Release 5.1 Highlights

This document contains a brief overview and highlights of the latest release of the SACS system.

SACS Release 5.1 Overview

With the release of version 5, the SACS™ system was redeveloped for the Windows™ 95/98/NT environment. Release 5.1 concentrates on improving the usability and efficiency of the system by providing a new more recognizable interface, giving new selection capabilities, more file naming flexibility, improved project features and elimination of ancillary input files.

In addition, the system has been expanded to accept alpha-numeric joint names which, in conjunction with the alpha-numeric load case names added in Release 5.0, allows for unlimited model sizes.

New development has also concentrated finite element capabilities. Through improved finite element implementation and strategic partnership with finite element modelers and results viewers such as FEMGV by Femsys Limited, the SACS system can now meet the FEM needs of the offshore structural analysis community.

SACS System Enhancements

System Interface

- The SACS front end has been redesigned to use an interface found in most Windows applications.
- New file naming capabilities allows for unlimited number of characters for file types, program names and file ID.
- Project data is now readily accessible and allows for modification and deletion of project data.
- Sub-tasks which allow items to be further grouped within a project have been added.
- Input files for many intems such as Joint Connection Check, extracting mode shapes, wave response, earthquake, etc. have been made optional.
- Plot soil and create super element icons have been added.
- System Accessories pane contains utilities to view system installation data, e-mail EDI directly, update keys and view key contents.

Interactive Capabilities

- The selection capabilities have been expanded to allow for selection by selected elements, elements in a window, on the screen, all elements or elements in a group.
- Details and properties of an element is available by right clicking on the element.
- Tooltips that show the joint, member or plate name are available.
- Toolbars in the interactive modules are customizable.
- Interactive modules can register generated files to the project.
- Solid and shell elements now supported by interactive modules.
- X-brace and K-brace load case dependant buckling length data is supported.

General System Enhancements

- Alpha-numeric joint names allow more versatility in naming nodes as well as allows for unlimited model size.
- Long filenames as well as filenames with blanks are supported
- Gap element data can be specified directly in the model.
- Ring spacing can be specified directly on the member.
- Enhanced tee and angle sections that allow fully braced stem or leg have been added.
- Norsok N-004 code and ABS 2000 wind criteria has been added.

- Added new plate temperature load capabilities.
- Allows P-delta analysis in conjunction with a PSI analysis.
- Includes option to generate buoyancy of elements below the mudline when generating self weight and buoyancy.
- Added the ability to ignore self weight of structure when generating inertia loading for tow analysis.

SACS Finite Element Overview

New finite element technology incorporates DKT theory which results in convergence of plate models to theoretical values with much courser mesh, offers improved stress computation capability including results at each Gauss point in addition to the center point and allows maximum stresses to be determined wherever they occur within the plate. It also offers improved plate element loading methods based on element elasticity and strain energy increase such as plate distributed loading and point loading and gives results at each Gauss point.

FEMGV, now offered in conjunction with the SACS suite, is a powerful and versatile interactive system used for generating finite element models and viewing analysis results. It enables easy creation of complex models to which information governing mesh control, material data, boundary conditions may be attached. FEMGV contains advanced functionality enabling generation of sophisticated shell and brick element models as well as automatic tetrahedral meshing capabilities which allows large multifaceted bodies with penetrations and voids to be created easily.

Using SACS and FEMSYS

Modeling

- Ability to model items such as tubular connections, semi-submersibles, ship hulls, lift eyes, etc. In FEMGV now offered as part of the SACS suite.
- Model can be passed between SACS and FEMGV.

Post Processing

- Results information can be passed between SACS and FEMGV.
- Post processing of plate elements using SACS.
- Post processing of shell and solid elements now available through FEMGV.

Future Development

SACS System Database

The development of the SACS database which will allow project data to be stored in an ODBC compliant database format continues. This capability will allow results for numerous model configurations to be compared and analyzed.

Compliant Tower

Various improvements and enhancements have been implemented to allow design and analysis of Compliant Tower structures. The attached development table summarizes the development along with the status of each item.

Diffraction Wave

The SACS system is interfaced to Wamit and Mora wave diffraction packages and has been used extensively for lift studies as well as transportation and fatigue studies of barges. Interface with a new wave diffraction program and development of a new motion study module is under way, in addition to enhancing the current capabilities to allow analysis of floating structures such as semi-submersibles and other hulled structures.

Compliant Tower Development Status

Item	Description	Module	Initial Check	Final Check
a.	P-Delta effects	General	Done	Required
b.	Time history integration for random waves	Wave Resp	Done	Required
c.	Random Wind Analysis	Wave Resp	Done	Required
d.	Random Wind & Random Wave Combination	Wave Resp	Done	Required
e.	Wheeler stretching for random waves	Wave Resp	Done	Required
f.	Selecting critical time points based on user specified criteria	Global Load	Done	Required
g.	Allow numerous random wave seeds to be input and executed as one analysis	Wave Resp		
h.	Combine response for all of the seeds into one response file	Wave Resp		
i.	Automatically create the wave response input file used to generate the EQS loading based on the critical time points	Global Load		
j.	Generate bending moment and shear envelopes	Global Load		
k.	Run the equivalent static load cases through a large deflection analysis	Collapse		
l.	Add ability to specify ring stiffener spacing directly on members for hydrostatic collapse check	General		
m.	Automate the analysis cycle	Executive	Done	Required