SACS™

Design and analysis software for fixed offshore structures

SACS is an integrated finite element structural analysis package for the design of fixed offshore structures, such as oil and gas platforms, wind farms, and topsides of FPSOs. Offshore engineers around the world have used SACS for 50 years, and many energy companies specify their engineering firms to use SACS software across the lifecycle of offshore platforms. Because every project has different needs, there are four flexible SACS options:

SACS Offshore Structure

This application contains capabilities required for static structural analysis. It includes:

- The interactive graphics modeler with advanced 3D capabilities
- SACS IV Solver and interactive graphics 3D post-processor
- Combine
- Gap
- Tow
- LDF large deflection
- Automatic model generation
- Beam and finite element capability
- Steel code check and redesign
- Inertia and moving load generation
- Tension/compression nonlinear elements with initial gap
- Load case combination
- Linear large deflection analysis
- Full output report and plotting capabilities

SACS Offshore Structure Advanced

This application contains the capabilities of SACS Offshore Structure and additional capabilities required for typical topside and deck analysis. It includes:

- Topsides wind loading
- Wind and gravity load generation

SACS Offshore Structure Ultimate

This application contains contains the capabilities of SACS Offshore Structure Advanced and additional capabilities required for typical offshore jackets, wharfs, and dolphin structures. It includes:

- Seastate
- Joint can
- Pile
- Environmental load generation
- Tubular connection check
- Single pile/soil interaction
- Inertia and moving load generation

SACS Wind Turbine

SACS Wind Turbine is comprised of SACS Offshore Structure Ultimate, Pile Structure Design, SACS Collapse, and Fatigue Ultimate. The package also contains the SACS interfaces to the Bladed and OpenFAST wind turbine aeroelastic modules.

Key components

Modeling

- Precede graphical modeler
- Datagen intelligent editor
- Gap elements
- Superelements

Loads

- Seastate wave, wind, current
- Buoyancy, mud flow
- Gravity, inertial
- Skid, moving

Analysis

- SACS IV Solver
- Large deflection (LDF)
- PSI pile/structure analysis
- Liquefaction
- Wind turbine analysis
- Dynpac modal analysis
- Collapse
- Collapse advanced

Design

- Combine solution files
- Post offshore code design
- Concrete design
- Postvue graphical redesign
- Joint can
- Interactive fatigue
- Dynamic fatigue
- Wave fatigue

The following add-on modules can be combined with SACS Offshore Structure Ultimate for comprehensive offshore capabilities.

SACS Pile Structure Design: Soil/pile/structure interaction analysis

This nonlinear add-on package permits nonlinear soil/pile/structure interaction analysis of fixed offshore structures with multiple fixed supports using the PSI program modules. It requires the use of SACS Offshore Structure, SACS Offshore Structure Advanced, or SACS Offshore Structure Ultimate.

SACS Collapse: Plastic nonlinear add-on

Perform advanced nonlinear plastic analysis using SACS Offshore Structure, SACS Offshore Structure Advanced, or SACS Offshore Structure Ultimate.

SACS Fatigue Ultimate: Fatigue with wave response and dynamic response

This package contains the modules required to perform any dynamic deterministic, time history, or spectral fatigue analysis. It also includes modules for general dynamic and seismic analyses. SACS Fatigue Ultimate contains SACS Fatigue, Interactive Fatigue, Dynamic Response, and Wave Response. It requires the use of the SACS Offshore Structure, Offshore Structure Advanced, or Offshore Structure Ultimate.

The following software modules extend the functionality of any of the three offshore structure packages and are either included within those packages or available as add-on modules. Please consult your Bentley representative for product specifics relative to your needs.

SACS Executive: Common interface to program suite

- Controls and connects all elements of the SACS system
- Launches all SACS interactive programs
- Executes all batch program analyses
- Allows access to all SACS system configuration settings, including system file location and security key settings
- Includes command line help and power buttons for the commonly executed tasks
- Specifies analysis options without changing data input file
- Generates SQLite database for reporting in Excel and other programs
- Integrated with ProjectWise® for sharing of all files acrossmultiple sites

Precede: Interactive full screen 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 modeling, including plate and shell elements
- Automatic offshore jacket and deck generation
- User-defined input units
- Cartesian, cylindrical, or spherical mesh generation
- Automatic weight or load generation, including gravity, pressure, and skid-mounted equipment loads
- Seastate data generation capabilities
- Decommissioning capabilities to automatically slice structure by volume into smaller components of given weight and center of gravity
- Interactive GUI to mesh stiffened joints for automatic stress concentration factor (SCF) extraction and strength analysis
- Extensive plotting and reporting capabilities
- Code check parameter generation, including K-factors and compression flange unbraced lengths
- Allows SACS model files to be converted into 3D SAT file format compatible with AutoCAD, and other CAD systems
- Allows the definition of sets comprised of joints, members, plates, shells, and solid elements
- Allows views to be saved as models
- Physical member support capabilities
- Professional and other ACIS-enabled CAD packages
- Supports full 3D geometry and section properties
- Allows SACS model files to be ported directly to a PDMS macro file, which creates the 3D model in PDMS
- Supports PDMS section libraries in addition to creating PDMS sections defined in the SACS model
- Logging functionality
-

- ISM Export to ProSteel and other steel detailing systems
- Integrated with ProjectWise for sharing all projects files across multiple sites

Data Generator: Interactive data generation for all programs

- Intelligent full-screen editor that labels and highlights data fields and provides help for data input
- Form-filling data input available as well as full-screen mode
- Automatic data checking
- Integrated with ProjectWise for sharing of all project files across multiple sites

Chartvue: Interactive chart and plot viewer

- Ability to view plot files on screen
- Sends viewed plots to printer/plotter
- Supports HP-GL, Postscript, DXF, Windows devices
- Metafile (WMF), and SACS NPF plot file
- Allows plot size, character size, margins, and formats
- Ability to modify chart settings

Seastate: Environmental loads generator

- Full implementation of API 21st edition
- Supports five wave theories
- Current included or excluded
- Generates load due to wind, gravity, buoyancy, and mud flow
- Marine growth, flooded, and non-flooded members
- RAO and acceleration loading, including nonstructural weights
- Moving loads generation
- Diameter, Reynolds number, and wake encounter effects dependent drag and inertia coefficients
- Weight load cases
- Forces on nonstructural bodies
- Deterministic and random wave modeling for dynamic response
- Member hydrodynamic modeling for static and dynamic analysis
- Corrosion modeling

SACS IV Solver: Static beam and finite element analysis

- Advanced linear static solver methods, including Krylov subspace solvers for statically indeterminate structures
- Beam elements, including tubulars, tees, wide flanges, channels, angles, cones, plate and box girders, stiffened cylinders, and boxes
- Solid and plate elements (isotropic and stiffened)
- Discrete Kirchoff Theory (DKT) thin plates
- Isoparametric 6-, 8-, and 9-node shell elements
- Library of AISC, United Kingdom, European, German, Chinese and Japanese cross sections, as well as user-defined libraries
- 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
- Unlimited load cases
- P-delta effects
- Retained/Constrained DOF

Post: Beam and finite element code check and redesign

- Beam and plate element code check and redesign
- API (including 22nd edition), ISO 19902, AISC, LRFD, Norsok, Eurocode 3, Canadian, DNV, British Standards, and Danish
- DS449 code check
- Plate panel checks in accordance with DNV-RP-C201 and DNV-RP-C202
- Creates updated model with redesigned elements
- Modify code check parameters
- Load combination capabilities
- Supports codes from 1977 to present
- Detailed and summary reports
- Hydrostatic collapse analysis
- Span (multimember effects)
- Automatic extrication of SCFs from a finite element analysisof a meshed joint

Joint can: Tubular joint code check and redesign

- Present and past codes including latest API 22nd edition,
- API LRFD, ISO 19902, Norsok, DS449, and Canadian
- API earthquake and simplified fatigue analysis
- Connection strength (50%) check
- Overlapping joints analyzed
- Minimum and extreme seismic analysis

Postvue: Interactive graphics post processor

- Interactive member and tubular joint code check and redesign,
- with the option to print code check details for latest AISC,
- ASD and LRFD, API, ISO 19902 codes
- Display shear and bending moment diagrams
- Display rendered deflected shapes for static and dynamic analyses
- Animate deflected and modal shapes
- Color plate stress contour plots
- Code check and 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 re-analysis
- Labels UC ratio, stresses, and internal forces on elements

Concrete: Reinforced concrete code check and redesign

- Rectangular, Circular, Tee, and L cross sections
- Beam, bi-axial beam-column, slab, and wall elements supported
- Multiple reinforcement patterns can be specified
- Code check per ACI 318-89 (revised 1992)
- Shear reinforcement check and redesign
- Reinforcement development length check
- Deflection and creep calculation
- Second order/P-delta analysis capabilities

Fatigue: Fatigue life evaluation and redesign

- Spectral, time history, and deterministic fatigue analysis
- Cyclic stress range calculation procedures include wave search, curve fit, and interpolation
- SCF calculations recommended by API (including 21st edition supplements), HSE, DNV, DS449, and Norsok Codes
- Automatic redesign
- API (including 21st edition supplements), ISO 19902, AWS, HSE, and Norsok thickness dependent recommended S-N curves
- Multiple run damage accumulation
- Pierson-Moskowitz, JONSWAP, Ochi-Hubble double peak, simplified double peak, and user-defined spectra
- Automated or user-specified connection details
- Pile fatigue analysis
- Creates wave spectra from scatter diagram
- Uses Paris equation to predict crack growth rate due to cyclic stresses
- Load path dependent joint classifications
- Includes wave spreading effects
- Reservoir (rain flow) cycle counting method

Interactive fatigue: Interactive fatigue life evaluation

- Shows the 3D view of the connection and allows braces to be selected with the mouse
- Reads connection defaults when joint and/ or brace is/are selected, thus eliminating the need to calculate and display
- SCFs before viewing capacity or modifying properties
- Recognizes all SCF and S-N options available in the batch program
- Allows SCF theory to be changed for any type of connection, including inline connections and connections with user-defined SCFs
- Reports have been expanded and reworked to make them easier to read
- Reports and plots can be displayed on the screen and/or saved to a file
- Automatic redesign
- GAP
- Nonlinear analysis with one-way elements
- Accurate simulation of load out or transportation analysis using one-way elements
- Tension or compression gap elements with initial gap
- General nonlinear elements
- Friction elements
- PSI
- Nonlinear Soil, Pile, and Structure Interaction
- Beam column effects included
- Nonuniform piles
- P-Y and T-Z curves, axial adhesion, and springs
- API P-Y, T-Z, skin friction, and adhesion data generated from soil properties per API
- Full structural analysis and pile code check API, LRFD, Norsok, HSE, DS449, Canadian, and DNV
- Offset P-Y & T-Z curves for mudslides
- Full plotting and graphical representation of soil data and results, including stresses, P-Y, T-Z curves
- Soil liquefaction effects

Pile: Isolated 3D pile analysis

- Beam column and pile batter effects included
- Uses PSI soil data
- Optional pile head springs
- Specify force at or below pile head
- Specify pile head displacements
- Specified pile head forces or displacements
- Automatic generation of linear equivalent pile stubs for dynamic or static analysis
- Soil liquefaction effects
- Same plotting and code check features as PSI

Superelement: Automated substructure creation and application

- Unlimited number of superelements
- Up to 1,000 interface joints per superelement
- Translation and rotation of superelements
- User-defined stiffness matrices
- Full stress recovery
- Superelements can contain other superelements
- Translation and rotation of superelements

Dynamic superelement

- Can be based on either Guyan or Craig-Bampton methods
- Uses Modal Assurance Criterion to check accuracy
- User-defined output format

Combine: Common solution file utility

- Combines dynamic and static results from one or multiple solution files
- Combines results from analyses having different member, plates, etc.
- Superimposes mode shapes
- Worst-case combination of dead loads with earthquake response
- Determine extreme wave loads from input spectra

Large deflection (LDF): Large deflection analysis

- Iterative solution for geometric
- Solves plate membrane problems
- Accounts for P-delta effects nonlinearities

Collapse: Nonlinear collapse analysis

- Linear and nonlinear material behavior
- Nonlinear springs
- Sequential load stacking capability
- Activate and deactivate elements
- Joint flexibility options
- Impact analysis with automatic unloading, built-in DNV ship indentation curves and energy absorption functionality
- Load cases may contain loading and/or specified displacements
- Includes geometric nonlinearities
- Plastic members and finite elements
- Includes piles with nonlinear soils and plasticity
- Soil liquefaction effects
- Plastic DKT plates

System requirements

Processor CPU: Pentium 4 or higher

Operating system: Windows 8/8.1, 10, 11

Memory - RAM: Minimum 2 GB–SACS performance is dependent on model size and resources available

Hard disk: Minimum 10 GB partition for SACS installation recommended

Display: Graphics card with a chipset that supports Open GL

Network: A network connection is required. 100 Base-T or greater local area ethernet network TCP/IP network protocol is supported.

For the most up-to-date system requirements, visit Bentley Communities.

Dynpac: Dynamic characteristics

- Householder-Givens solution
- Guyan reduction of non-essential degrees of freedom
- Lumped or consistent structural mass generation
- Automatic virtual mass generation
- Complete seastate hydrodynamic modeling
- User input distributed and concentrated mass
- Nonstructural weight modeling
- Full 6 DOF modes available for forced response analysis

Wave response: Dynamic wave response

- Deterministic and random waves
- Pierson-Moskowitz, Jonswap, Ochi-Hubble, and user wave spectra
- Harris, Von Karmon, and Kaimal wind spectra
- Fluid-structure relative velocity and acceleration accounted for "Modal Acceleration" and nonlinear fluid damping
- Closed form steady state response in the frequency domain
- Equivalent static load output for accurate stress recovery
- Zero crossing and RMS responses
- Time history analysis of wave and wind and time history load
- Buoyancy dynamic loads included
- 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
- Special features for wind turbine analysis
- Nonlinear foundation response for time history analyses

Dynamic response: General dynamic response and earthquake analysis

- Frequency domain analysis
- Time history, response spectrum, or PSD-base driven input for seismic analysis
- Force time history and harmonic-force driven analysis
- SRSS, CQC, and peak modal combinations
- API response spectra library and user input spectra
- Wind spectral loading capability
- Structural and fluid damping
- Vibration analysis with multiple input points with user specified frequencies and phasing
- General periodic forces decomposed by Fourier analysis
- Ice dynamics analysis
- Engine/compressor vibration analysis
- Response spectrum output at any joint
- Equivalent static load and incremental load output resulting from earthquake, ship impact, dropped object and blast analysis. This loading can be used for subsequent linear static analysis or for nonlinear collapse analysis
- Ship impact analysis
- Dropped object analysis

Tow: Transportation inertia load generator

- Input motion for six degrees of freedom
- Output location for selected points
- Automatic weight calculation
- User input member and joint weights
- Generates distributed member and plate loads
- Converts user defined loads into inertias

MTO: Material takeoff, weight control, and cost estimation

- Member lengths including cuts
- Steel tonnage and CG location
- Material list, cost estimate, and weight control reports
- Weld volume requirements and cost
- Required protective anodes and cost
- Surface area calculations by elevation
- Anode calculation in accordance with NACE SPO176-2007 (formerly RP0176-2003) and DNV-RP-B401