PAPER

GUIDELINES FOR STRUCTURAL INTEGRITY ASSESSMENT OF EXISTING PLATFORMS

PRESENTED BY:

ZEE ENGINEERING CONSULTANTS
ENGINEERING DYNAMICS INC.

IN ASSOCIATION WITH:
PT. WIRAZEE ADHI

PT. WIRAZEE ADHI ENGINEERING
JL. CONDET RAYA 62W
JAKARTA 13520
INDONESIA

ZEE ENGINEERING CONSULTANTS
31B, JALAN SS 15/4B
SUBANG JAYA,
47500 PETALING JAYA,
SELANGOR DARUL EHSAN,
MALAYSIA
PAPER
GUIDELINES FOR STRUCTURAL INTEGRITY ASSESSMENT OF EXISTING PLATFORMS

PRESENTED BY:
ZEE ENGINEERING CONSULTANTS
ENGINEERING DYNAMICS INC.

IN ASSOCIATION WITH:
PT. WIRAZEE ADHI

PT. WIRAZEE ADHI ENGINEERING
JL. CONDET RAYA 62W
JAKARTA 13520
INDONESIA

ZEE ENGINEERING CONSULTANTS
31B, JALAN SS 15/4B
SUBANG JAYA,
47500 PETALING JAYA,
SELANGOR DARUL EHSAN,
MALAYSIA
ACKNOWLEDGEMENTS

The following organizations have made major contributions to this paper.

1. Petronas Carigali Sdn Bhd (Malaysia)
2. Engineering Dynamics Inc. (U.S.A.)
3. Sarawak Shell Berhad (Malaysia)
4. Mobil (U.S.A)
5. Gulf (U.S.A)
6. Exxon (U.S.A)
# TABLE OF CONTENTS

1.0 INTRODUCTION

2.0 PHILOSOPHY AND METHODOLOGY
   2.1 STRUCTURAL MODEL
   2.2 JACKET INTEGRITY
   2.3 MEMBER VERIFICATION
   2.4 JOINT VERIFICATION

3.0 DATA COLLATION AND STRUCTURAL MODEL
   3.1 DATA COLLATION
   3.2 COMPILATION OF STRUCTURAL MODEL
   3.3 VALIDATION OF STRUCTURAL MODEL

4.0 LINEAR ELASTIC PUSH OVER ANALYSIS
   4.1 METHODOLOGY
   4.2 COMPUTER SIMULATION

5.0 PLASTIC COLLAPSE ANALYSIS
   5.1 BASIC PHILOSOPHY
   5.2 METHODOLOGY
   5.3 COMPUTER SIMULATION

6.0 FATIGUE ANALYSIS OF JOINTS
   6.1 SPECTRAL ANALYSIS

7.0 FINITE ELEMENT / FRACTURE MECHANICS ANALYSIS OF JOINTS
   7.1 METHODOLOGY

8.0 RISK ASSESSMENT

9.0 CODES AND STANDARDS
APPENDICES

APPENDIX A - SACS SUITE OF PROGRAMS

APPENDIX B - THEORETICAL BASIS OF SACS COLLAPSE PROGRAM
1.0 INTRODUCTION

Currently in Indonesia there are many platforms in operation which have exceeded their design/certified life. In most cases the oil companies (OPERATORS) wishes to continue to operate these platforms. Then there is a need to re-certify these platforms, as it is mandatory as per MIGAS regulations.

At the present time there are no clear cut codes, specifications nor guidelines on how to carry out such studies. This report presents a methodology for the re-assessment/re-certification of existing platforms. The design criteria recommended, and the development of software to carry out these re-assessment studies are a collection of ideas and recommendations, from many oil companies, and statutory organizations.

Nearly two (2) years, many man months of engineering and computer programming and other resources have been utilised in the compilation of this report and writing of the necessary software programs to carry out such studies. The recommended system is to the latest API guidelines.

Today all the resource required to undertake such projects are available in Indonesia. They include:

i) Expertise
ii) Computer software
ii) Experience

We as a company will be only too pleased to meet appropriate oil companies and related statutory bodies to discuss ways to localize and customize the system and, if required to improve the methodology and the technology.
2.0 PHILOSOPHY AND METHODOLOGY

The structural integrity philosophy for assessment and certification of existing offshore platforms is briefly described in this section. Detailed methodology and narrative for the major activities associated with the study are described in the following sections of this report.

For design Philosophy, please refer to Flow Chart Fig. 2.01.

2.1 STRUCTURAL MODEL

1. At the commencement of the project all available information on the platforms will be collected into a design criteria document to be adopted for the study.

2. A accurate structural model of the platform will be compiled and validated. The validity of the integrity study will depend primarily on the accuracy of model. Hence it is important that the platform model is as accurate as possible. It must be checked and verified.

2.2 JACKET INTEGRITY

Once the structural model is validated, the study will follow two (2) paths. First member checks for reserve strength, and secondly, joint checks for fatigue life of the jacket.

2.3 MEMBER VERIFICATION

Platform structural members are verified in the following manner.

1. Check for Platform Reserve Strength Ratio (RSR) of 1.32 for unmanned and 1.5 for maned platforms using linear push over analysis.

2. If this conditioned is satisfied no further analysis is necessary.

3. If the linear push over analysis is not satisfactory using Full Plastic Collapse Analysis (FPCA) check RSR of platform.
4. If the RSR value is greater than the required value no further analysis is necessary.

5. If RSR is not acceptable check platform rating and establish if is critical.

6. If not critical postpone remedial action but monitor platform. Establish critical members. Revise platform inspection schedule.

7. If critical, carry our risk assessment.

8. If the risk is acceptable postpone remedial action but monitor platform as stated in step 6.

9. If the risk is not acceptable undertake remedial action.

2.4 JOINT VERIFICATION

Jacket Joint Fatigue Life Criteria will be ascertained in the following manner.

1. Carry out fatigue life assessment for jacket using spectral method. The methodology and the criteria to be adopted for spectral fatigue analysis has to be formulated and verified.

2. Check whether the platform has joints with low fatigue life.

3. If all joints have acceptable fatigue life no action will be taken.

4. If critical joints are identified, carry out underwater inspection of critical joints to confirm cracks.

5. If no cracks are found, monitor joints and review inspection schedule.

6. If cracks are located conduct fracture mechanics / finite element analysis. Check whether the cracks will propagate, during life of platform.

7. If it is concluded that the cracks will not propagate monitor critical joints and review inspection schedule.

8. If cracks are likely to propagate check platform rating and follow from step five (5) as stated in Section 2.3 member verification.
3.0 DATA COLLATION AND STRUCTURAL MODEL

3.1 DATA COLLATION

With assistance from the OPERATOR, all data pertaining to the platform will be collected, reviewed and documented as the design criteria for jacket. These information will include:

i) As built drawings
ii) Survey reports
iii) Design manuals
iv) Technical reports
e tc.

3.2 COMPILATION OF STRUCTURAL MODEL

Based on these documents and OPERATOR approved design criteria, a space frame finite element computer model for platform will be compiled. This structural model will include jacket, piles and soil, representative super structure, all appurtenances and all applicable load data. Strict quality control procedure will be implemented to check the computer model.

The items to be checked will include the following:

1. Substructure geometry, connectivity, cans, member types and sizes.
2. Member eccentricities, offsets are to be correctly defined in the model.
3. All risers, conductors, caissons, boat landings and all the other appurtenances which attract hydrodynamic loading are correctly defined and located in the model.
4. For deck, only equivalent deck frames need to be included. Checks are to ensure that main beams connecting legs are modelled accurately, and the stiffness of the deck is adequately simulated by representative beams.
5. Restraints and releases at connections are correctly simulated.
6. All hydrodynamic parameters are accurately simulated.
7. Loads and masses are accurately defined.

8. Foundation simulation including soil data, pile make-up and connectivity to ensure the correct soil-pile-structure interaction.

9. Check all material and geometric non-linear properties and their accuracy.

10. Actual marine growth.

11. Corroded, and damaged members.

12. Any other check that is deemed necessary.

3.3 VALIDATION OF STRUCTURAL MODEL

Once the structural model is compiled and checked, a static in-place analysis including soil-pile-structure interaction will be carried out for all load cases and possible combinations.

Results of the analysis shall be compared with the results previously obtained for the corresponding load cases. The following specific results will be compared.

1. Summary of platform loads and load summary at the mudline.

2. Summary of overall displacement at pile head, jacket top and deck elevations.

3. Summary of pile loads and pile unity check.

4. Summary of jacket critical members and unity checks.

Any discrepancies detected will be investigated, and justified in case of major differences. Until and unless the current structural model is excepted by the OPERATOR analysis will not proceed to the next stages.
4.0 LINEAR ELASTIC PUSH OVER ANALYSIS

In linear push over analysis a series of elastic analysis are carried out with engineer interpretation, and manual data updates. The basic methodology is to update the jacket stiffness by manually changing the member properties and or boundary conditions to reflect the current status of the jacket i.e. at the commencement of each cycle (analysis).

As this method is highly manual oriented, it tends to be very conservative or inaccurate depending on the engineer and or the procedure adopted. Further the 'linear push over' procedure is very time consuming and uneconomic. With the emergence of Full Plastic Collapse Analysis as a tool it is highly recommend not to utilise the linear elastic method.

However, for completeness a brief description of the linear push over procedure is included in this document.
4.1 METHODOLOGY

1. Perform a series of linear elastic analysis using 100 years environmental condition, which includes wind, wave and current and their respective directional effect. Wave spreading and current blockage factors are to be incorporated.

   The critical load condition producing maximum base shear will be identified and used for incremental environment load increase.

2. Apply a load factor of 1.15 to all non environmental loads and increase the environmental load factor from 1.00 to 1.32 or 1.5 (depending on platform classification) at predetermined load increments. Load increment factors of 0.05 is generally adopted.

   Locate any overstressed members in structure for particular load increment. A member is considered overstressed when its unity ratio exceeds 1.0 with all factors of safety associated with buckling, tension, bending and for joints removed.

3. For foundation, the soil-pile-structure interaction effect will be accounted for and pile overload analysis shall be used to check the piles.

   For determination of ultimate pile capacity adequacy, the capacity curve from soil reports shall be divided by 1.13 and for compression piles the unplugged capacity is used.

4. At each load increment (cycle) the overstressed component (member, pile or joints) is degenerated and substituted by forces acting at joints. The environmental loads on the degenerated member is to be retained.

5. The procedure is repeated until the structure as a whole is incapable of accepting any further load increase. At this stage the platform is deemed collapse as the matrix solution will fail to converge. Platform may also be considered collapse (unserviceable) due to excessive rotation or deformation.

   The limits for rotation and deflection has to be specified by the OPERATOR.
Generally the values are:

Rotation = One (1) degree
Deflection = 20 inches / 100 ft of jacket length

Note:

There are a number of techniques and modelling to degenerate members at failure. The most common one are:

1. Introducing plastic hinges
2. Replacing member with its 'Collapse forces' at the two ends. In this case a non structural member have to be re-introduced to capture subsequent environmental load increments.

If the OPERATOR specifies a linear elastic collapse analysis has to be carried out the methods and techniques used to degenerate members at collapse will be submitted for approval before the commencement of work.

4.2 COMPUTER SIMULATION

Structural Analysis Computer System (SACS) can be used for linear push-over analysis. SACS has been developed by Engineering Dynamics Inc. (EDI) U.S.A. Today SACS is the most widely used computer suite for the design of offshore jackets and topsides. SACS is commercially available in this region.

API 20th edition is fully implementing in the SACS suite. This includes LRFD and NPD checks.

For details of the SACS suite of programs, please refer to Appendix A.
5.0 PLASTIC COLLAPSE ANALYSIS

'Plastic collapse' mode of assessment offers an improved design concept over the linear 'Elastic push over' theory in the analysis/re-analysis of structures. The elastic method is based on the allowable stress criteria while in the plastic method the ultimate stresses are taken into consideration. Hence the plastic mode of assessment offers an economical solution over the conservative elastic method in the re-certification of existing offshore jackets.

The basic concept of the plastic theory is as follows:

When a member reaches the ultimate stress, plastic hinges will be formed at the hot spots. This will make the structure less rigid. Additional loads due to subsequent load increments at the member gone plastic will be re-distributed to the adjacent members. This phenomena (progressive collapse of members) will continue until the structure as a whole will collapse or 'pushed over'. The ratio of the ultimate load to that of working load will be the factor of safety for the structure which a also known as Reserve Strength Ratio (RSR).

5.1 BASIC PHILOSOPHY

The fundamental objective in Full Plastic Collapse Analysis (FPCA) is to establish the Reserve Strength Ratio (RSR) of a platform at its current status to its ultimate collapse stage.

The RSR is to quantify the amount of reserve strength in a platform beyond the point of first component failure. The RSR is the ratio of base shear value at platform collapse to that of the 100 year environmental load.
The basic philosophy is as follows:

\[ R \left( \frac{Rs}{Fs}, \frac{Rf}{Ff} \right) > Fd \times D + Fe \times E \]

Where:

- \( R \) = Ultimate platform strength
- \( Rs \) = Non-linear structural strength
- \( Rf \) = Non-linear foundation strength
- \( D \) = Non-Environmental Load
- \( E \) = Environmental Load
- \( Fd \) = Load factor on non-environmental load = 1.0
- \( Fe \) = Load factor on environmental loads
  - = 1.15 for unmanned platforms not used for oil or gas storage
  - = 1.30 for all other platforms
- \( Fs \) = Material factor for structure = 1.15
- \( Ff \) = Material factor for foundation = 1.30

Rewriting the inequality above, we have

\[ R \left( \frac{Rs}{Fs/Ff \times Rf} \right) > Fd \times Fs \times D + Fe \times Fs \times E \]

Substituting the factors, the inequality will reduce to:

\[ R \left( \frac{Rs}{Rf/1.3} \right) > 1.15 \times D + 1.32 \times E \]

for unmanned platform not used for oil or gas storage.

or \( 1.15 \times D + 1.50 \times E \) for all other platforms.
5.2 METHODOLOGY

1. Perform a series of linear elastic analysis using the 100 year environmental condition which include wind, wave, current and their respective directionality effect. Wave spreading and current blockage factors are to be incorporated.

The critical load condition producing the maximum base shear shall be identified and used for incremental environmental load increase.

2. Commence plastic analysis introducing geometric and material non-linearity in the members. Also take into consideration, joint flexibility, formation of plastic hinges and full non-linearity of the soil-pile-structure interaction.

3. Apply 1.15 to all non environmental loads and increase the environmental load from 1.0 till global collapse of platform at 0.05 increments. Locate the overstressed member in the structure and also identify the respective critical load condition.

4. At this stage amend structure stiffness to take into consideration the non-linearities as described in item 2.

5. Proceed with the load increment and non-linearities as described in items 3 and 4.

6. At collapse determine the RSR value.

The platform is deemed collapse when the analysis fail to converge or jacket rotation exceeds one (1) degree.

7. A "Collapse Contour" for the platform will be constructed base on the final Platform RSR factor in its respective directions.

The number of load cases necessary for the direction of this "collapse contour" will include a minimum of at least six (6) load conditions, two each from end-on, broad side and diagonal directions.

8. If the platform collapse with RSR factor less than the required value the specific structural component shall be identified for further analysis or remedial action.
5.3 COMPUTER SIMULATION

The complete COLLAPSE analysis simulation can be carried out by using the COLLAPSE analysis module of the SACS program.

Please refer to Appendix B for SACS COLLAPSE program theoretical basis.
PLOTS FROM
COLLAPSE ANALYSIS
**FIGURE 8a** - Detailed view of the secondary crack near the fusion zone of the SW pipe spiral-seam weld (visible at right). Nital etch/20X magnification.

**FIGURE 8b** - Close-up view of the crack initiation at the tip of the corrosion pit shown in the middle of Figure 8a. Note plastic flow on both sides of the crack. Nital etch/200X magnification.
6.0 FATIGUE ANALYSIS OF JOINTS

For platform joint verification a fatigue life analysis will be undertaken. The validated structural model will be used and full spectral fatigue analysis will be carried out.

The objective of the fatigue analysis is to identify joints with fatigue life below expected values. If there are no joints with fatigue life below, the proposed operational life of jacket no further analysis is required.

6.1 SPECTRAL FATIGUE ANALYSIS

Spectral fatigue is a statistical approach for calculating the fatigue damage to a structure. Research has shown spectral fatigue analysis to yield more realistic and liable results than deterministic analysis.

The spectral fatigue approach utilizes wave spectral and transfer functions which allows the relationship of the ratio of structural response to wave height as a function of wave frequency to be developed for the wave frequency range. Therefore, spectral fatigue accounts for the actual distribution of energy over the entire wave frequency range.

In spectral fatigue analysis, the wave height-stress range relationship is defined by transfer functions. Not all waves of the fatigue environment are used to develop the stress ranges and transfer functions. It is necessary to select only the seastates required to yield an accurate and sufficiently detailed transfer function to represent the response of the structure to the complete wave spectrum for all wave approach directions. These transfer functions define the ratio of cyclic stress to wave height as a function of wave frequency (usually for one wave direction at a time). In this case, the input is the elevation of the sea at a point above its undisturbed position (wave height) and the responses are the brace stress at the connections with the chords.

The SEASTATE program module is used to obtain the hydrodynamic force coefficients for the structural system. Using this data along with the structural dynamic characteristics of the jacket structure as obtained with DYNPAC, the WAVE RESPONSE program is used to obtain steady state system responses for various wave/height combinations from each wave direction. These response are used to determined brace stresses throughout the structure.
The FATIGUE program uses this information and brace stress concentration factors (SCF's) to calculate stress transfer functions for eight positions around the circumference, and on each side, of each brace/chord connection in the structure. These SCF's are automatically calculated based upon structural geometry and member internal loads according a set of equations specified by the user. A number of different sets of these equations for determining SCF's are already available in the FATIGUE program. The user is also able to enter their own SCF's if this is required.

These stress transfer functions and the wave spectra specified are used by the FATIGUE program to calculate the stress/cycle data for each of the sixteen points around and on each side of the brace chord connections of the structure. The FATIGUE program then uses this data with the specified S-N curves to determine the fatigue damage of each brace/chord connection in the structure. The Palmgren-Miner accumulation of damage hypothesis is used to determine the fatigue life of each connection of the structure.

Using this method of damage calculation, it is possible to determine what amount of fatigue life remains for the jacket and to also determine what are the most critical joints of the jacket. This information can be used to determine the need for, the locations throughout the structure and the type of inspection to be conducted.
7.0 FINITE ELEMENT / FRACTURE MECHANICS ANALYSIS OF JOINTS

On identification of the possible 'hot' joints following Finite Element / Fracture Mechanics Analysis will be undertaken for these joints.

7.1 METHODOLOGY

A consistent method based on either stress intensity factor or the CTOD method shall be used. Should CTOD method be used. A realistic CTOD value will be proposed for OPERATOR'S approval, as there is no available test data.

The platforms anomalies data supplied by OPERATOR shall be used as a basis for the evaluation of the joint adequacy and any assumptions made to supplement the data for the analysis shall have the OPERATOR's approval prior to analysis execution.

The methodology described in BSI PD6493 (preferably using Level II and III) or DNV codes as given in Section 9.0 shall form the basis to check the joints, in term of crack propagation evaluation, acceptance of defect and fatigue life of the joint.

The assessment would require global and local stress analysis to ascertain the stress range of the joint in an uncracked situation. The stress components critical for fracture evaluation shall be that perpendicular to the plan of the defect. The loads used shall be that which derived the stress ranges i.e. due to wave loading alone. Spectral approach will be used for fatigue damage analysis.

Should no defect data be available, finite element analysis shall be done on the joints identified by the Company to re-asses the Stress Concentration Factors used and thus derive the new fatigue life of the joints in question.
8.0  **RISK ASSESSMENT**

Risk assessment basis and data are generally unique to each operator as it is targeted at the various remedial actions proposed to strengthen the structure such that the RSR will not fall below 1.32 or 1.50 respectively at the point of structure collapse.

A generalised method is described in this section.

**Methodology**

The basic equation guiding the risk assessment is given below:

\[ E(t) = E(i) + C(f) \times Pa \times N \]

where

- \( E(t) \) = Expected total Cost
- \( E(i) \) = Expected Initial Cost
- \( C(f) \) = Future Cost associated to collapse
- \( Pa \) = Annual probability of failure
- \( N \) = Number of years of remaining service life.

\( E(t) \) shall be computed for each and all of the feasible/practical options which could improve the RSR of the structure. Study will also include the "Do Nothing" option. For a manned platform, the RSR of 1.50 is minimum. Remedial action shall be carried out should the platform's RSR falls below this value.

The environmental load level at which collapse occurred (as per criteria of collapse in the analysis) shall be translated into the corresponding return period in a deterministic manner.

The sensitivity of different cost scenarios shall be investigated by considering the upper and lower bound values.

<table>
<thead>
<tr>
<th>Cost Options</th>
<th>Upper Bound Consideration</th>
<th>Lower Bound Consideration</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Plug and abandon wells</td>
<td>Not Required</td>
</tr>
<tr>
<td>2</td>
<td>Salvage Structure and Remove</td>
<td>Not Required</td>
</tr>
<tr>
<td>3</td>
<td>Replace Structures</td>
<td>Strengthening only (for each strengthening option, cost and Pa shall be determined)</td>
</tr>
<tr>
<td>4</td>
<td>Redrill wells</td>
<td>Not Required</td>
</tr>
<tr>
<td>5</td>
<td>Loss of Production=3 years</td>
<td>Loss of Production=6 mths</td>
</tr>
<tr>
<td>6</td>
<td>Environmental Cost</td>
<td>Not Considered</td>
</tr>
</tbody>
</table>
To predict crack propagation the Paris equation shall be applied.

\[ \frac{da}{dN} = C(dK)^m \]

Where

- \( dK = K_{\text{max}} - K_{\text{min}} \)
- \( N = \) Number of cycles
- \( a = \) Crack depth
- \( C = \) Crack growth parameter
- \( m = \) Material constant

The stress intensity factor is:

\[ K = \sigma g \sqrt{\pi a} \]

Where:

- \( \sigma = \) Nominal stress in the member normal to the crack
- \( g = \) A factor depending on the geometry of the member and rack

The crack growth is found by integrating the Paris’ equation as follows:

\[ Nb = \int \frac{d\alpha}{(g \sqrt{\pi a})^m / (C/W^{m/2-1} \Delta \sigma^m)} \]

Where

- \( Nb = \) Number of cycles necessary to increase the defect from the initial size, \( \alpha_0 \), to final effect size \( \alpha_f \).
- \( \Delta \sigma = \) Stress range
- \( \alpha = \frac{a}{W} \)
- \( W = \) Characteristics length, i.e. member thickness

To determine the number of stress cycles to cause a certain amount of crack growth in a specific geometry the integral:

\[ I = \int \frac{d\alpha}{(g \sqrt{\pi a})^m} \]

has to be evaluated for the specific geometry and boundary conditions.

Based on the results of the analysis above, a joint reability index is to be ascertained and a comparison shall be made with the real inspection results to derive an approximate inspection schedule for joints.
9.0 CODES AND STANDARDS

The codes and standards to be used for platform verification are as follows:

API RP 2A : American Petroleum Institute,  
"Recommended Practice for Planning, designing and  
Construction of Fixed Offshore Platform" both for WSD  
and LRPD, latest edition.

AISC : American Institution of Steel Construction,  
"Specification for Structural Steel Buildings", latest  
edition.

AWS D1.1 : American Welding Society,  

DNV : Det Norske Veritas,  
"Rules for the Design, Construction and Inspection of  
Offshore Structures and its relevant appendices", latest  
edition.  
"Classification Note 30.2 - Fatigue Strength Analysis for  
Mobile Offshore Units", latest edition.

BSI PD6493 : British Standards Institute, PD6493  
"Guidance on Some Methods for the Derivation of  
Acceptance Level for the Defects in Fusion Welded  

OPERATOR’S specifications and codes if applicable.

Precedence

Where the code is deficient and or in conflict with the procedure/criteria  
contained herein, the following order of precedence shall apply:

- This specification  
- OPERATOR’S specs  
- API RP 2A, latest edition  
- All other relevant industry’s codes  
- Consultant’s recommended code
APPENDIX A

SACS SUITE OF PROGRAMS
PRECEDE: Interactive full screen color graphics modeler

- Model generation capabilities include geometry, material and section properties and loading.
- Automatic input error detection.
- Maintains data backup.
- Beam and/or finite element modelling including plate and shell elements.
- Automatic offshore jacket & deck generation.
- Automatic cartesian, cylindrical or spherical mesh generation.
- Automatic load generation including gravity, pressure and skid mounted equipment loads.
- SEASTATE data generation capabilities.
- Extensive plotting and reporting capabilities.
- UC parameter generation including K-factors and compression flange unbraced lengths.

SACSDGN: Interactive data generation for all programs

- Full screen editor which labels and highlights data fields, and provides help for data input.
- Automatic data checking.
- Form-filling data input available as well as full screen mode.

SEASTATE: Environmental loads generator

- Full implementation of API 20th edition.
- Supports five wave theories.
- Current included or excluded.
- Generates load due to wind, gravity, buoyancy and mud flow.
- Marine growth, flooded and un-flooded members.
- Diameter, Reynolds number and wake encounter effects dependent $C_d$ and $C_m$.
- User defined waves.
- Forces on non-structural bodies.
- Automatic wave positioning for max/min base shear or overturning moment.
- Deterministic and random wave modeling for dynamic response.
- Member hydrodynamic modeling for static and dynamic analysis modelling.

SACS IV: Static Beam & Finite element analysis

- Beam elements including tubulars, tees, wide flanges, channels, angles, cones, plate and box girders and stiffened cylinders and boxes.
- Solid & plate elements (isotropic & stiffened).
- Isoparametric 6, 8 and 9 node shell elements.
- Library of AISC, UK, European, German, Chinese and Japanese cross sections.
- Member, plate and shell local and global offsets.
- Beam and finite element thermal loads.
- Elastic supports defined in global or reference joint coordinate system.
- Specified support joint displacements.
- Up to 400 load cases.

POST: Beam & Finite element code check and redesign

- Beam and plate element code check/redesign.
- API (incl. 20th edition), AISC, LRFD, NPD, DNV, BS5950 and Danish DS449 code check.
- Detailed and summary reports.
- Hydrostatic collapse analysis.
- Automatic member redesign.
- Updates model with redesigned elements.
- Modify code check parameters.
- Load combination capabilities.
- Supports codes from 1977 to present.
POSTVUE: Interactive graphics postprocessor

- Interactive member code check and redesign.
- Display shear and bending moment diagrams.
- Display deflected shapes for static and dynamic analyses.
- Color plate stress contour plots.
- User control of all code check parameters.
- Code check & redesign by individual or group of elements.
- Supports same codes as POST module.
- Extensive reporting and plotting capabilities.
- Color coded results and unity check plots.
- Creates updated input model file for reanalysis.
- Labels UC ratio, stresses and internal forces on elements.

REVIEW: Interactive code checking and plotting

- Joint can code checking and reporting.
- Full reporting and plotting capabilities.

COMBINE: Common solution file utility

- Combines dynamic and static results from one or multiple solution files.
- Formats solution files for transfer between different types of computers.

LDF: Large deflection analysis

- Iterative solution for geometric non-linearities.
- Solves plate membrane problems.

COLLAPSE: Non-linear collapse analysis

- Linear and non-linear material behavior.
- Incorporates non-linear springs & superelements.
- Sequential load stacking capability.
- Activate and deactivate elements.

RISK: Ultimate Capacity and Failure Risk analysis

- Determines ultimate strength of each joint, member and/or pile in the structure.
- Calculates failure probability of each element.

DIFFRACT: Diffraction Wave Analysis Interface

- Converts SACS model and wave information to wave diffraction program model.
- Creates all input required for wave diffraction analysis.

- User control of load path and connection type.
- Simplified joint can fatigue analysis.

- "Worst case" combination of dead loads with earthquake response.
- Superimposes mode shapes.
- Determine extreme wave loads from input spectra.

- Accounts for P-Δ effects.

- Plastic material properties determined automatically or defined by user.
- Load cases may contain loading and/or specified displacements.

- Includes ultimate pile capacity, excessive pile, member and joint stress as failure criteria.
- Reports joint and element order of failure.

- Converts frequency and wave direction dependent coefficients into SACS transfer functions.
- Transfer functions include real and imaginary portions for fatigue or extreme wave analysis.
DYNPAC: Dynamic characteristics

- Householder-Givens solution.
- Guyan reduction of non-essential degrees of freedom.
- Lump or consistent mass generation.
- Automatic virtual mass generation.
- Complete SEASTATE hydrodynamic modeling.
- User input distributed and concentrated mass.
- Ability to consider loading in model as mass.
- Full 6 DOF modes available for forced response analysis.

WAVE RESPONSE: Dynamic wave response

- Deterministic and random waves.
- Pierson-Moskowitz and JONSWAP spectra.
- Fluid-structure relative velocity, acceleration and displacement.
- Buoyancy dynamic loads included.
- "Modal Acceleration" & non-lin. fluid damping.
- Closed form steady state response in the frequency domain.

- Stress, internal load, base shear and overturning moment transfer function plots available.
- Full coupling with FATIGUE program.
- Elastic dynamic response of floating structures including stingers.
- Input and output Power Spectral Densities with Probability Distributions.
- Zero crossing and RMS responses.

DYNAMIC RESPONSE: General dynamic response and earthquake analysis

- Frequency domain analysis.
- Time history, response spectrum or PSD base driven input.
- Time history and harmonic force driven input.
- SRSS, CQC and linear modal combinations.
- API response spectra library and user input spectra.
- Earthquake time history library.

- Wind spectral loading capability.
- Structural and fluid damping.
- Response spectrum output at any joint.
- Vibration analysis with multiple input points with user specified frequencies and phasing.
- General periodic forces decomposed by Fourier analysis (eg., gas torques).
- Ice dynamics and Impact load analysis.

FLOTATION: Jacket flotation and upending analysis

- Color coded snapshots of each upending step.
- Stability and upending analyses.
- Initial floating and on bottom positions provided.
- Upending steps can include multiple commands.
- Dual hook capabilities.
- Buoyancy tanks, valves, user specified buoyancy and weights and hydrodynamic overrides.
- Properties, forces and positions plotted vs. step.
- Upending forces including gravity, sling loads, buoyancy, and buoyancy tank loads generated for any step of the upending sequence.
- Upending phase summary reports including pitch, roll, and yaw angles, mudline clearance etc.
LAUNCH: Jacket launch analysis

- Full 3D launch analysis.
- Time history of jacket and barge motions.

TOW: Transportation inertia load generator

- Input motion for six degrees of freedom.
- Output location for selected points.
- Automatic weight calculation.

GAP: Non-linear analysis with one-way elements

- Accurate simulation of loadout or transportation analysis using one-way elements.

PRECAD: Full screen drafting and detailing

- Input from SACS design model.
- Utilizes standard views from PRECEDE.
- Place views on drawing sheet interactively.
- User controlled tubular detailing.
- Automatic structural and joint can drafting.
- Tubular and wide flange connection details.
- DXF file format supported.
- Tubular joint lofting data.
- Fabrication drawings including bevels, API, AWS or user input.
- Templates with member end reference.
- Exact C.G., weight & weld volume calculation.
- Coupled to tubular cutting equipment.

MTO: Material take-off, weight control and cost estimation

- Member lengths including cuts.
- Steel tonnage and C.G. location.
- Material list, cost est. & weight control reports.

- Weld volume requirements and cost.
- Required protective anodes and cost
- Surface area calculations by elevation.

SACS3D: 3-D structural visualization & tubular joint mesh generator

- Screen and hardcopy plots.
- 3D hidden line drawings.
- Individual joint views.
- Automatic finite element joint mesh generation.
APPENDIX B

THEORETICAL BASIS OF SACS COLLAPSE PROGRAM
SACS COLLAPSE PROGRAM

The SACS Collapse program is a result of combining an enhancing the SACS IV, PSI and Large deflection (LDF) programs into a new non-linear structural analysis system for offshore platforms. EDI has developed in the Collapse program an easy to use unique solution procedure that uses a large deflection, interactive, tangent direct stiffness solution technique to solve for the geometric and material non-linearities associated with the ultimate load capacity of a structure. This new program is fully integrated into the SACS suite of programs and uses as input the same structural model data used for standards SACS IV/PSI analyses. With a small amount of control information a complete plastic collapse can be performed by an engineer.

Members in the Collapse program are automatically descritized by using sub-segments along the length and sub-areas to define the cross section. Plate finite elements are automatically sub-layered through the thickness. Material properties default to perfectly elastic/perfectly plastic. Specified stress-strain relations based on sub-layered material properties and including strain hardening are optional. Tubular connection flexibility, capacity and failure are calculated empirically. The pile foundation can be represented using the same mathematical procedure found in the SACS PSI program. The pile is modeled as a beam-column on a non-linear foundation (soil). The pile finite element model is the same as for members described above. Load incrementation is completely controlled by the user and includes multiple load cases with multiple load increment step size. Converge is controlled through displacements and maximum iterations per load increment. The Collapse View program allows the user to display the analysis procedure graphically including color coded plasticity, hinges and connection failures.
PROGRAM FACILITIES

A. MEMBER

Member elements are divided by default into 8 sub-segments. The user has control over the number of sub-segments used for each member in the analysis. Each sub-segment is treated as a separate member for stiffness calculation. The complete member is treated as a superelement with the intermediate nodes along the member being reduced as the stiffness is calculated. The deflected shape of the member is represented by all subsegments.

Member cross sections are represented by 8 to 16 sub-areas depending on cross section type. As an example, tubular cross section is divided into 12 sub-areas or arc lengths. Each sub-area (arc length) is checked for plasticity using Von Mises stresses during each stiffness iteration. When a sub-area to reduce the load to the elastic limit. This allows for the gradual plastification of the member cross section. Permanent set and strain hardening are included for each sub-area after plasticity occurs. When all sub-areas of the cross section become fully plastic, a temporary hinge is formed for the sub-segment of the member containing that sub-area. The strain history is recorded for each sub-area which allows the member element to retain plastic deformation and residual stress. Distortion of the cross section during loading is not included in the stiffness analysis. Hinges along the member length can form from other causes. Local buckling of the member cross section is calculated empirically and is treated as permanent hinge in the member. Connection capacity exceedance at the member ends is calculated empirically and a permanent hinge is placed at that location. Other cross section types are treated similarly.

An iterative large deflection solution is completed for each member to generate the member elemental stiffness. The stiffness solution includes the end displacements from the global structural analysis and the distributed loads along the length of the member. During the solution iteration process for the stiffness of the member the sub-segment properties are modified due to plasticity and local buckling along the length and through the cross section. Also, the end stiffnesses are modified due to connection plasticity or failure. Member buckling is an intrinsic part of the member large deflection solution. The number of buckling modes for a member is limited to the number of sub-segments used for that member.
B. PLATES

Flat plate elements are divided into 5 sub-layers thorough the thickness. Each sub-layer is treated as a separate plate for stiffness calculation. The complete plate is treated as superelement with the intermediate nodes being reduced as the stiffness is calculated. The plate element is not segmented along its surface. Each sub-layer can become plastic and self equilibrating forces are added to that sub-layer to reduce the load to the elastic limit. Plate buckling and snap through are included in the large deflection finite element mesh solution. Also, the strain history of each sub-layer is recorded which allows the plate element to retain plastic deformation residual stress.

Tubular connection flexibility is modeled empirically using Efthymiou formulas. This optional feature inserts a set of springs between the member ends and the connecting nodes. When this option is selected the member/connection rotational failure mode is also calculated. For tubular member the failure mode is based on the Marshall and Gates procedure. When a failure mode is reached, the member stiffness is permanently removed from the structure. Connection yielding is modeled as a permanent hinge and is calculated using API-LRFD ultimate strength formulas.

C. CONNECTIONS

D. LOADING

The loads are applied to the model in increments specified by the user. Any number of load cases can be used during the load incrementation and the increment step size can be changed for any load case during the analysis. Load factors can be increased and decreased the total load for any load condition. Any member, plate or nodal load type available in SACS IV can be used in the Collapse program. This includes concentrated and distributed loads and specified nodal displacements.
E. PILE / SOIL

The Collapse program accepts pile / soil input from the SACS PSI program. Soil data is input as lateral P-Y curves, axial T-Z curves and end bearing Q-Z curves. Soil properties can vary with pile penetration. The pile is represented as a set of segmented members as described in the MEMBER section above. The user has control over the number of sub-segments used for the analysis with 100 sub-segments being the default values. A large deflection beam-column supported on non-linear springs analysis is performed. Pile plasticity and hinges are calculated using the same method described in the MEMBER section above. The effective stiffness is calculated at the pilehead and this is added to the platform stiffness for each solution iteration of the entire structure. The pile stiffnesses are calculated independently for each pile.
GENERAL SOLUTION PROCEDURE

The collapse program uses a double level iteration solution procedure to account for elemental and global non-linearities. The following is a description of the complete non-linear analysis procedure.

Initial Solution

For the initial solution for the first load increment the following procedure is used:

1) All structural elements are assumed to be linear.
2) Pile stiffness is based on zero lateral displacement and rotation at the pilehead.
3) Deflections are calculated for all nodes.

Iterative Solution

For all subsequent solution iterations the following procedure is used:

1) Member and deflection from the previous global solution are used as boundary conditions for the non-linear member stiffness solutions.
2) Members are divided into 8 sub-segments.
3) Member cross sections are divided into 8 to 16 sub-areas.
4) Connection stiffnesses are calculated.
5) Distributed and concentrated loads are applied along the members.
6) A large deflection, iterative, tangent direct stiffness solution is performed for each member and plate element.

a) Partial plastic cross section can develop for each sub-segment or sub-layer.

b) Total plastic cross sections can develop and are treated as temporary hinges for each sub-segments or sub-layer.

c) Member local buckling can develop and is treated as permanent hinge for each sub-segment.
d) Member connection plasticity can develop and is treated as a permanent plastic hinge.

e) Member connection failure due to excessive rotation can develop and is treated as permanent failure.

f) Strain histories are recorded to retain plastic deformation and residual stress.

7. Each pile stiffness is calculated using a large deflection beam column solution supported by non-linear springs. Pile plasticity and hinges are treated the same as the members.

8. A global solution is performed and new deflections are calculated for all nodes.

9. Steps 1 through 8 are repeated until the convergence requirements are met and equilibrium conditions are satisfied.

10. The next load increment is applied and a global solution is performed using the final stiffness matrix from the previous converged load increment.

11. Steps 1 through 10 are repeated until one of the conditions below is met:

   a) All load increments are complete.

   b) User specified global maximum deflection is exceeded.

   c) Excessive iterations due to global collapse.

The complete incremental history of the structures displacements, loads, plasticity, buckling and Connection failures is retained for all elements to be processed by the Collapse View program.