

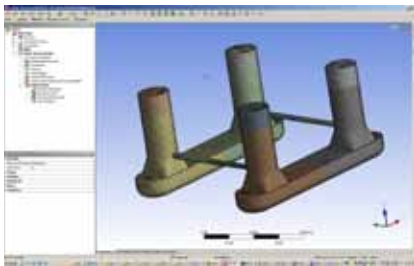
ANSYS ASAS

Proven Technology for the Design and Analysis of Offshore Structures

Offshore structure projects today require that the design and analysis are performed efficiently in terms of time and money. Engineering simulation software must be powerful, fast and easy-to-use. ANSYS® ASAS™ and ANSYS ASAS Offshore are comprehensive engineering analysis suites for performing structural assessments of most types of offshore structures, including jackets, jackups, concrete gravity-based platforms, risers, floating production storage and offloading (FPSO) systems, tension leg platforms (TLPs), semi-submersibles and ships. The breadth of capabilities, large modeling capacity, and analysis and post-processing flexibility allow all stages of the design process to be undertaken with the chosen single suite of software, greatly increasing productivity.

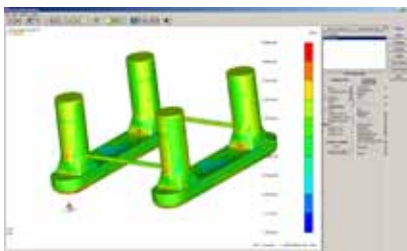
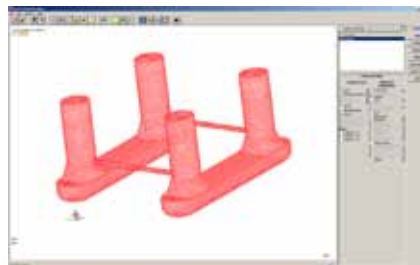
ANSYS ASAS

The ANSYS ASAS package is a general-purpose analysis tool with linear and nonlinear options including code checks. Model creation can be performed using ANSYS® Structural Mechanics products and ANSYS® DesignModeler™ software solutions or with FEMGV®.



Semi-submersible modeled in ANSYS Mechanical software

Transferred to ANSYS ASAS software and viewed in ANSYS ASAS Visualizer



Results visualization

Product Features

Modeling

- Models can be created from ANSYS software products or FEMGV
- Extensive data input flexibility using parameters
- All data files easily editable
- Section library for beams
- Freedom releases to permit pin and sliding connections
- Stiffened plate structure modeling
- Cartesian, cylindrical and spherical coordinate systems
- Family of force-equilibrium membranes and shear panels to represent areas where high-shear flows are evident
- 2- and 3-node bars and beams including tubes and box elements with local and global offsets
- Full range of solid elements including high-accuracy 8-node and 20-node bricks, 6- and 15-node wedges, 10-node tetrahedral and 15- and 20-node laminated elements
- Low- and high-order shell elements with thick- and thin-shell formulations
- Special crack elements and a shell-brick interface
- Most elements can be used for field problems (heat transfer, etc.)
- Isotropic, orthotropic and anisotropic material
- Units can be modified within the data
- Interfaces to ANSYS® AQWA™ software for transferring pressure loads and accelerations for hull design
- Stiffened plate definitions
- Utilization of up to four simultaneous processors for higher productivity
- Multilevel substructure analysis for modeling efficiency
- Load case combinations and factoring
- Set and group definition for efficient analysis management

Product Features

- Joint flexibility for tubular joints
- Dynamic analysis in both frequency and time domain
- Composite mode damping in frequency domain
- Hysteretic damping in time domain
- Local and global buckling
- Large displacement
- Plasticity with various laws including partial plasticity
- Creep with various laws including user defined
- Fracture including line-spring formulation
- Multiple load cases with full range of load types including nodal, pressure, distributed, face and element temperatures, body forces, centrifugal loads and angular acceleration
- Seismic analysis
- Transient nonlinear heat conduction analysis capabilities including convection and radiation; time dependent thermal loads automatically transferred to stress analysis
- Tube-in-tube analysis including contact
- Composite material analysis for shells and bricks
- Laminated, woven roving and chopped strand material definitions
- Composite elastic failure covered by Azzi-Wu, Norris, Tsai-Wu, maximum stress or maximum strain
- Composite inelastic failures covered by combination of Tsai-Wu and maximum stress laminate and lamina definitions

Code Checks

- Member and joint code checks performed to most commonly used codes for offshore structures
- Reports for data input, members and unity checks
- Intermediate forces and stresses along a member
- Fiber stresses produced for standard or user-defined sections
- New load cases by combining and factoring existing cases
- Local buckling and column buckling checks

The ANSYS ASAS package is the basic building block for performing linear and nonlinear finite element analysis. Unlike many other tools, ANSYS ASAS software has powerful capabilities to address both continuous and framed structures including code checks. The product contains three main solvers plus a number of special modules to provide tremendous flexibility and power.

ANSYS ASAS Solvers

- ASAS(L) is primarily a linear solver (although it includes gap elements) with static and dynamic capabilities. Solid, shell and plate elements are available and are coupled with a range of beam elements that may be assigned predefined or user-supplied cross sections. Key solver features offer high capacity (even in 32-bit PC environments); modal response including composite mode damping, multipoint constraints and rigid elements; load case combination and factoring even from separate simulations and multilevel substructuring with automatic superelement connectivity.
- ASAS(NL) extends ANSYS ASAS software capabilities to provide nonlinear analysis such as plasticity, dynamics, buckling and large deflections. Also included is fully coupled wave structure interaction with both regular and irregular (random) waves, and tools for coupling with third-party wind turbine blade dynamics software. Response amplitude operators (RAOs) may be imported from ANSYS AQWA software as a time history forcing function for simulating floating-body effects.
- BEAMST (also available separately as ANSYS BEAMCHECK product) performs joint and member utilization checks to various codes of practice for framed structures including AISC 10th edition working stress and 2nd edition LRFD; API RP2A-WSD 21st edition working stress (updated in 2007) and RP2A-LRFD 1st edition; ISO 19902 2007 (including dented member checks); BS5950 part 1 1992; NORSOK 2000; NORSOK, NS3472 1984; NPD 1992; DS449 1984 (with 1994 amendments); DS412 1984 (with 1994 amendments). The code checking module may also be used to process results from one or more ANSYS structural mechanics simulations, independently or via the Design Assessment analysis system.



Nonlinear dynamic analysis of offshore wind turbine structure
Courtesy REpower Systems AG.

ANSYS ASAS Offshore

The ANSYS ASAS Offshore package provides extended capabilities to the ANSYS ASAS tool for the assessment of framed structures such as jackets, jackups, subsea templates, risers and topsides.

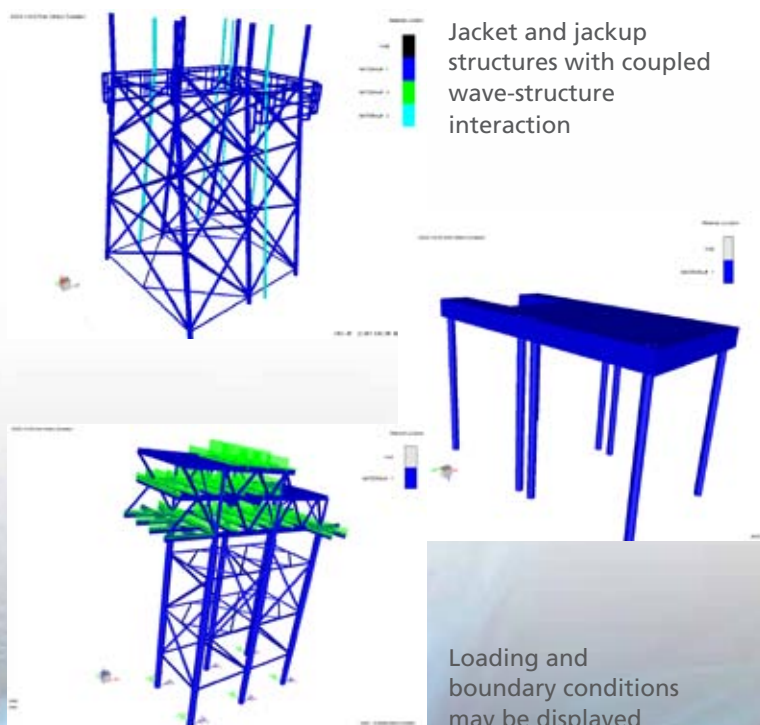
ANSYS ASAS Offshore Solvers

ANSYS ASAS Offshore software includes all of the solvers from ANSYS ASAS, i.e., ASAS(L), ASAS(NL) and BEAMST, plus the following:

- ASAS-WAVE/MASS calculates the wave loading and added mass on tubular framed structures such as jackets; a range of wave types are available and applicable to shallow- and deep-water conditions
- SPLINTER performs soil-pile-structure interaction and soil-pile interaction
- FATJACK (also available separately as ANSYS FATJACK product) performs deterministic, spectral and time-history (rainflow counting) fatigue analyses of jackets and topside structures constructed from framed members

Importantly, like ANSYS BEAMCHECK, ANSYS FATJACK can also be used independently with relevant ANSYS structural mechanics products or via Design Assessment analysis system.

ANSYS ASAS Offshore software has been developed over a period of more than 30 years and has been honed through continuous applications in the demanding North Sea environment where many of the jacket structures have been designed or analyzed with the aid of ANSYS ASAS software.



Product Features

- Different joint types can be specified
- Response spectrum analysis results can be processed
- Searching facilities to obtain maximum stresses and forces
- Can perform code checks for members with elastic deformations within non-linear analysis
- Interfaces to one or more ANSYS Mechanical simulations

Wave Loading

- Wave theories include linear wave (Airy), solitary wave (Cnoidal 1st), Stokes 5th order, stream function and Shell new wave (dispersive wave)
- Input of wave kinematics in the form of a grid of velocities and accelerations
- Forces on submerged sections calculated using Morison's equation
- Drag and mass coefficients can be user defined or calculated from a table of Reynolds and/or Keulegan-Carpenter numbers
- Linear, exponential or user-defined current variation
- Marine growth with optional drag coefficients and density
- Self weight and buoyancy calculated for some or all members
- Full hydrostatic loading effects on members calculated including induced end loads

Soil-Pile-Structure Interaction

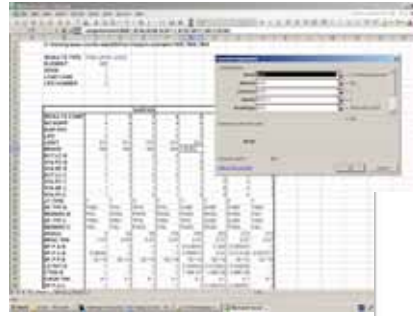
- Single pile or pile group
- Variable pile cross sections
- Automatic subdivision of piles based on pile cross section and number of soil layers
- Automatic calculation of Mindlin coefficients for pile groups
- Automatic generation of P-Y and T-Z curves
- Efficient nonlinear solution for soil-pile-structure interactions via superelement approach
- Stress recovery to structural elements following convergence

Product Features

- Tangent and secant stiffness methods for iterating with nonlinear P-Y and T-Z curves for soil
- Skewed system option for constraints or loading
- Non-tubular cross sections and tubes within tubes
- Linearized pile cap stiffness option
- Code checking to API RP2A

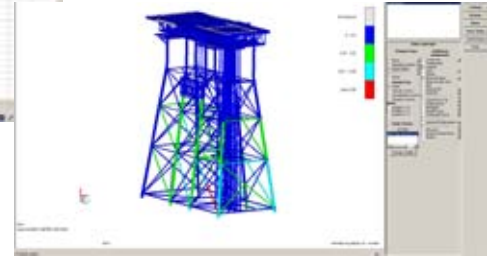
Fatigue

- Deterministic, spectral or rainflow counting methods
- Tubular framed structures subject to waves and current or wind including wind gusts
- Scott-Weigel, Jonswap, Pierson-Moskowitz or user-defined wave spectra
- Empirically derived stress concentration factors including Efthymiou influence functions
- User-defined stress concentration factors for both box and tubular members
- Wave spreading effects



Code check results displayed in the ANSYS ASAS Visualizer

Excel functional interface



Key Characteristics of ANSYS ASAS Packages

Productivity tools available for ANSYS ASAS packages include:

- Direct interfaces for Excel® and Mathcad® for data and results extraction
- A toolkit providing a programmatic interface to data and results
- ANSYS ASAS Visualizer for interactive results presentation and manipulation

ANSYS ASAS is a software suite that is part of the comprehensive range of ANSYS applications for collectively satisfying demanding engineering and design requirements of the offshore industry. The ANSYS ASAS suite is supported by ANSYS Mechanical software for structural mechanics analysis, and ANSYS® CFX® and ANSYS® FLUENT® products for fluid flow analysis. In addition, the ANSYS AQWA solution enables radiation-diffraction, mooring configuration and articulation simulations of multibody floating structures. ANSYS offers the most complete range of solutions for the offshore industry, from front-end engineering and design (FEED) studies through global analysis and component design. Solutions are available for contractors, consultants, fabricators, certification authorities, operators and equipment manufacturers.

The ANSYS Advantage

With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading edge design concepts into innovative products and processes that work. Today, almost all of the top 100 industrial companies on the "FORTUNE Global 500" invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world's most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of solutions from ANSYS delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world's most successful companies turn to ANSYS — with a track record of 40 years as the industry leader — for the best in engineering simulation.

ANSYS®
www.ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, CFX, FLUENT, and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

