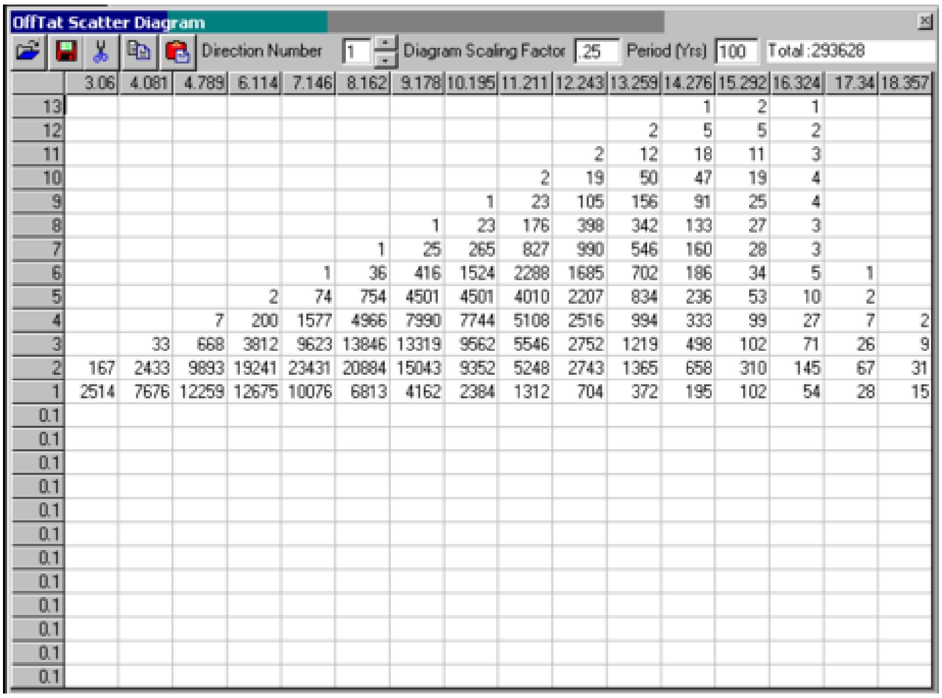
It the storm duration for the bin is for a different duration then this can be accounted for using the Diagram Scaling Factor (DSF). This factor can also be used to account for approximating directional variations when using omni-directional wave data.

***Changing Block Intervals***

The Scatter diagram grid editor fixes the block intervals to those shown directly below. The block intervals cannot be edited in the grid editor but they can be changed by editing the DBS'n' file. When the file is loaded into the the grid editior the block intervals will take the values of the edited file. This is shopwn in the lower diagram

The gid size cannot be changed. Blank entries will appear as zeros in the DBS file.


OffFAT SPECTRAL FATIGUE ANALYSIS - Batch Control File Utility
Fatigue Batch File E:\DEVELOPMENT_MODELS\SPECTRALFAT\MONOTOWER\SPEC_FAT_TOWER.BRSF
Number of Directions
Wave Energy Spectrum
ITTC Spectrum
Pierson-Moskowitz PM
JONSWAP
Edit Scatter Diagram
Frequency Case
Static Only
Load Case (Start)
Critical Damping %
Tranfer Function Wave Data
No of Wave Steps
Water Depth
Paste
Use Default Waves
Height
Period
Theory
Global Assessmen
Tubular Joints
Member Joints
Output ID Number
Selective Assessm
Group SET No
Restrict to Group N
OffTat Scatter Diagram
Direction Number
Diagram Scaling Factor
Period (Yrs)
Total :31180


OffTat Scatter Diagram
Direction Number
Diagram Scaling Factor
Period (Yrs)
Total :293628

-o-

### 4.7.6 Stress Response Spectrums

This module uses a Wave Surface Elevation Spectrum and the Stress Transfer Function (RAO) to establish the Stress Point Response Spectrum for each stress point.

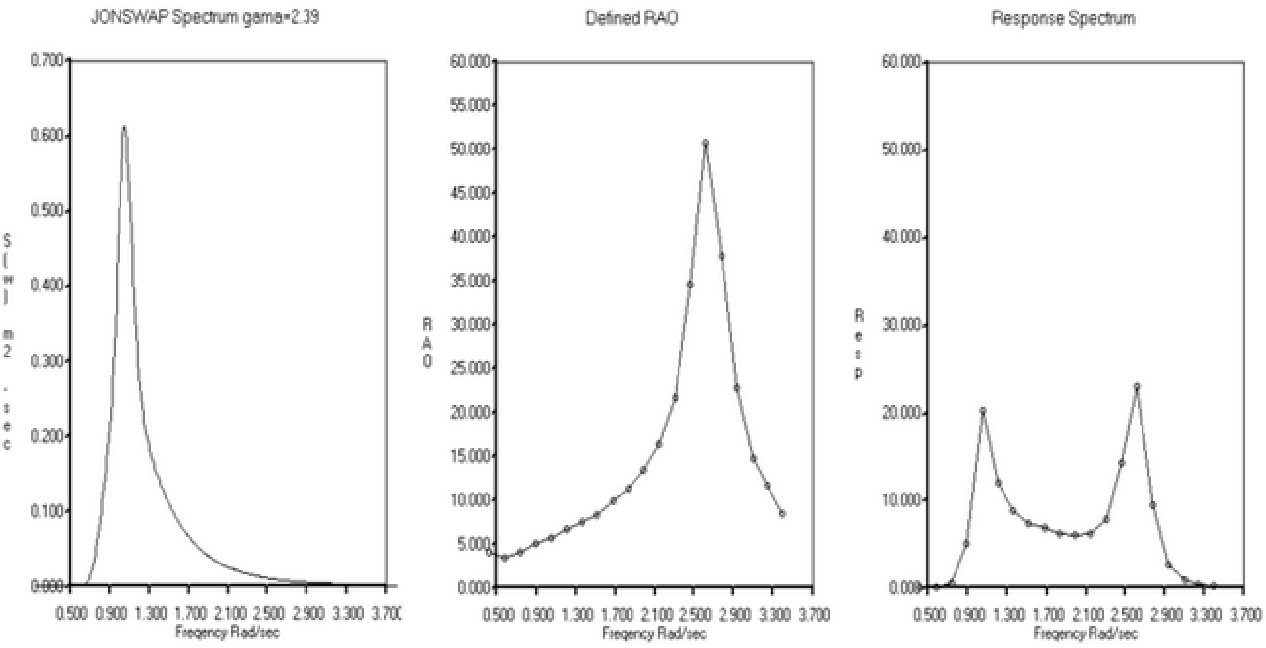
$$S_\sigma(\omega) = |H(\omega)|^2 . S_S(\omega)$$

Where $S_\sigma(\omega)$ is the Stress Point Response Spectrum

$H(\omega)$ is the Stress Transfer Function (RAO)

$S_S(\omega)$ is the Wave Surface Elevation Spectrum

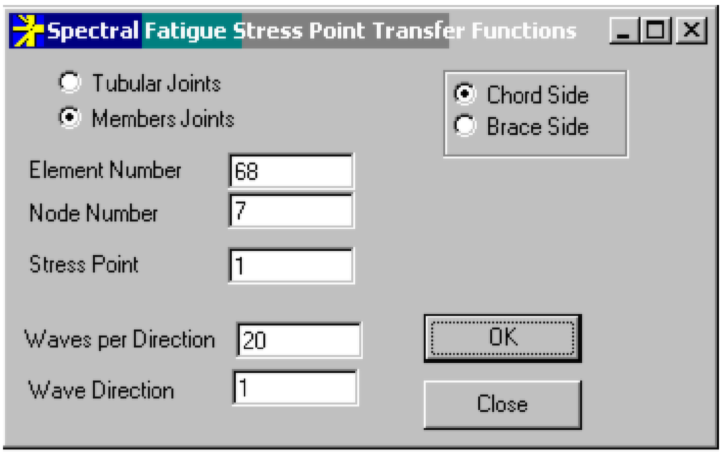
This data can be inspected and plotted for each stress point using the utilities describe below.




**Stress Transfer Function (RAO)**

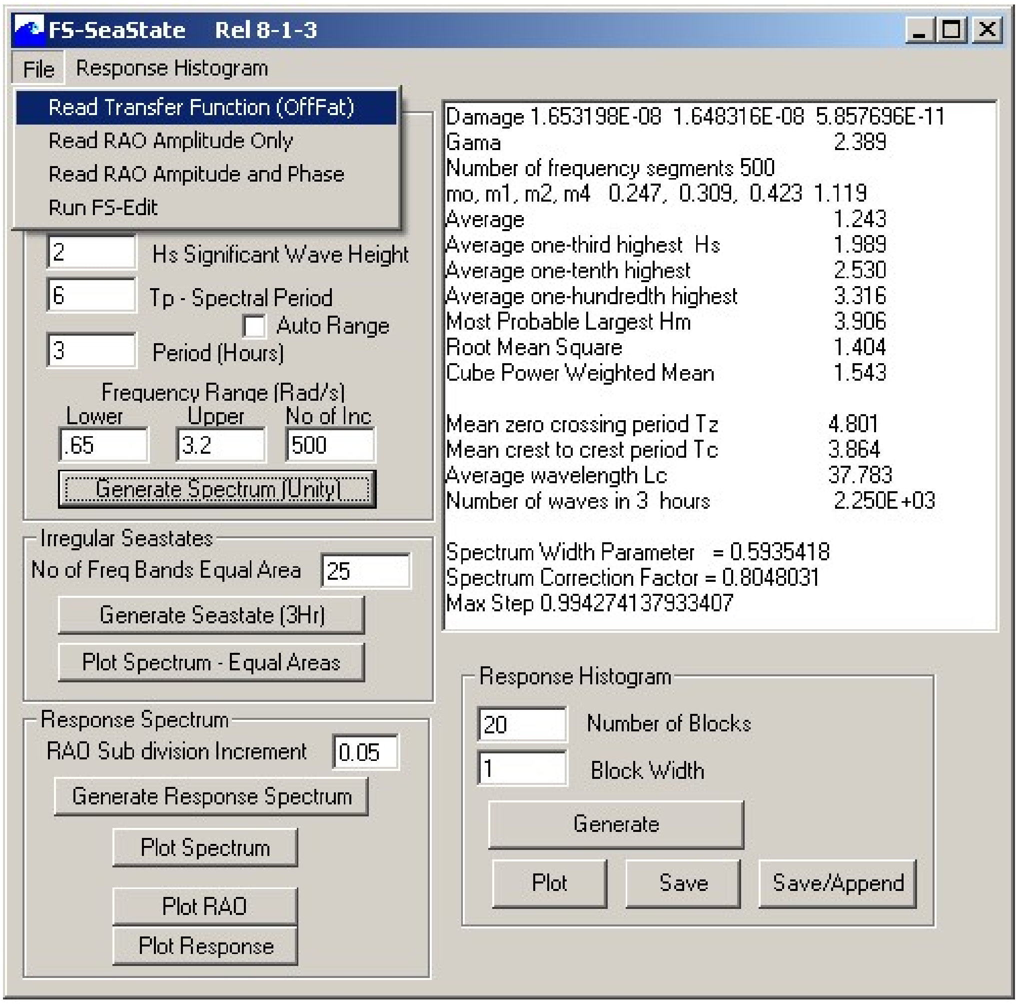
A utility (StressTranFunc.EXE) can be used to create a transfer function file <Modelname>.~TRFu. It will also plot the file.

The transfer function plot will show the suitability of the wave period selection used to create the transfer function.



### Stress Point Response Spectrum

A utility (Seaway.EXE) can be used to generate response spectrum from a transfer function file <Modelname>.~TrFu.




Use the **File:Read Transfer Function (RAO**) command to read a <Modelname>.~TrFu file.

Select the wave spectrum and enter the wave parameters (Check Auto Range).

Enter 0 in the **RAO Sub Division Inc** (can be used but OffFat only uses the RAO data points ie no interpolation).

Click the **Generate Response Button** to generate the stress response button.

-o-

## 4.8 FATIG 4

This module sums the damage ratio at all stress points for all wave directions and produces the result data in both text and graphical life plot data.

FATSPEC3 processes the fatigue data for all stress points using the current fatigue parameters. When selective assessments are undertaken it is this module that selects what is to be output.

-o-

### 4.8.1 Program Input

The inputs to this program modules are:

1) **Command Line Switches**

2) .D"d" files - Damage Ratio Files for each wave direction

-o-

### 4.8.2 Program Output

The output for the fatigue analysis will be output as Tubular Joint Design Results using the following categories.

| | |
|---|---|
| Summary Output | - Minimum fatigue life at joint, output text |
| Stress Output | - Fatigue life at all joint stress points, output text |

Section 4.8 described in more detail how the results may be used.

-o-

### 4.8.3 Command Line Switches

FATIG4 SC1/C2/C3/C4/C5/C6/C7/C8/

C1 First DSM file number ("d") ;Note the file number is preceded with the letter S

C2 Final DSM file number ("d")

C3 Fatigue life limit for .JF output text files

C4 RC - Output Results Case Number

C5 Fatigue Category

1 for Tubular Joints

2 for DS Member Joints or

3 for VM Member Joints

C6 Group SET to be read.

C7 Restrict output to Group defined in SET C6

C8 Restriction Applies to 0 - Elements 1 - Nodes

C9 Description for output

If C7 is preceded by a - the output will be sorted into groups up to the group attribute limit defined by C7

-o-

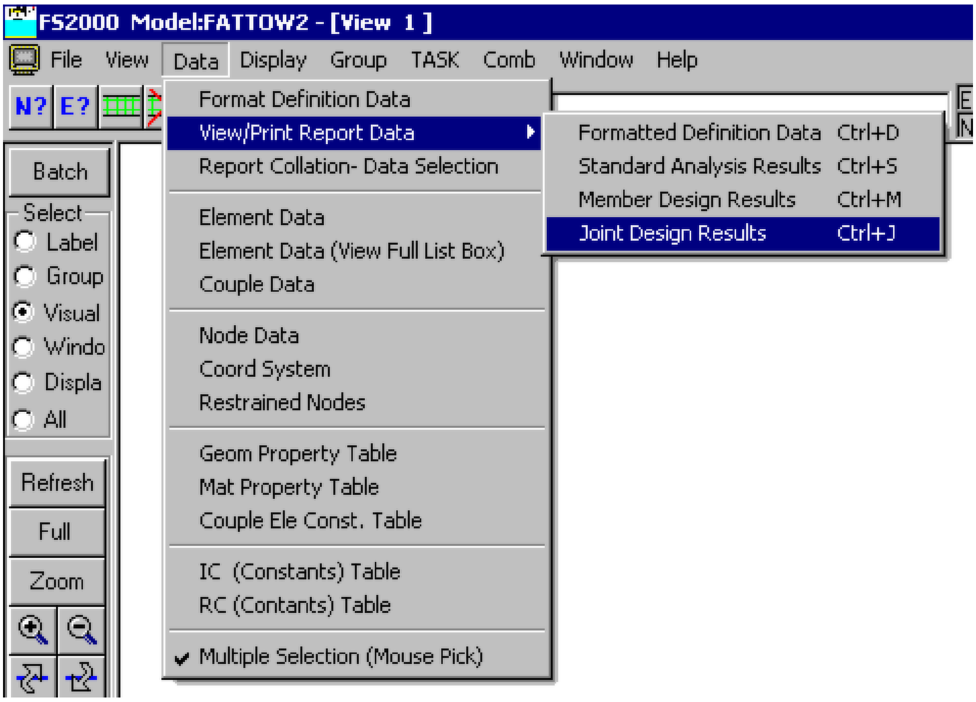
## 4.9 Output Results

The output for the fatigue analysis will be output as Tubular Joint Design Results using the following categories.

| | | |
|---|---|---|
| Summary Output | - | Minimum fatigue life at joint, output text |
| Stress Output | - | Fatigue life at all joint stress points, output text |
| Life plots | - | Damage ratio plot file |

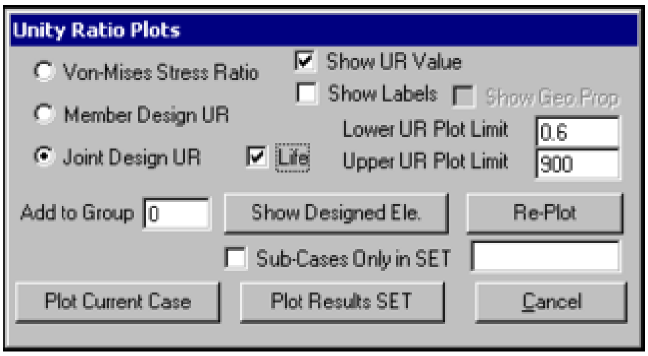
The result case number (RC) to which the output is assigned is defined by C4 in the Command Line Switches.

The FS2000 menu commands shown below can be used to view the fatigue results

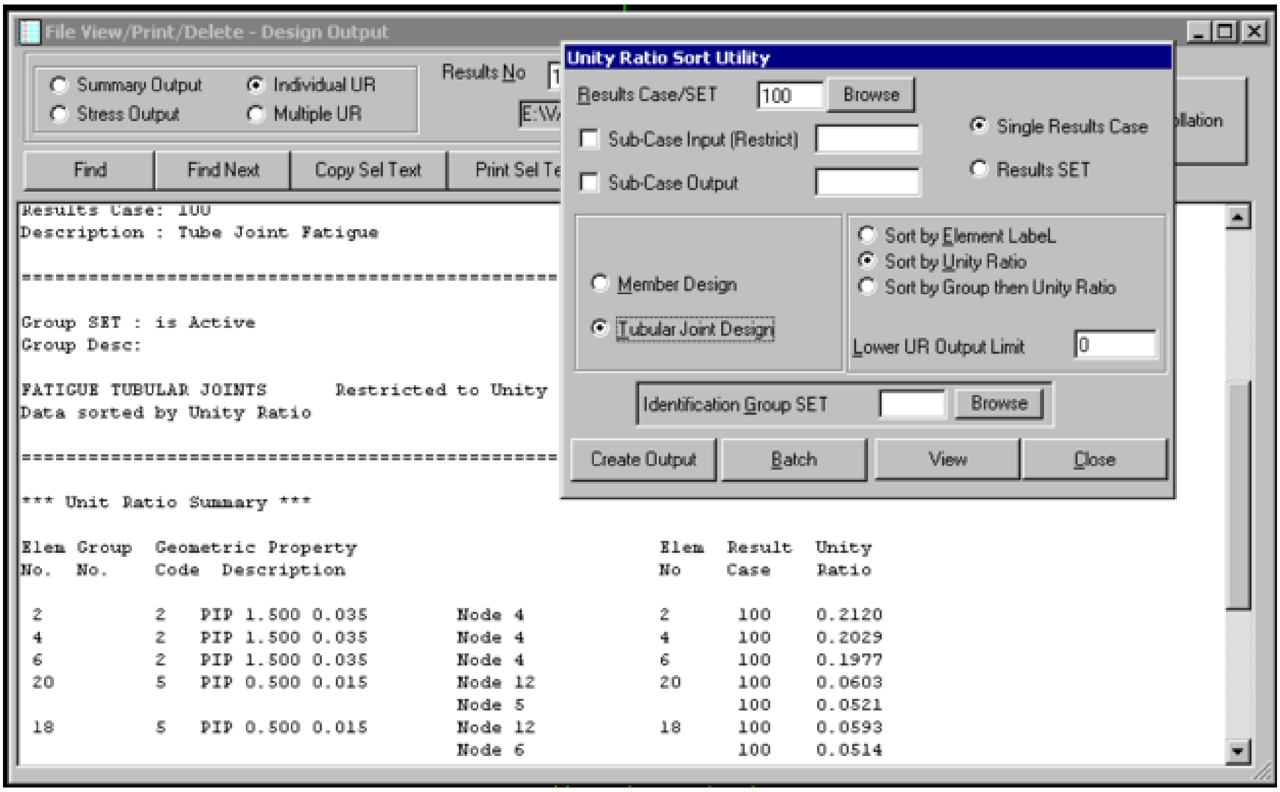



### Plotting Fatigue Lives

Life plots are obtained by using the tubular joint design UR plot facility. By default this will plot the damage ratio but if the **Life** option is active the life in years will be plotted. Note: The Lower UR plot limit is interpreted as a damage ratio, therefore lower the value if longer lives are to be plotted. Ensure that the **Life** option is active.




The tubular joint Sorted Unity Ratio (UR) can be used to sort the fatigue life output into an ascending order list based on damage ration.




-o-

## 4.10 Using Non Global Parameters - Selective Assessments

When the Batch File Creation Utility (see Section 5) is used the set up the fatigue batch files, the fatigue prediction for all stress ranges will be based on global default values. It is likely that non-default parameter will be required at different locations on the structure, this is achieved by undertaking selective assessments.

The parameters that often required to be changed are:

- Local SCF
- SN Curve
- Reference wall thickness for SN-Curve correction.

Results Case for assessment. These additional command lines are added to the bottom (after the Analysis Phase) of the fatigue batch to form an additional assessment phase. Alternatively, they could be included in a separate batch used only for selective assessments.

It is highly **recommended** that when undertaking selective assessments on large models the elements processed by FATSPEC3 be limited to only those of interest using the element group attribute (see 4.7.3) so that the solution time is minimised.

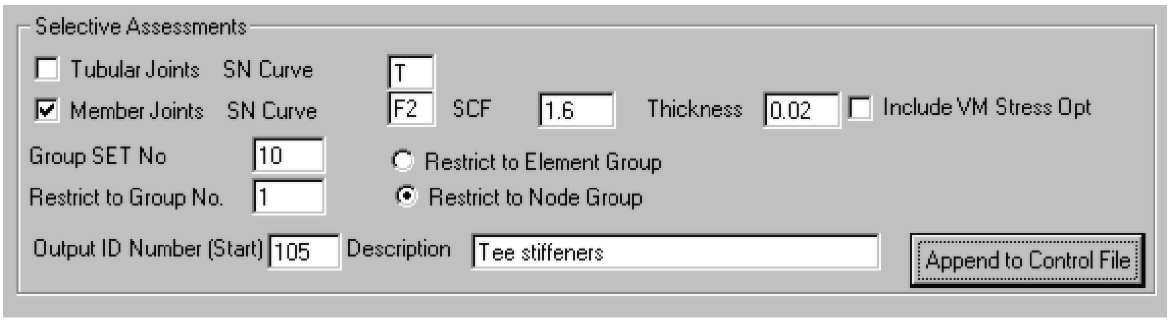
The following show an example of the command to achieve this. A review of the module command line switches will indicate the resulting output.

**Non-Tubular Joint Assessments**

```
-----------------------------------------------------
--------------------------------------------------------------------
REM  SCF=1.6: F2 Curve  t=0.02: SET 10: Group 1: Result Case 105
---------------------------------------------------------------------
FATSPEC3 2/20/0/1.6/F2/0.02/
REM ******************COMBINE ALL DIRECTIONS
FATIG4 S1/0/10000/105/2/10/1/1/Tee stiffeners/


--------------------------------------------------------------------
REM  SCF=2.8: F2 Curve  t=0.02: SET 10: Group 1: Result Case 106
--------------------------------------------------------------------
FATSPEC3 2/20/0/2.8/F2/0.02/
REM ******************COMBINE ALL DIRECTIONS
FATIG4 S1/0/10000/106/2/10/1/1/A1-Transitions/
```

Note that the Batch File Creation Utility has the capability to append selective assessments to a batch file ( see Sect 5). The settings below would would enter the first of the above batch entries.




**Tubular Joint Assessments**

The Batch File Creation Utility will always add a unity SCF to tubular joints. It this is required to be changed, this has to be done by editing the Batch file. Element Group restrictions also have to be manually added.

The example below shows a line in which a non unity SCF is applied and an element group restriction is applied.

```
----------------------------------------------------------------------------------------------------
----
REM  T Curve: SET 10: Group 1: SCF=0.5 Result Case 500
----------------------------------------------------------------------------------------------------
----
FATSPEC3 1/20/3/0.5/T/0/10/1/
REM *******************COMBINE ALL DIRECTIONS
FATIG4 S1/3/10000/500/1/10/1/1/Single Brace/
```

-o-

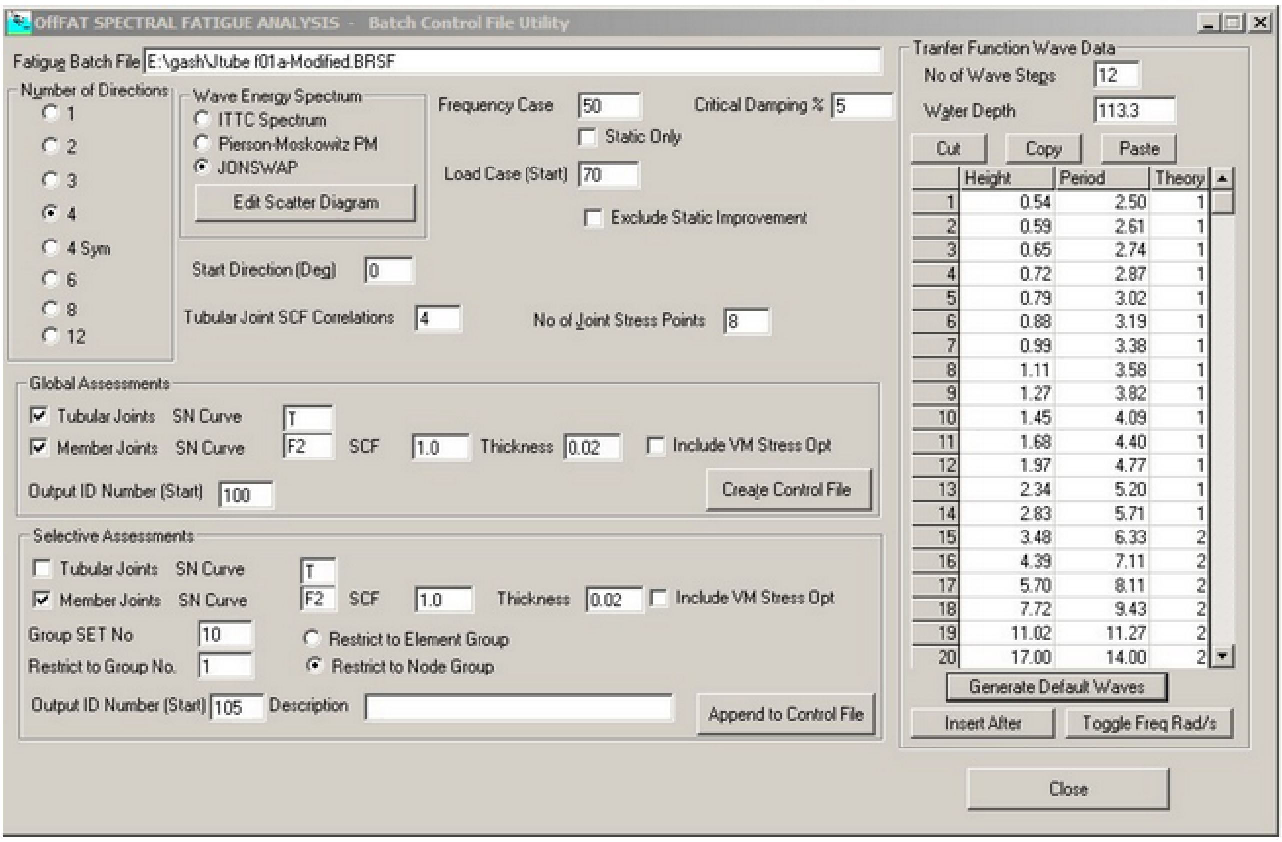
## 5.0 Batch File Creation Utility

This utility program is started from the Windows Start menu by clicking the OffFat-Spect icon in the FS2000 program menu.

This utility will generate the Batch file and associated model files necessary for a fatigue analysis. It can be used as a 'black box' for fatigue analysis, the user should have an understanding of the how the resulting Batch file undertakes the fatigue analysis.

The default-input data is as for Example 1 given in Appendix F. The input form below will appear when the program is activated is given below.

The set up data is saved to a file with the extension **.DFS.** This file will always be archived.



The following describes how the above data relates the operation of the Fatigue Modules described in the Section 4.

The **Create Control** button in the **Global Assessment** frame is used to create the batch control file. This will overwrite any other previous batch file with the same name. When the batch file is created it will also create all data to run a fatigue analysis based on the current model,

The **Append to Control File** in the **Selective Assessments** frame is used to identify sections in the models where specific SN curves, SCF and thickness are to be applied. It does this by appending additional Assessment Phase commands (FATSPEC3 & FATIG4) to the bottom of the existing command file. Node or elements groups identify these sections. See Section 4.9.

Note that **Selective** assessments can be appended at any time to the main batch file or added to new batch files (Append will also create). It is not necessary to run the whole batch file when undertaking selective assessments (FATPEC3 & FATIG4). However, it is essential that the Analysis Phase of the batch has been run so that the stress range data is available for the assessments. Also, if Tubular or Member Joints are required to be selectively assessed that they must have also been selected for the Analysis Phase of the global selective, this ensures that appropriate stress range data exists.

**Fatigue Batch File** This is the file name of the Fatigue Batch Control File. It may be convenient to create a separate batch file for selective assessments which do not contain the Analysis Phase.

**Number of Directions** This is used to define the number of directions to be considered. 4 Symm is applied at 45 deg increments and the wave action is assumed to be symmetrical in the 4 directions.. For each direction a wave scatter diagram must be defined.

**Start Direction** This defines the direction of the wave relative to the model co-ordinate system. See FS-Wave Manual.

**Start Load Case** This is sthe first load case used to evaluate This may be set to any load case number, just avoid overwriting any other cases. The number of load cases created and used is defined by the **Number of Wave Steps.** Although a larger number of load cases are used analysed the cases are repeatedly overwritten.

The **Wave Energy Spectrum** defines the type of random seastates to be used in conjunction the wave scatter diagrams. The **Edit Scatter Diagram Button** is used to access a the scatter diagram grid editor.

The **Fequency Case** references the mass case used to evaluate the free vibration modes of the structure. Note that the uses is required to set up the batch commands for the frequenct solution.

The **Critical Damping** defines the critical damping ratio for each vibration mode during the dynamic response analysis

The **Static** check box if active will evaluate the Stress Point Transfer Functions based on the the static response of the structure

The **Transfer Function Wave Data** section defined the waves to be used to evaluate the Stress Point Transfer Functions.

**Number of Wave Steps** Defines the number of wave steps (load cases) used to evaluate the loading from a given wave as it passes through the structure.

**Wave Depth** Defines the water depth for the wave evaluation.

**Number of Waves** This defines the number of waves and associated characteristics for the wave to be used for load generation. The waves defined here enable the Stress Point Transfer Functionsa to be obtained for each stress point. Wave Theory : 1 is Airy, 2 is Stokes 5th, 3 Stream Function. The **Use Default Waves** button will add 20 wave with a frequency interval of 0.25 Hz and a constant wave steepness of 1:18. ( $H = (g/2/\pi/18).T^2$)

The **Generate Default Waves** button can be used to generate a series of waves between two wave periods with a equal frequency increment.

The **Insert After** button will insert a wave below the selected wave. The wave frequency will be the mean of two existing waves (height based on a steepness of 18).

The **Toggle** Feq Rad/s will show the frequency in Rads/s.

**ASSESSMENTS**

**Tubular Joints** If checked will evaluate fatigue lives at tubular joints using the default SN curve. Joints can be referenced to specific curves.

**Member Joints** If checked will evaluate fatigue lives at using element nodes based on nominal stresses using the defaults SN curve. The **VM Stress** option is used to evaluate fatigue damage based on the Von-Mises stress at the stress point. This can be used to scan for areas where shear stresses (torsional or direct) may be significant. Normally only direct and bending stresses are considered at member joints.

**Number of Stress Points** Defines the number of stress point used in fatigue life evaluation of tubular joints. Max is 12. For non- tubular joints the 8 stress points are used for tubular elements and 4 stress points for beam elements.

SCF Correlations - Code 1 to 3 to identify SCFs for tubular joints (see 4.4)

1 Design Recommended Combination

2 UEG Recommended Parametric Equations

3 HSE Recommended Parametric Equations

4 Efthymiou (DNV/API) Parametric Equations

The **Output ID Number (Start)** define the case number to which the results will be assigned (see 4.8). Two cases will be created if both tubular and member joints are active.

**Selective Assessments** see Section 4.10

-o-

## APPENDIX A - Basic Fatigue Analysis Process

The following describes the basic analysis process with reference to the batch file commands.

1) Frequency Analysis - The first stage obtains the lower natural modes of vibration.

```
REM******************* FREQUENCY ANALYSIS
BAND 10
WAVESORT Y
LOADA 1/0/0/
DYNFRAMB
```

2) Evaluate the stress range at all stress points for each wave in each direction

```
REM ************************************************* DIRECTION    1
REM ************************************************* WAVE         1
REM ************************************************* WAVE NUMBER 1
MFCOPY "WAV" "WAVF1"
WAVELOAD WAVF1
```
Generates waves cases as the wave stepped through the structure

```
---------Dynamic Response
FATSPEC1 70/81/90/D/0.05/
```
Identifies maximum and minimum loads on elements from stepped wave

Generates equivalent nodal harmonic load case (L90) at wave frequency

```
DYNRESP1 1/1/90/1/
```
Modal response analysis to establish dynamic response at wave frequency

The response analysis produced equivalent dynamic static as the wave passes

```
FATSPEC1 70/81/110/S/0.05/
```
Identifies maximum and minimum loads on elements from stepped wave

Generates equivalent nodal harmonic load case (L110) at near zero wave frequency

```
DYNRESP1 1/1/110/1/
```
Modal response analysis to establish dynamic response at near zero frequency

The response analysis produced equivalent dynamic static as the wave passes

```
---run W=Wn
LOADA C90/1/1/
```
Runs the equivalent static dynamic case at the wave frequency

```
OFRAME
POST6 91-102
FATIG1 8/91/102/4/0/
FATIG2 91/102/SW1/1/
FATIG1S 8/91/102/
FATIG2 91/102/SW2/2/
```
Produces stress point stress ranges for the subject wave at the wave frequency

```
---run W=0
LOADA C110/1/1/
```
Runs the equivalent static dynamic case at the near zero wave frequency

```
OFRAME
POST6 111-122
FATIG1 8/111/122/4/0/
FATIG2 111/122/S01/1/
FATIG1S 8/111/122/
FATIG2 111/122/S02/2/
```
Produces stress point stress ranges for the subject wave at near zero frequency

```
---run STATIC
LOADA C70/1/1/
```
Runs the static wave loads cases

```
OFRAME
POST6 70-81
FATIG1 8/70/81/4/0/
FATIG2 70/81/ST1/1/
FATIG1S 8/70/81/
FATIG2 70/81/ST2/2/
```
Produces stress point stress ranges for a static response

```
---Stress ranges using static improvement
```

Applies the static improvement method and stores each stress point stress range

`FATSPEC2 SW1/S01/ST1/1/1/J/` Appends to the tubular joint stress range transfer function

`FATSPEC2 SW2/S02/ST2/1/1/M/` Appends to the member joint stress range transfer function

3) Damage Assessment

```
REM **********************************GLOBAL ASSESSMENTS
```

`FATSPEC3 1/20/3/1/T/` Wave Apectrum/Stress Transfer Function to establish the Stress Response Spect and Fatige Dammage

`FATSPEC3 2/20/3/1.0/F2/`

From the Stress Response Spectra the significant stress range can be evaluated from the spectral moments ($S_{sig} = 4\sqrt{m0}$).

Using an upper bound value of 1.8$S_{sig}$ and lower value of zero, 40 stress blocks are formed.

Using a Rayleigh probability distribution the number of stress cycles for each block can be established.

The total number of stress cycles is calculated by dividing the average value for Tz by the storm durations from the scatter diagram.

For each stress block the allowable number of cycles can be obtained from the S-N curve and hence the damage ratio.

```
REM *******************COMBINE ALL DIRECTIONS
```

`FATIG4 S1/3/10000/100/1/` Combines all direction an creates the out data

`FATIG4 S1/3/10000/101/2/`

At all stress points the damage ratio is required to be summed for the different wave directions considered. This process is accomplished by FATIG 4, which also produces the final output data.

-o-

# APPENDIX B - Batch Command Line Summary

### Command Line Summary

**FATSPEC1** C1/C2/C3/C4/C5/

C1 Start Load Case Number

C2 Final Load Case Number

C3 Dynamic Response Case Number ( From FatSpecSET Start Load + 20 or 30 if No of Wave Steps > 20)

C4 D or S Response frequency. S is freq at the wave freq and S is freq >> 0 ie at almost zero

C5 Critical damping for dynamic response analysis ( Typically 0.02 to 0.05)

**DYNRESP1** C1/C2/C3/C4/

C1 The fanalysis type – always 1 i.e. frequency for spectral fatigue analysis

C2 Mass (Frequency Results ) Case - The mass case used to evaluate the eigen modes and frequencies

C3 Dynamic Response Case Numbe

C4 Engineers Units 1 or 0 1 - Activates

**FATIG1** C1/C2/C3/C4/

C1 Number of Locations (stresspoints) at Brace End (12 max)

C2 First Results Case Number (Element force file)

C3 Final Results Case Number

C4 SCF equations (1 - Design Rec: 2 - UEG Recommended: 3- HSE : 4 Efthymiou

**FATIG1S** C1/C2/C3/ (Non-tubular Joints)

C1 Number of Locations on Tubes (Up to 12)

C2 First Load Case Number (Element force file)

C3 Final Load Case Number

**FATIG2** C1/C2/C3/C4/C5/

C1 First .Z file

C2 Final .Z file

C3 File ID for the .Y output file e.g. ST1 would produce .YST1

C4 1 for Tubular Joints : 2 for Member Joints

C5 S used to identify that two frequency response cases are to be processed.

**FATSPEC2** C1/C2/C3/C4/C5/C5/

C1 is the freq=wavefreq dynamic stress point stress range file ID.

C2 is the freq>>0 dynamic stress point stress range file ID.

C3 is the static stress point stress range file ID

C4 is the direction wave number.

C5 is the direction number.

C6 is the joint type J - Tubular joint M - Member

**FATSPEC3** C1/C2/C3/C4/C5/C6/

C1 Joint type 1-Tubular joint 2-Member

C2 Number of Wave used to define the Stress point Transfer Function

C3 Number of Directions

C4 Additional global SFC

C5 S-N Curve Identification String (see 4.5.7)

C6 Wall thickness - used for non tubular joints (optional -if zero or omitted default is SN curve reference thickness)

C7 Group Set to be read (Faster operation - Use is optional)

C8 Restrict processing to nodes or elements assigned to Group C8 in Group SET C7(Faster Operation - Use is optional)

**FATIG4** SC1/C2/C3/C4/C5/C6/C7/C8/C9/

C1 First DSM"d" file number . Note the the file number is preceded with the letter S

C2 Final DSM"d"file number

C3 Fatigue life limit for .JFD output text files in Years

C4 RC - Output Results Case Number

C5 Fatigue Category

1 for Tubular Joints

2 for DS Member Joints or

3 for VM Member Joints

C6 Group Set to be read

C7 Restrict output to Group C7 in Group SET C6

C8 Restriction applies to 0 - Elements or 1 - Nodes

C9 Description for RC output

-o-

## APPENDIX C - Fatigue Related Files

***Fatigue Programs***

FATSPEC1.EXE

DYNRESP1.EXE

FATIG1.EXE

FATIG1S.EXE

FATIG2.EXE

FATSPEC2.EXE

FATSPEC3.EXE

FATIG4.EXE

***Batch Utility Options File***

| | |
|---|---|
| .DFS | Data file for Batch File Creation Utiliy |

***Model Independent Data Files***

| | |
|---|---|
| SNCURVE.DAT | SN Curve Definition |

***Specific Fatigue Related Definition Files***

| | |
|---|---|
| .DBS(d) | Directional Scatter Diagram Data |
| .ECJ | Optional User Defined SCF Data |

***Raw Data Output Files***

| | |
|---|---|
| .F"n" | Element Force File |
| .EF1 | Joint Parameters and SCF data |
| .Z"n" | Peak Stresses for each wave step (overwritten for each wave) |
| .Y"ID" | Peak Stress Ranges for each each wave (overwritten for each wave). |
| .YTRF_J(d) | Tubular Joint Stresss Range Transfer Functions for all stress points |
| .YTRF_M(d) | Member Joint Stresss Range Transfer Functions for all stress points |
| .DSJ"d" | Damage Ratios for Direction reference number |
| .DSM"d" | Damage Ratios for Direction reference number |
| .EC1 | Joint can and stub wall thickness - Tubular Joints check |
| .EC2 | Joint can and stub wall thickness - Member Joints |

***Processed Output Files***

| | |
|---|---|
| Tubular Joint Summary Output | Minimum Fatigue Life at Tubular Joints |
| Tubular Joint Stress Output | Fatigue Lives at all Stress Points |
| Tubular Joint UR Graphic Plot | Damage Ratio/Fatigue Life Plot data |

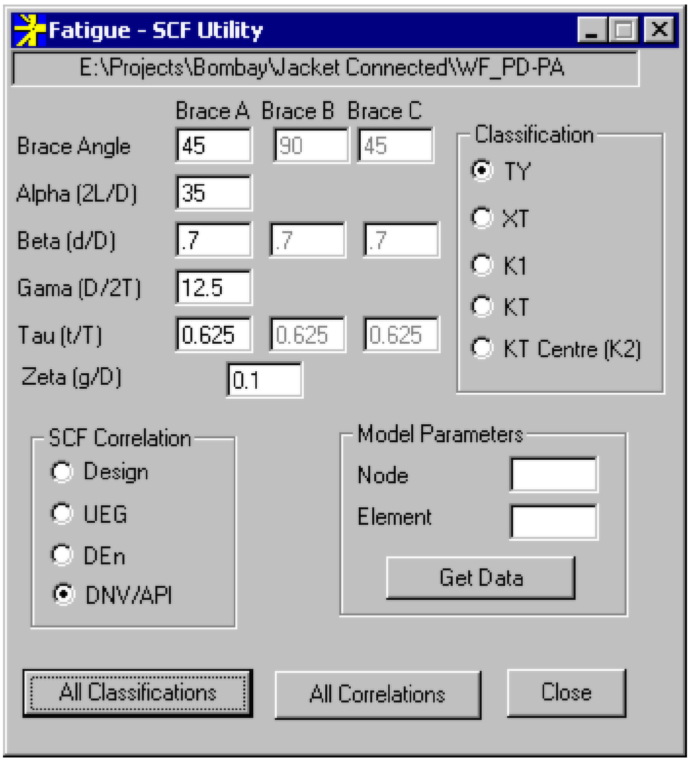
-o-

## APPENDIX D - SCF Equation Utility

The SFC utiliy is started from the WINDOWS Start Menu - All Programs - FS2000

The program can be used to evalute the SCF values using the different SCF correleations and joint classifications available in OffFat.

If a fatigue analysis has been run on a model then the utility (Get Data button) can be used to obtain the classifications and parameters used by the program at a specific element node.




-o-

# APPENDIX E - Selected File Formats

### Wave Scatter File Format (.DBS"n") for FATIG3

```
3
0.5
1
12.25 11.75 11.25 10.75 10.25 9.75 9.25 8.75 8.25 7.75 7.25 6.75 6.25 5.75 5.25 4.75 4.25 3.75 3.25
2.75 2.25 1.75 1.25 0.75 0.25
0.5 1.5 2.5 3.5 4.5 5.5 6.5 7.5 8.5 9.5 10.5 11.5 12.5 13.5 14.5 15.5
0  0  0  0  0  0  0  0  0  0  0  0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0
0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 2 0 0 0 0 0 0 0 0 0 0 0 0 0 0 19 0 0 0 0 0 0 0 0 0 0
0 0 0 0 22 37 0 0 0 0 0 0 0 0 0 0 0 0 0 5 130 16 1 0 0 0 0 0 0 0 0 0 0 0 0 55 178 18 2 0 0 0 0 0 0
0 0 0 0 0 4 341 135 7 1 0 0 0 0 0 0 0 0 0 0 4 276 705 66 10 1 0 0 0 0 0 0 0 0 0 0 43 1558 442 83 10
1 0 0 0 0 0 0 0 0 0 65 1273 2361 413 103 16 1 0 0 0 0 0 0 0 0 0 782 5278 1535 449 137 35 11 0 0 0 0
0 0 0 0 0 4411 4030 1308 418 156 60 23 1 3 1 0 0 0 0 0 0 2245 1134 454 166 78 37 24 0 0 0 0 0 0
```

The format of the data is given below

Spectrum Type (1 = ITTC; 2 = PM; 3 = JONSWAP)
Diagram Occurence Scaling Factor
Return Period (Years)
Frequnecy Blocks Hz - ω
Wave Height Block - H
Number of wave in Bins (row( ω) by row()

#### *Changing Block Intervals*

The above was created by the Scatter diagram grid editor which fixes the block intervals to those shown. The block intervals cannot be edited in the grid editor but they can be changed by editing the DBS'n' file. When the file is loaded into the the grid editior the block intervals will take the values of the edited file.

### .EF1 Files Joint and SCF Data

1 14 TY, 0.050 23.107 40.000 0.800 21.429 0.714 0.033
1.853 2.167 7.609 5.794 3.125 2.969 7.710 5.857
2 15 TY, 0.050 23.107 40.000 0.800 21.429 0.714 0.033
1.853 2.167 7.609 5.794 3.125 2.969 7.709 5.857
3 16 TY, 0.050 23.107 40.000 0.800 21.429 0.714 0.033
1.853 2.167 7.609 5.794 3.125 2.969 7.710 5.857

**No Ele Class g** θ α β γ τ ξ
4 2 TY, 0.050 16.574 40.000 0.600 20.833 0.833 0.020
**ACs ABs ACc ABc IPCc IPBc OPCs OPBs**
1.846 2.163 7.438 5.686 2.354 2.483 4.545 3.863

4 4 TY, 0.050 16.574 40.000 0.600 20.833 0.833 0.020
1.846 2.163 7.438 5.686 2.354 2.483 4.545 3.863
4 6 TY, 0.050 16.574 40.000 0.600 20.833 0.833 0.020
1.846 2.163 7.438 5.686 2.354 2.483 4.545 3.863
5 7 TY, 0.050 75.698 40.000 0.508 21.429 0.571 0.033
12.959 9.164 9.021 6.683 3.452 3.175 9.382 6.911

5 9 TY, 0.050 75.698 40.000 0.508 21.429 0.571 0.033

12.959 9.164 9.021 6.683 3.452 3.175 9.382 6.911

**ACs = Axial Chord Saddle**

**ABs = Axial Brace Saddle**

**ACc = Axial Chord Crow n**

**ABc = Axial Brace Crown**

**IPCc = In Plane Chord Crow n**

**IPBc = In Plane Brace Crown**

**OPCs = Out Plane Chord Saddle**

**OPBs = Out Plane Brace Saddle**

$\alpha$ **= 2L/D** $\beta$ **= d/D** $\gamma$ **= D/2T** $\tau$ **= t/T** $\xi$**= g/D**

## .Z Files - Peak Stresses (71)

8

1 14

1.944E+07 8.852E+06 -6.861E+06 -1.858E+07 -1.936E+07 -8.637E+06 7.207E+06 1.880E+07

1.479E+07 5.787E+06 -6.551E+06 -1.506E+07 -1.469E+07 -5.606E+06 6.815E+06 1.524E+07

4 2

-2.950E+06 3.716E+06 1.039E+07 9.981E+06 5.911E+06 3.729E+06 1.543E+06 -2.537E+06

-2.032E+06 3.781E+06 9.223E+06 9.107E+06 5.500E+06 2.512E+06 -1.039E+05 -2.813E+06

**No Ele**

4 4

8.892E+05 3.739E+06 8.99002E+06 5.940E+06 4.002E+06 8.560E+06 1.171E+07 6.359E+06 Chord

1.543E+06 2.882E+06 5.579E+06 4.753E+06 4.188E+06 7.516E+06 9.487E+06 5.645E+06 Brace

**Pt 1 Pt 2 Pt 3 Pt 4 Pt 5 Pt 6 Pt 7 Pt'n'**

4 6

8.506E+06 1.113E+07 9.791E+06 1.556E+06 -5.031E+06 -2.387E+06 4.215E+06 7.185E+06

7.790E+06 9.842E+06 8.294E+06 1.706E+06 -3.717E+06 -2.453E+06 2.412E+06 5.684E+06

5 7

-1.317E+07 -1.175E+07 -7.031E+06 -4.829E+05 2.763E+06 2.924E+06 -1.213E+06 -8.341E+06

-9.547E+06 -8.897E+06 -4.899E+06 -5.988E+05 2.188E+06 2.534E+06 -4.677E+05 -5.763E+06

5 9

-1.084E+07 -1.037E+07 -3.579E+06 5.657E+06 1.183E+07 1.121E+07 4.269E+06 -4.817E+06

-7.996E+06 -8.151E+06 -3.354E+06 3.652E+06 8.696E+06 8.756E+06 3.864E+06 -3.047E+06

## .Y Files - Peak Stress Ranges (16)

8

2 5

0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

2 4
7.293E+07 4.348E+07 1.726E+08 1.832E+08 1.293E+08 1.050E+08 6.269E+07 4.051E+07
5.319E+07 4.415E+07 1.480E+08 1.634E+08 1.193E+08 8.088E+07 3.876E+07 4.176E+07
4 6
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00

**Ele Node**
4 4
7.405E+07 1.706E+08 2.214E+08 1.180E+08 1.138E+07 2.987E+07 8.177E+07 6.730E+07 **Chord**
7.479E+07 1.552E+08 1.905E+08 1.104E+08 1.734E+07 1.919E+07 4.784E+07 4.814E+07 **Brace**
**Pt 1 Pt 2 Pt 3 Pt 4 Pt 5 Pt 6 Pt 7 Pt'n'**

6 7
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00 0.000E+00
6 4
1.481E+08 1.416E+08 7.002E+07 5.566E+07 1.261E+08 8.398E+07 2.491E+07 1.096E+08
1.296E+08 1.235E+08 7.008E+07 4.454E+07 1.038E+08 7.441E+07 1.450E+07 9.011E+07

## .DSJ(d) & DSM(d) Files - Directional Damge Ratio (1)

8
.25
1
T
2 5
1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14
1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14
2 4
2.000E-02 5.420E-03 3.265E-07 4.592E-03 1.954E-02 4.899E-03 5.031E-10 4.944E-03
9.015E-03 2.161E-03 1.435E-07 1.743E-03 8.646E-03 1.884E-03 2.707E-10 1.897E-03
4 6
1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14
1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14

**Ele No**
4 4
5.877E-05 1.667E-03 7.021E-02 4.358E-02 5.606E-03 2.236E-03 7.260E-04 4.620E-06
3.993E-06 1.164E-03 3.004E-02 2.536E-02 3.600E-03 4.183E-04 2.374E-05 8.657E-06
**Pt 1 Pt 2 Pt 3 Pt 4 Pt 5 Pt 6 Pt 7 Pt'n'**

6 7
1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14
1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14 1.050E-14

6 4

5.134E-03 4.092E-02 5.727E-02 1.635E-03 5.520E-05 4.480E-06 6.941E-04 2.022E-03

3.367E-03 2.492E-02 2.843E-02 1.106E-03 3.702E-06 8.714E-06 2.019E-05 3.753E-04

7 5

8.998E-05 5.471E-06 2.797E-07 1.010E-06 1.107E-06 2.099E-06 2.140E-05 1.283E-04

1.029E-05 8.579E-07 1.308E-07 3.232E-07 1.479E-07 4.743E-07 5.515E-06 2.165E-05

-o-

## APPENDIX F - Demonstration Example Tripod Tower

The purpose of this example are to illustrate how batch control files are set up and used. Although the OffFat-Spec utility described in Sect. 5.0 will create all the control and definition files necessary to undertake a fatigue analysis, an understanding is useful.

To run the models simple recover the model from archive. Then run the model batch file.

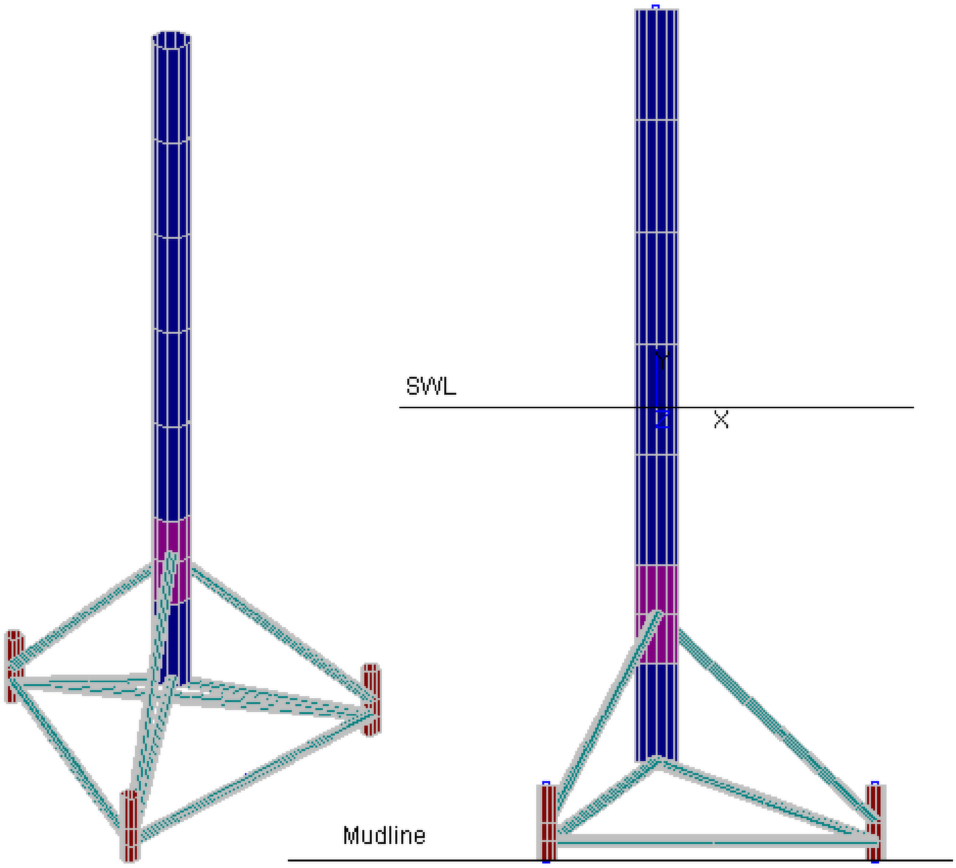
### Validation Examples

Unlike linear structural analysis it is difficult to obtain benchmark examples for such a specialised and numerically intensive process as wave induced fatigue analysis. As OffFat-Spec uses text intermediate files it is possible to trace all stages of the fatigue analysis and hence validate any result obtained.

### Example 1 - Filename SpecFatTower

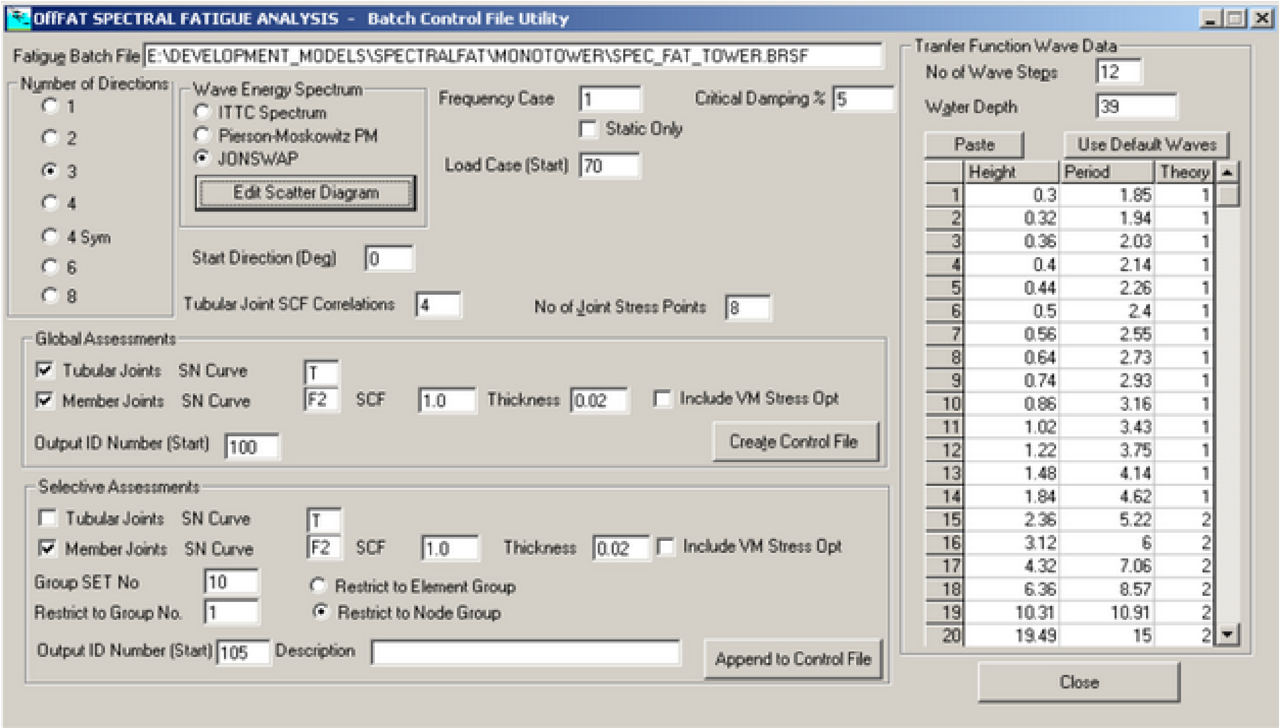
This example illustrates the spectral fatigue analysis of a Tripod Tower. The model is a simple tripod stucture located in a water depth of 39m. The mass case (frequency case) is Load Case 1. This gives the following first 6 nateral frequencies. In practice the first two would be sufficient because of the high frequencies of the other modes relative to the wave frequencies.

| Mode | Freq(Rad/s) | Freq(Hz) | Period(s) |
|---|---|---|---|
| 1 | 1.995322 | 0.3175653 | 3.148959 |
| 2 | 1.995322 | 0.3175653 | 3.148958 |
| 3 | 25.64842 | 4.082072 | 0.2449736 |
| 4 | 25.65569 | 4.083230 | 0.2449041 |
| 5 | 26.74327 | 4.256323 | 0.2349446 |
| 6 | 28.76550 | 4.578172 | 0.2184278 |




Hydrodynamic model data is defined for the model in the usual way using FS-Wave

The setting used in the OffFat input form are shown below




Being a tripod stucture 3 or 6 direction would apply loading in the principle directions. In this example nnly wave 3 directions have been considered.

The stress transfer function are to be obtainf from 20 waves of different frequency. The basis for their choice is a frequency interval of 0.25 Hz and a constant wave steepness of 1:18 based on linear deepwater theory. The Use Default Waves will generate wave data based on the criteria.

To establish the stress range from each wave the waves are stepped through the structure in 12 steps.

In total 3(dir) x 10(waves) x 12(steps) = 360 separate FS-Wave load cases are analysed.

The wave numbers and corresponding direction numbers are shown in the following table. For each wave number 12 load cases are created (L70 to L81) and overwritten by the next wave.

The full batch file to control the above analysis is given later. The various additional control files are described below.

**FS-Wave Batch Control Files (.BC)**

A full description of the .BC file is given in the FSWave User Manual.

60 WaveLoader Batch Control Files called MODEL.BCWAVF1 to MODEL.BCWAVF60 have been created .

MODEL.BCWAVF1 to MODEL.BCWAVF20 for direction 1.
MODEL.BCWAVF21 to MODEL.BCWAVF40 for direction 2.
MODEL.BCWAVF41 to MODEL.BCWAVF60 for direction 3.

The control files MODEL.BCWAVF1 file and MODEL.BCWAVF20are given below. The only differences between the files are shown in **bold**. The first is the corresponding environmental file. The second is the wave period.

```
1
MODEL.@F1
0,1.85,12
70
.
.
1
MODEL.@F20
0,15,12
70
```

### FS-Wave Environmental Files

In this example 20 environmental file are required for each wave and there are 3 wave directions. Therefore a total of 16 waves and the associated environmental files are used. The differences between the environmental files tare the wave height, wave period and wave direction. In this example the files are called MODEL.@F1 to MODEL.@F60

The environmental files corresponding to the above WaveLoader batch control (.BC) files are given below. The only differences between the files are the wave height, wave period and wave direction. These differences are shown in bold. Note that the SPECT, 1 command has to be present when generating loads for spectral fatigue analysis.

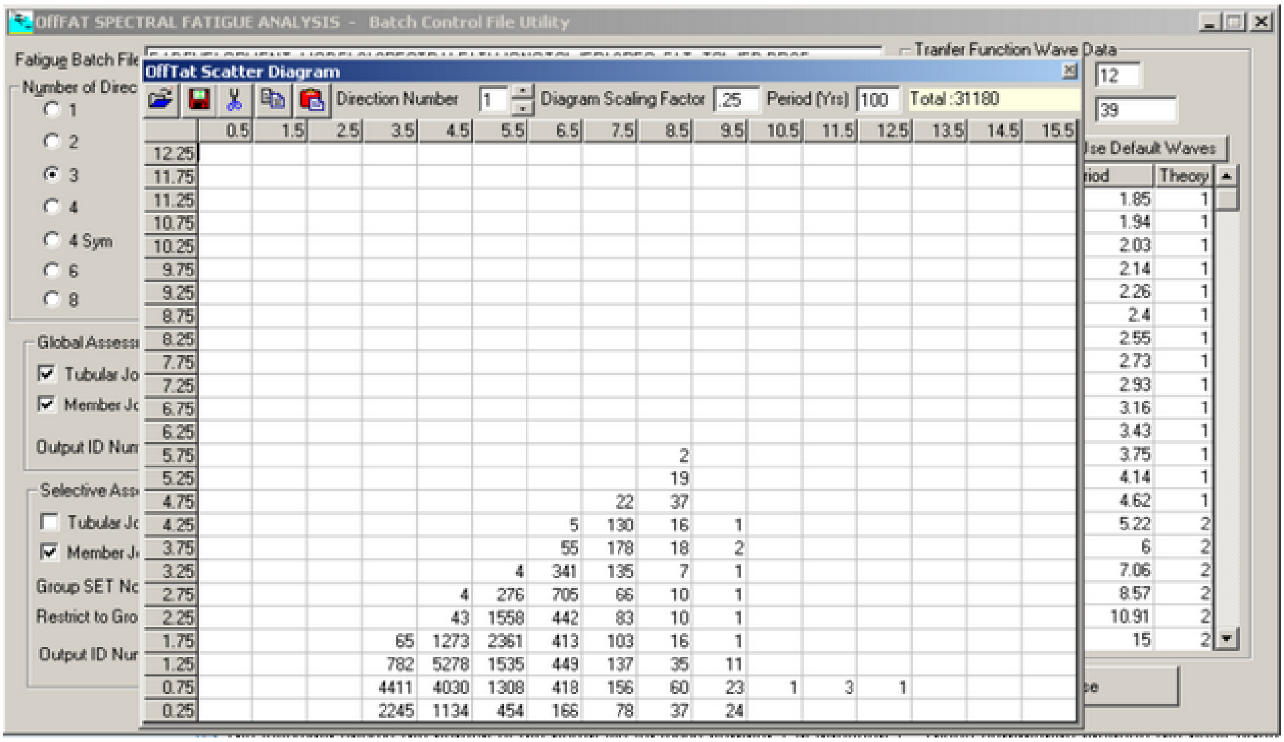
```
FS2000 Stuctural & Pipework Analysis
FSWave - Dynamic Fluid Loading for FS2000
Date:27/02/2010   Time:11:58:25
ENVIRONMENTAL LOAD DATA     Filename:
E:\Development_Models\SpectralFAT\MonoTower\SPEC_FAT_TOWER.@F1
**** WATER  DEPTH *****
WDEPTH, 39
**** WAVE DATA ****
Wave Theory
WTH, 1
Linear (Airy)
     Wave Height   Wave Period   Wave Direction
WAVE,     .3,              1.85,              0
**** Generate Spectral Loads *****
SPECTL, 1
```

```
FS2000 Stuctural & Pipework Analysis
FSWave - Dynamic Fluid Loading for FS2000
Date:27/02/2010   Time:11:58:25
ENVIRONMENTAL LOAD DATA     Filename:
E:\Development_Models\SpectralFAT\MonoTower\SPEC_FAT_TOWER.@F60
**** WATER  DEPTH *****
WDEPTH, 39
**** WAVE DATA ****
Wave Theory
WTH, 2
Stokes V Order
     Wave Height   Wave Period   Wave Direction
WAVE,     19.49,           15,                240
**** Generate Spectral Loads *****
SPECTL, 1
```

### FATIGUE Wave Data File

The wave data occurrence is defined by a scatter diagram for each wave direction. This data in used in conjecntion with the selected wave energy spectrum. In the exampl the JONSWAP spectrum is used.

The same scatter diagram has been used for all three directions. The data has been entered using the Scatter diagram grid editor as shown below.




**Main Fatigue Batch File** - Called SPEC_FAT_TOWER.BRFS

The following shows the portion of the batch file for wave number 1 in direction 1. These commands produce the peak stresses ranges for wave 16. These line are repeated 16 times to represent the 16 wave numbers. The entries shown in bold are the only differences between the repeated portions.

```
REM******************** FREQUENCY ANALYSIS
BAND 10
WAVESORT Y
LOADA 1/0/0/
DYNFRAMB
REM *************************************************** DIRECTION    1
REM *************************************************** WAVE         1
REM *************************************************** WAVE NUMBER 1
MFCOPY "WAV" "WAVF1"
WAVELOAD WAVF1
```

Generates 12 waves cases as the wave stepped through the structure

```
---------Dynamic Response
FATSPEC1 70/81/90/D/0.05/
```

Generates equivalent harmonic loads base on stepped waves

```
DYNRESP1 1/1/90/1/
```

Modal response analysis to establish dynamic response at wave frequency

```
FATSPEC1 70/81/110/S/0.05/
```

Generates equivalent harmonic loads base on stepped waves

```
DYNRESP1 1/1/110/1/
```

Modal response analysis to establish dynamic response at near zero frequency

```
---run W=Wn
LOADA C90/1/1/
OFRAME
POST6 91-92
FATIG1 8/91/2/4/0/
FATIG2 91/92/SW1/1/S/
FATIG1S 8/91/92/
FATIG2 91/2/SW2/2/S/
```

Produces stress point stress ranges for the subject wave at the wave frequency

```
---run W=0
LOADA C110/1/1/
OFRAME
POST6 111-112
FATIG1 8/111/112/4/0/
FATIG2 111/112/S01/1/S/
FATIG1S 8/111/112/
```

`FATIG2 111/112/S02/2/S/` Produces stress point stress ranges for the subject wave at near zero frequency

```
---run STATIC
LOADA C70/1/1/
OFRAME
POST6 70-81
FATIG1 8/70/81/4/0/
FATIG2 70/81/ST1/1/
FATIG1S 8/70/81/
```

`FATIG2 70/81/ST2/2/` Produces stress point stress ranges for a static response

```
---Stress ranges using static improvement
```

`FATSPEC2 SW1/S01/ST1/1/1/J/` Appends to the tubular joint stress range transfer function

`FATSPEC2 SW2/S02/ST2/1/1/M/` Appends to the member joint stress range transfer function

The above section will be repeated for the additional 19 waves in direction 1 and repeated for the other two directions. The final stages of the batch file are;

At this stage all 1 .Y"w" files have been created and the final stages is the fatigue life evaluation. This is achieved by the following command lines.

```
REM ***********************************GLOBAL ASSESSMENTS
```

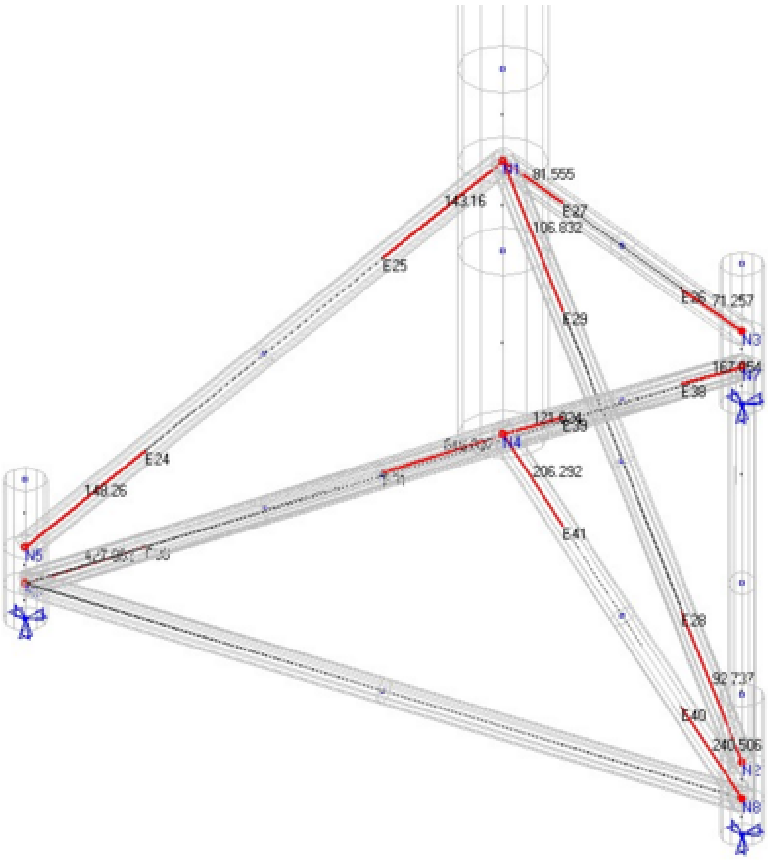
`FATSPEC3 1/20/3/1/T/` Wave Apectrum/Stress Transfer Function to establish the Stress Response Spect and Fatige Dammage

```
FATSPEC3 2/20/3/1.0/F2/
REM *******************COMBINE ALL DIRECTIONS
```

`FATIG4 S1/3/10000/100/1/` Combines all direction an creates the out data

```
FATIG4 S1/3/10000/101/2/
```

FATTOW Results



```
================================================================================
FS2000 Structural/Pipework Analyisis System  Ver 8           (c) AES Ltd

Date:08/08/2010  Time:16:42:30
================================================================================
Model File:SPEC_FAT_TOWER              Date Created:07/10/2010 Time:10:01:17

Analysis Desc. :Tripod Structure

Created/Checked:JM

Job Reference  :Example
================================================================================
              O F F A T   F A T I G U E (S P E C T R A L )   R E S U L T S
================================================================================

Scatter Diagram for Direction 1
Spectrum Type: JONSWAP
Diagram Occurence Scaling Factor: 0.25
Diagram Return Perion (Years)   : 100
Diagram Total                   : 31180
 H         3.5s  4.5s  5.5s  6.5s  7.5s  8.5s  9.5s 10.5s 11.5s 12.5s
5.75         0     0     0     0     0     2     0     0     0     0
5.25         0     0     0     0     0    19     0     0     0     0
4.75         0     0     0     0    22    37     0     0     0     0
4.25         0     0     0     5   130    16     1     0     0     0
3.75         0     0     0    55   178    18     2     0     0     0
3.25         0     0     4   341   135     7     1     0     0     0
2.75         0     4   276   705    66    10     1     0     0     0
2.25         0    43  1558   442    83    10     1     0     0     0
1.75        65  1273  2361   413   103    16     1     0     0     0
1.25       782  5278  1535   449   137    35    11     0     0     0
0.75      4411  4030  1308   418   156    60    23     1     3     1
0.25      2245  1134   454   166    78    37    24     0     0     0

Scatter Diagram for Direction 2
Spectrum Type: JONSWAP
Diagram Occurence Scaling Factor: 0.25
Diagram Return Perion (Years)   : 100
Diagram Total                   : 31180
 H         3.5s  4.5s  5.5s  6.5s  7.5s  8.5s  9.5s 10.5s 11.5s 12.5s
5.75         0     0     0     0     0     2     0     0     0     0
5.25         0     0     0     0     0    19     0     0     0     0
4.75         0     0     0     0    22    37     0     0     0     0
4.25         0     0     0     5   130    16     1     0     0     0
3.75         0     0     0    55   178    18     2     0     0     0
3.25         0     0     4   341   135     7     1     0     0     0
```

| | | | | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| 2.75 | 0 | 4 | 276 | 705 | 66 | 10 | 1 | 0 | 0 | 0 |
| 2.25 | 0 | 43 | 1558 | 442 | 83 | 10 | 1 | 0 | 0 | 0 |
| 1.75 | 65 | 1273 | 2361 | 413 | 103 | 16 | 1 | 0 | 0 | 0 |
| 1.25 | 782 | 5278 | 1535 | 449 | 137 | 35 | 11 | 0 | 0 | 0 |
| 0.75 | 4411 | 4030 | 1308 | 418 | 156 | 60 | 23 | 1 | 3 | 1 |
| 0.25 | 2245 | 1134 | 454 | 166 | 78 | 37 | 24 | 0 | 0 | 0 |

Scatter Diagram for Direction 3

Spectrum Type: JONSWAP

Diagram Occurence Scaling Factor: 0.25

Diagram Return Perion (Years)   : 100

Diagram Total                   : 31180

| H | 3.5s | 4.5s | 5.5s | 6.5s | 7.5s | 8.5s | 9.5s | 10.5s | 11.5s | 12.5s |
|---|---|---|---|---|---|---|---|---|---|---|
| 5.75 | 0 | 0 | 0 | 0 | 0 | 2 | 0 | 0 | 0 | 0 |
| 5.25 | 0 | 0 | 0 | 0 | 0 | 19 | 0 | 0 | 0 | 0 |
| 4.75 | 0 | 0 | 0 | 0 | 22 | 37 | 0 | 0 | 0 | 0 |
| 4.25 | 0 | 0 | 0 | 5 | 130 | 16 | 1 | 0 | 0 | 0 |
| 3.75 | 0 | 0 | 0 | 55 | 178 | 18 | 2 | 0 | 0 | 0 |
| 3.25 | 0 | 0 | 4 | 341 | 135 | 7 | 1 | 0 | 0 | 0 |
| 2.75 | 0 | 4 | 276 | 705 | 66 | 10 | 1 | 0 | 0 | 0 |
| 2.25 | 0 | 43 | 1558 | 442 | 83 | 10 | 1 | 0 | 0 | 0 |
| 1.75 | 65 | 1273 | 2361 | 413 | 103 | 16 | 1 | 0 | 0 | 0 |
| 1.25 | 782 | 5278 | 1535 | 449 | 137 | 35 | 11 | 0 | 0 | 0 |
| 0.75 | 4411 | 4030 | 1308 | 418 | 156 | 60 | 23 | 1 | 3 | 1 |
| 0.25 | 2245 | 1134 | 454 | 166 | 78 | 37 | 24 | 0 | 0 | 0 |

:

=============================================================================

Fatigue Output Case Number: 100

Fatigue Tubular Joints

Local SCF                        1

Global Fatigue Curve             T 12.48 3 67.09 16.13 5 0 .016 .25

Group SET                        0

Thickness for non tube joints    0.02

All Groups & Non Groups - Sorted by El Label

=============================================================================

| Ele | Node | Point | Location | Life |
|---|---|---|---|---|
| 24 | 5 | 7 | Chord | 149.385282545837 |
| 25 | 1 | 4 | Chord | 143.494667032689 |
| 26 | 3 | 7 | Chord | 70.7684052106169 |
| 27 | 1 | 2 | Chord | 80.8185298707413 |
| 28 | 2 | 7 | Chord | 91.4427839603975 |
| 29 | 1 | 2 | Chord | 104.381953014012 |
| 30 | 6 | 3 | Chord | 427.313895199183 |

| | | | | |
|---|---|---|---|---|
| 31 | 4 | 5 | Chord | 652.613706374481 |
| 32 | 7 | 3 | Chord | 1768.09648536376 |
| 33 | 6 | 8 | Chord | 2641.2053849459 |
| 34 | 6 | 8 | Chord | 2906.40229692429 |
| 35 | 8 | 3 | Chord | 3038.09167406069 |
| 36 | 7 | 2 | Chord | 2101.14934437412 |
| 37 | 8 | 3 | Chord | 2880.43323010086 |
| 38 | 7 | 3 | Chord | 163.963996087674 |
| 39 | 4 | 1 | Chord | 120.358667651016 |
| 40 | 8 | 3 | Chord | 234.263354295455 |
| 41 | 4 | 1 | Chord | 206.028400697164 |

Lowest Life is 70.7684 years

-o-