Exclusive Distributor in the Middle East for CSI Software licensing, technical support and training solutions – www.techiesoft.com

For Sales: sales@techiesoft.com For Technical Support: support@techiesoft.com
For Licensing: license@techiesoft.com

SAP2000

Integrated Software For Structural Analysis And Design

Version: 16.1.0

HOME

SAP2000 Overview
The SAP name has been synonymous with state-of-the-art analytical methods since its introduction over 30 years ago. SAP2000 follows in the same tradition featuring a very sophisticated, intuitive and versatile user interface powered by an unmatched analysis engine and design tools for engineers working on transportation, industrial, public works, sports, and other facilities.

From its 3D object based graphical modeling environment to the wide variety of analysis and design options completely integrated across one powerful user interface, SAP2000 has proven to be the most integrated, productive and practical general purpose structural program on the market today.

This intuitive interface allows you to create structural models rapidly and intuitively without long learning curve delays. Now you can harness the power of SAP2000 for all of your analysis and design tasks, including small day-to-day problems. Complex Models can be generated and meshed with powerful Templates built into the interface.

The Advanced Analytical Techniques allow for Step-by-Step Large Deformation Analysis, Multiple P-Delta, Eigen and Ritz Analyses, Cable Analysis, Tension or Compression Only Analysis, Buckling Analysis, Blast Analysis, Fast Nonlinear Analysis for Dampers, Base Isolators and Support Plasticity, Energy Methods for Drift Control and Segmental Construction Analysis.

Bridge Designers can use SAP2000 Bridge Templates for generating Bridge Models, Automated Bridge Live Load Analysis and Design, Bridge Base Isolation, Bridge Construction Sequence Analysis, Large Deformation Cable Supported Bridge Analysis and Pushover Analysis.

SAP2000 is for everyone! SAP2000 is for every project! From a simple small 2D static frame analysis to a large complex 3D nonlinear dynamic analysis, SAP2000 is the answer to all structural analysis and design needs.

**SAP2000 FEATURES:**

**Features-SAP2000 User Interface**
Single Unified User Interface

SAP2000 offers a single unified user interface for modeling, analysis and design.

- Autocad-like snap tools for fast and easy model creation
- Zoom controls
- Standard views XY, XZ, YZ, 3D
- Live multiple graphical windows that make for easy navigation of complex models
- Active planar view is automatically highlighted in the 3D view window
- Developed elevation option to define custom views, including "unwrapping" cylindrical and irregular geometry
- Define working planes for creating and editing complex geometries
- Coordinate Systems/Grids
  - Cartesian
  - Cylindrical
  - Generalized Grid Systems - allows for a system comprised of arbitrarily defined grid lines.
  - Models can have multiple coordinate systems/grids which can be rotated in any direction.

Features-SAP2000 Modeling

Section Designer
• Specialized sections
• Allows users to create any arbitrary shape and any user defined material
• Automatically calculates all section properties
• Generates biaxial interaction diagram for concrete sections
• Moment curvature diagrams

Interactive Database Editing

• All SAP2000 data can be viewed and edited using spreadsheets
• Edit within SAP2000
• Bi-directional direct link to MS Excel for editing
• Allows users to define a portion of the model, or even the entire model using spreadsheets®
Frame Elements

- The Frame Element in SAP2000 can be either a straight or curved element.
- Intermediate joints will automatically be generated where other members intersect with frame to ensure finite element connectivity.
Tendon Elements

- Tendons are easily drawn as independent objects, with geometry specified as straight lines, parabolas, circular curves, or other arbitrary shapes.
- SAP2000 automatically connects the tendons to the frame, shell, or solid objects that contain them.
- External tendons can also be modeled.
- Tendon loads, including all losses, are easily specified. For simple analyses, tendons can be simply used to create loads that act on the structure.
- Tendons can instead be modeled as elements that interact with the rest of the structure. Using this approach, staged-construction analysis can consider the sequence with which tendons are added, tensioned, and affected by time-dependent creep and shrinkage.
Cable Elements

- Geometry definition options
  - Minimum tension at I and/or J end
  - Tension at I and/or J end
  - Horizontal tension component
  - Maximum vertical sag
  - Low-point vertical sag
  - Undeformed length
  - Relative undeformed length
  - Deformed cable geometry

- Define as:
  - Single Cable
  - Multiple cable segments

- Catenary cable behavior
- Large-displacement cable element can model tension-stiffening and the effects of large rotations. The cable shape automatically adapts to the applied loading, and will buckle out of the way under applied compression.
- Applications include suspension and cable-stayed bridges, guyed towers, pipeline risers, and more. When used in conjunction with staged construction, realistic stresses and stiffnesses of the structure can be found.
Shell Elements

- Shell elements (plate, membrane, full-shell) used to model walls, floors, tank/vessel shells and other thin-walled areas, as well as two-dimensional solids (plane-stress, plane-strain, and axisymmetric solids).
- Layered shell element considers mixed material composite behavior, nonlinear material behavior options for each layer based on stress-strain, with shearing behavior considered for rebar layered shell sections.
- Incompatible modes automatically included in the area element stiffness formulation to improve in-plane bending behavior.
- Thick shell/plate plate option when shear deformations become significant. The thick plate formulation captures both shear and bending deformations, whereas the thin plate formulation is based only on bending deformations and neglects shear deformations.
Solid Elements

- Eight-node solid element based on isoparametric formulation with incompatible modes.
- Solid elements can support degenerate solids where nodes are collapsed (duplicated) to make wedges and tetrahedra.
- Useful for modeling three-dimensional objects in which loading, boundary conditions, section properties or reactions vary by thickness.
- Area elements can be extruded into solids
Meshing Tools

- Automatic mesh generation
  - Many different meshing control options
  - Will always create quadrilateral sub elements
- User has full control of how mesh gets generated
- Reshaper tool can be used to reshape and control mesh geometry
- Mesh by gridlines, mesh by selected lines, by intersecting objects or by selected joints.
- Area surface loads can be distributed as one-way or two-way.
- User controls which elements receive load distribution and which do not. For example, a user may not wish to distribute area loads from a meshed floor to selected bracing modeled on the same plane
- Object based model automatically converts into an element based model for internal analysis. Frames, areas, and/or solids can have refined meshes for analysis, but then the model is reformulated to report results as if these elements were singular unmeshed objects.
- Meshed area and solid finite elements can automatically add joints to adjacent frame elements for internal analysis.
Automatic Edge Constraint

- Automatic Edge Constraint technology for mismatched meshes
- Analytically connects all mismatched meshes using joint interpolation algorithms
Link Elements

- SAP2000 has a many different link elements available for users to accurately represent the behavior of a structure.
  - Linear
  - Multi-linear Elastic
  - Multi-linear Plastic
  - Gaps
  - Hooks
  - Dampers
  - Friction Isolators
  - Rubber Isolators
  - T/C Isolators
  - Frequency-dependent Springs
  - Frequency-dependent Dampers
Hinge Properties

- Users can create and apply hinge properties to perform pushover analyses in SAP2000.

- **Nonlinear Fiber Hinges**
  - Nonlinear material behavior in frame elements (beam/column/brace) can be modeled using fiber hinges. This approach represents the material in the cross section as discrete points, each following the exact stress-strain curves of the material. Mixed materials, like reinforced concrete, and complex shapes can be represented.
  - Yielding, cracking, and hysteresis are all captured.
  - Using multiple hinges along the length of an element represents the full 3-D nonlinearity in a member, although for most practical cases this is not needed.

- Fiber hinges are utilized in nonlinear static and dynamic analysis.
Features-SAP2000 Loading

Automatic Code Based Wind Loading

- SAP2000 will automatically generate wind loads based on various domestic and international codes including but not limited to:
  - UBC 94; 97
  - BOCA 96
  - ASCE 7-95; -02; -05
  - NBCC 2005;2010
  - Mexican
  - Chinese 2002
  - IS875 1987
  - User defined

- Open Structure Wind Loading
  - Users can use the code-based wind loading features of SAP2000 to apply wind loads to open structures. SAP2000 will use the web flanges as the sail areas when applying loads on open structures.
Automatic Code Based Seismic Loading

- SAP2000 will automatically generate seismic loads based on various domestic and international codes including but not limited to:
  - UBC 94; 97
  - BOCA 96
  - NBCC 95; 2005; 2010
  - IBC 2003; 2006
  - Chinese 2002
  - ISI1893 2002
  - NEHRP 97
  - User coefficient
  - User loads
Moving Load Generation

- SAP2000 has a sophisticated moving load generator that allows users to apply moving loads to lanes on frame elements and/or lanes on shell elements.
- The moving load generator will move a vehicle within the lane to determine maximum envelope conditions. Included is a library of various AASHTO vehicles, International vehicle definitions, Rail Loads, and General Vehicles that are fully customizable.
Wave Load Generation

- The wave-loading feature automatically generates loading on the structure resulting from waves, current flow, buoyancy and wind.
- Multi-stepped static linear loading can be generated to simulate the wave moving through the structure.
- In addition, dynamic (time-history) loading can also be generated to include inertial effects as the wave moves through the structure. The dynamic loading is idealized as a sequential series of overlapping triangular pulse loads applied to the structure.
**Features-SAP2000 Analysis**

**General Analysis Details**

- Solvers that have been tried and tested by the industry for over 35 years.
- Advanced SAPFire Analysis Engine
- Multiple 64-Bit Solvers for analysis optimization
- Eigen Analysis
- Autoshifting for ill-conditioned problems
- Ritz Analysis
Response Spectrum Analysis

- Response-spectrum analysis is a statistical type of analysis for the determination of the likely response of a structure to seismic loading.

- Response-spectrum analysis seeks the likely maximum response to these equations rather than the full time history.

- The earthquake ground acceleration in each direction is given as a digitized response-spectrum curve of pseudo-spectral acceleration response versus period of the structure.
Power Spectral Density

- Power-spectral-density analysis is available to determine the probabilistic response of a structure due to cyclic (harmonic, sinusoidal) loading over a range of frequencies. This is useful for fatigue studies, random response due to earthquakes, and other applications.
- Multiple loads may be applied at different phase angles, and may be correlated or uncorrelated.
- The structure may be damped or undamped.
- Frequency-dependent stiffness and damping (complex impedance) properties may be included for modeling foundations and far-field effects, including radiation damping.
- Power-spectral-density curves may be plotted for any response quantity, and the integrated expected value is automatically computed.
Steady State Analysis with Damping

- Steady-state analysis is available to determine the response of the structure due to cyclic (harmonic, sinusoidal) loading over a range of frequencies.
- Multiple loads may be applied at different phase angles.
- The structure may be damped or undamped.
- Frequency-dependent stiffness and damping (complex impedance) properties may be included for modeling foundations and far-field effects, including radiation damping.
- The response may be viewed at any phase angle. The effects of multiple machines operating at different frequencies can be considered by combining the results of several analyses in the same model.
Buckling Analysis

- Linear buckling (bifurcation) modes of a structure can be found under any set of loads.
- Multiple buckling modes can be found, each giving the mode shape and the buckling factor of safety.
- Multiple sets of loads can be considered. Buckling modes can be found for the structure at the end of any staged construction case or any nonlinear static or dynamic analysis.
- Nonlinear buckling analysis is also available considering P-delta or large-deflections effects. Snap-through buckling behavior can be captured using static analysis with displacement control.
- Dynamic analysis can also be used for modeling buckling, including follower-load problems.
- Linear and nonlinear buckling analysis can be combined for the greatest flexibility in understanding structural instabilities.
Time History Analysis

- Two methods of Time History Analysis:
  - Modal Time History
    - Uses the method of mode superposition
  - Direct Integration Time History
    - Solves equations for the entire structure at each time step
  - Nonlinear

- Time History Functions
  - Sine
  - Cosine
  - Ramp
  - Sawtooth
  - Triangular
  - User defined

- Nonlinear direct-integration time-history analysis cases can be chained together with other nonlinear time-history or static cases (including staged construction), to address a wide range of applications.
Tension and Compression only Springs

- Frame elements may be assigned compression limits for modeling braces and stay cables, or tension limits for modeling masonry or special physical devices.

- In the example, the base plate is modeled with both tension and compression springs.
  - An elastic analysis allows springs to take both tension and compression.
  - A nonlinear analysis allows springs to take tension only or compression only.
P-Delta Analysis (large and small)

- The P-Delta effect refers specifically to the nonlinear geometric effect of a large tensile or compressive direct stress upon transverse bending and shear behavior. A compressive stress tends to make a structural member more flexible in transverse bending and shear, whereas a tensile stress tends to stiffen the member against transverse deformation.
Pushover Analysis

- FEMA 273/ATC-40 hinge and fiber hinge option based on stress-strain
- Nonlinear layered shell element enables users to consider plastic behavior of concrete shear walls, slabs, steel plates and other area finite elements in the pushover analysis.
- Force-Deformation relations for steel and concrete hinges
- Modal, uniform, or user defined lateral load patterns
- Capacity spectrum conversions
- Effective damping calculation
- Demand spectrum comparisons
- Performance point calculation
- Summary reports including plastic hinge deformations
Staged Construction

- Incremental Construction Sequence Modeling and Loadings
- Structures can be analyzed for the effects of staged construction, including changing support conditions, section properties, and the adding and removing of temporary shoring.
- Nonlinear effects can be considered such as large deflections, yielding, and gap opening and closing.
- Time-dependent creep, shrinkage, and strength-change effects can be included.
- Arbitrary loading sequences may be applied.
- Tendons can be jacked in multiple stages, and cables automatically tightened to target tensions. Multiple construction sequences may be analyzed in the same model and compared or enveloped.
- Dynamic, buckling, and other types of analyses can be performed at the end of any construction sequence, so that the behavior of a structure can be examined before and after a retrofit.
- A Gantt-chart scheduler is available for easy definition of staged-construction load cases.
Concrete Shrinkage and Time Dependent Creep Analysis

- Long term deflections due to creep and shrinkage can be computed along with staged sequential construction analysis.
- Time dependent material properties based upon the 1990 edition CEB-FIP code and user defined curves are used to compute creep strains.
Target Force Analysis

- During nonlinear static analysis, cable and frame elements can be automatically strained to achieve specified target axial force values. This is most commonly used to tighten cables to pre-specified tensions, but it can also be used to jack structures to a specified force using frame elements, as well as other applications.
- Multiple, simultaneous targets can be considered in a single load case.
- During staged construction, different target forces can be specified in different stages.

Model Alive

- For small to medium sized structures, analysis can be performed on-the fly as you build and modify the model.
- For each change you make to the geometry, properties, or loading, the structure instantly responds with the new deformed shape, moment diagram, or any other plot of results. It’s like working with a live model, and it is a very powerful tool for conceptual design and for testing "what-if" scenarios.

Features-SAP2000 Design

Steel Frame Design

- Automatic member sizing – No preliminary design required
- Virtual work based optimization for lateral deflections
- Grouping of members for member sizing
- Australian AS 4100-1998
- British (BS 5950-2000, BS 5950-1990)
- Canadian (S16-09, S16-01, CISC 95)
- Eurocode 3-2005, Eurocode 3-1993
- Italian UNI 10011
- New Zealand (NZS 3404-1997)
- Norway (Norsok N-004)
- Design for static and dynamic loads
- Code dependent or user defined loading combinations
- Automatic calculation of K-factors and P-delta effects
Integrated section designer for composite & built-up sections
Interactive options for design and review
Design for Effects of Torsion

Concrete Frame Design

- British (BS 8110-1997, BS 811-1989)
- Canadian (CSA A23.3-04, CSA A23.3-94)
- Eurocode 2-2004, Eurocode 2-1992
- Hong Kong CP 2004 (R2007)
- Indian IS 456:2000
- Italian DM 14-2-92
- Korean KCI-1999
- Mexican RCDF 2001
- New Zealand 3101-1995
- Singapore CP 65:99
- Design for static and dynamic loads
• Grouping for design envelopes
• Automatic or user defined loading combinations and design groups
• Automatic calculations of live load reduction factors
• Design for biaxial-moment/axial-load interaction & shear
• Automatic calculation of moment magnification factors
• Magnification override option with the evaluation of P-delta effects
• Integrated section designer for complex concrete sections
• Interactive options for design and review
• Design for effects of torsion
• Virtual work based optimization for lateral deflection control
Aluminum Frame Design

- Design for static and dynamic loads
- Straight and curved girder design
- Automatic calculation of moment magnification factors
- Automatic calculation of K-Factors and P-Delta effects
- Automatic calculation of axial and biaxial moment and shear design

- Supported aluminum frame design codes:
  - AA-ASD 2000
  - AA-LRFD 2000

Cold Formed Frame Design

- Design for static and dynamic loads
- Straight & curved girder design
- Automatic calculation of moment magnification factors
- Automatic calculation of K-Factors and P-delta effects
- Effective area and effective major/minor moment of inertia calculation

- Supported cold form frame design codes:
Features-SAP2000 Output and Display

Deformed Geometry

- Users can display deformed geometry based on any load, or combination of loads.
- Option to animate deflections
- Various mode shapes of the modal analysis can be shown and animated.
Moment, Sheer and Axial Force Diagrams

- Shear and moment diagrams display internal shear forces, moments, and displacements at all locations along the length of a frame element for any load case or load combination.
- Option to scroll along the length to display values or go to max location
Generalized Displacement Output

- Arbitrary displacement measures can be defined using generalized displacements. These are linear combinations of joint translations and rotations, and can be used to calculate averages, drifts, and other quantities of interest. Generalized displacement output can be obtained for all types of analysis cases and combinations.

Section Cuts

- The resultant (free-body) forces and moments across any cut in the structure can be defined using section cuts. A section cut can have any shape, and can be used to compute story shears, connecting forces, design forces in shear walls, and for many other purposes. Section cut results can be obtained for all types of load cases and combinations.

Video Animations with Time Varying Results

- SAP2000 has the ability to generate video (.avi) files to visually display a set of analysis results that vary over a particular time period, such as in a time history analysis.
- Output plot functions include:
  - Base functions
  - Energy functions
  - Frame functions
  - Generalized Displacement functions
Report Generator

- Pre-formatted printed reports are now available at the push of a button. These reports include all pertinent model data and the results of analysis and design.
- Data is presented in tabulated format, along with graphics, table of contents, and a cover sheet displaying project information and your company name and logo.
- Reports can be created in RTF file format for Microsoft Word or in HTML format for web presentations.
- You can use the built-in template, or create your own template as a company standard or specialized for a particular project.
Output Tables

- Output tables can be sorted, edited and/or exported to Excel or Access for easier, more efficient review of results. Sort by ascending or descending min and max values for any field in any direction in seconds.
- Exported SAP2000 tables can be merged within Microsoft Access to create a single table containing fields from multiple SAP2000 tables to further customize output.
Features-SAP2000 Import and Export Formats

Import and Export Formats

- Microsoft Access
- Microsoft Excel
- SAP2000 Text File (.s2k)
- CIS/2 STEP
- Steel Detailing Neutral File (SDNF)
- AutoCAD (.dxf/.dwg)
- Frameworks Plus
- IGES
- IFC
- Prosteel
- Other analysis software text files
Features-SAP2000 Open Application Programming Interface (OAPI)

In response to requests from many of our users, Computers and Structures, Inc. is pleased to announce the release of an Open Application Programming Interface (OAPI) for SAP2000. This OAPI provides developers of products for CADD and 3D modeling seamless and efficient access to all of the sophisticated analysis and design technology of SAP2000. Third-party developers can now create rich and tight two-way links with SAP2000, allowing for accurate transfer of models into SAP2000, complete control of SAP2000 execution, and extraction of analysis and design information out of SAP2000, all from within their application.

This OAPI is compatible with most major programming languages, including Visual Basic for Applications (VBA). Anyone familiar with Visual Basic programming should find the SAP2000 OAPI syntax easy and intuitive.

All of the OAPI functions are thoroughly documented in a searchable help file. This help file contains information on over 700 different SAP2000 OAPI functions, with the discussion for each function describing in detail the Syntax, the VB6 Procedure, and the Parameters along with Remarks on what the function does and a VBA Example.

Highlights of this OAPI include:
· Direct, fast and efficient access to all of the advanced numerical methods of SAP2000.

· Direct two-way data transfer without intermediate files provides the faster throughput important for large models.

· OAPI connections allow for multiple data exchange cycles between applications without the creation of a new model for each cycle.

· Use of the OAPI by third-party developers all but guarantees that their applications will remain compatible with future releases of SAP2000 – very important to the end-user.

· Because OAPI developed links are robust and transparent, users will have a high level of comfort pertaining to the integrity of the information being transferred – very important to the developer.

· And lastly, any user who does their own programming can now develop their own custom interface for SAP2000, using their own application or any other application that allows for user programming.

This new module allows the power of SAP2000 to be harnessed in innovative and revolutionary ways, offering the developer the ability to tailor the SAP2000 analysis and design engines for use with their own application.

A similar OAPI will soon be available for ETABS. Any interfacing work done using the SAP2000 OAPI will be transferable and reusable with the ETABS, and vice-versa, as the capabilities and protocols of both OAPI’s are practically identical, resulting in a valuable two-for-one advantage.

**BIM Integration: Scenario #1**

- Third-party BIM user and SAP2000/ETABS user reside on the same system.
**BIM Integration: Scenario #2**

- Third-party BIM user and SAP2000/ETABS user reside on the different systems.
Features-Sap2000 for Oil and Gas

SAP2000’s sophisticated analytical methods and design algorithms directly address the Oil and Gas industry’s structural engineering requirements. Analytical features include the ability to easily capture rotational acceleration due to pitch, roll and yaw; perform simple linear or complete nonlinear snap-through buckling analyses; compute frequency-domain response for harmonic and non-harmonic periodic loading; calculate power-spectral density response; apply automated wave loading and API 4F wind loading; pushover analysis and much more. The implementation of the API 4F, API 2A (offshore structures) and Norsok N-004 (offshore structures) design codes allows the engineer to optimize designs with greater accuracy and speed, including punching shear checks. Structural models are very easily created through the integrated 3D SAP2000 environment where users can model all structural geometry, complex built-up sections, loading, and analytical components. Spreadsheet capability is integrated within the graphical environment for creating or modifying the model as well as viewing or processing the analysis and design results.

Customizable reports can be produced that include graphical and tabular data. The Open Application Programming Interface (OAPI) can be used to develop special-purpose plug-in tools or to link with in-house programs.
Rotational Acceleration Loads

- In addition to translational acceleration loads, SAP2000 allows users to apply rotational acceleration loads to easily capture pitch, roll, and yaw effects. These accelerations can be applied as static and/or dynamic loads.

Cable Elements with Large Displacement Options

- SAP2000 offers true nonlinear cable elements with catenary behavior and large displacement analysis. Coupling this with dynamic analysis enables engineers to capture true inertial effects along with the complex changes in behavior as members forces change from tension to compression and vice versa during various lift or launch stages.

- Click Here to view SAP2000 video simulating the lowering of a subsea structure into the splash zone where it is subjected to wave loads at each stage.
API 4F

- For offshore and drilling structures, SAP2000 offers automated API 4F wind load generation and comprehensive design code unity checks. Wind loads can be applied from any angle relative to the structure and local axis of the members. Loads are automatically modified to include the effects of shielding, cladding, and ice.

- The API 4F design code check accounts for the difference between shielded and unshielded wind loads on an element-by-element basis per the design specification.
**Import from StruCAD*3D**

- Input files for StruCAD*3D (Engineering Dynamics Inc., formerly Zentech) can be imported directly into SAP2000 Version 14.
**Design Code Checks**

Comprehensive design code checks for steel and/or concrete frames are available for a multitude of international design codes, including: API 4F, AISC, CSA S-16 Canadian, Eurocode, and more. Code-dependent lateral loads based upon code parameters can be conveniently generated for API 4F wind, ASCE 7-05 wind (open frame or building structures), IBC2006 seismic, NBCC 2005 wind and seismic, and other international standards.

- API 4F / RP2A WSD
- ASCE
- IBC
- ACI
- UBC
- Canadian CSA S-16 / NBCC / CSA A23 (concrete)
- Chinese
- India IS 800 / 1893
- Mexico RCDF (concrete)
- Australia/New Zealand 1170
- Eurocode 3-2005
- Italian UNI 10011

**Buckling Analysis**

- SAP2000 offers linear Eigen buckling and nonlinear snap-through buckling analysis, either of which may be necessary when designing tall mast structures.

- Nonlinear buckling analysis also includes the effects of large displacements.
P-Delta Analysis

- SAP2000 nonlinear P-delta analysis automatically includes both sway ("large Delta") and local ("small delta") member deformations, satisfying the requirements of the AISC Specifications (13th edition) and other design standards.
**Offshore Module**

- The Offshore module offers the API 2A and Norsok N-004 design checks, which include checks for joint-can punching shear. The module also includes Airy, Stokes 5th-order, and cnoidal wave theory options. Water currents can simultaneously load the portions of the structure below the water line with automated corrections for marine growth as a function of depth, drag and inertia effects, and buoyancy.
- The user can view resultant wave velocities and accelerations graphically with color coded plots or produce output tables of the values that can be exported to Excel.
Advanced Dynamic Analysis

- Dynamic time-history analysis can include the effects of linear or nonlinear soil boundary conditions. Soil conditions can be modeled using compression-only springs, gap-friction behavior, and/or multi-linear plastic behavior specified by P-Y and T-Z curves.
- Automatic generation of sinusoidal time-history functions is available for modeling unbalanced rotating loads. Transient and steady-state response can be calculated for the displacements, velocities, accelerations, forces, and stresses, including step-by-step and max/min values.
- SAP2000 can perform periodic time-history analysis for non-harmonic loading, such as reciprocating machinery, operating at any frequency. * Arbitrary load assignments can automatically be converted to mass.
- SAP2000 steady-state harmonic analysis considers the entire “sweep” of machine-speed frequencies all in one load case. Loading may be specified with different phase angles. Supports and links may be given frequency-dependent properties, enabling consideration of radiation damping and complex support systems. Frequency plots of displacement, velocity, acceleration, force, and stress response can be produced at any phase angle, or for the resultant magnitude.
Wave Loading

- View resultant wave velocity and acceleration graphically with color coded plots, or through output tables which can be brought into Excel.
Section Designer

- SAP2000 has an integrated Section Designer makes it easy to define composite and built-up section properties.

- The user can see details of complex built-up section geometries using the rendered extruded view option of SAP2000.

- Using the concepts of insertion point and cardinal point, frame members may be modeled using centerline, top of steel, bottom of steel, or any other arbitrary reference lines. This allows the convenient alignment of members with a known datum. The effect of the eccentricity upon the connections is automatically accounted for. Shell elements may also be offset from the reference surface. Combining frames and shells offset from a common set of connection nodes is an effective way to model composite behavior.

- A layered shell is available to model composite materials. Each layer may have different isotropy and orientation, so that fiber-wrapped and other built-up constructions can be represented.
Prestressed Modal and Eigen Buckling Analysis Options

• Modal analysis and Eigen buckling analysis can be based on a prestressed condition, such as due to thermal, gravity, pressure, and or other loads.

• P-delta and tension-stiffening, as well as other nonlinear effects, may change the modal or Eigen buckling behavior, and the effect of both prestressed and unstressed conditions can be considered simultaneously in the same model.

• ASCE 41 and other design standards have codified the practice of analyzing seismic loads based on prestressed modal analysis.
**Center of Gravity**

- SAP2000 can automatically determine the center of gravity (CG) of the structure based on selfweight and user selected assigned loads.

- If piping, cladding, equipment or other gravity loads were applied as assigned loads, users have the option to include those assigned loads in the CG calculation and report.
Features-Earthquakes

CSI Uses 1906 San Francisco Quake Synthesized by USGS

CSI uses the 1906 San Francisco ground shaking recreated by new technology from USGS.

Using three-dimensional geological and seismic velocity models, scientists at the USGS have recreated what they think was the ground shaking due to the 1906 San Francisco earthquake.

Using seismic accelerograms produced by this new technology from the USGS, CSI software has been used to create animations that show the behavior of some California landmarks subjected to the 1906 San Francisco earthquake.
SAP2000 System Requirements

Processor:

- Minimum: Intel Pentium 4 or AMD Athlon 64
- Recommended: Intel Core 2 Duo, AMD Athlon 64 X2, or better
- A CPU that has SSE2 support is required
- The SAPFire® Analytical Engine includes a multi-threaded solver that can take advantage of multi-core CPUs

Operating System:

- Microsoft® Windows XP with Service Pack 2 or later, Microsoft® Windows Vista, or Microsoft® Windows 7, 32- and 64-bit versions
- With a 64 bit operating system, the SAPFire® Analytical Engine can utilize more than 4 GB of RAM, making it possible to more efficiently solve larger problems

Memory:

- Minimum: 2 GB for XP O/S, 4 GB for Vista/Windows 7 O/S
- Recommended: 4 GB for 32-bit O/S, 8 GB or more for 64-bit O/S
- The problem size that can be solved & the solution speed increases considerably with more RAM
- Vista/Windows 7 requires more RAM than XP for the operating system itself
Disk Space:

- 6 GB to install the program.
- Recommended: 500GB or larger Hard Disk Drive (7200 rpm SATA)
- Additional space required for running and storing model files and analysis results, dependent upon the size of the models

Video Card:

- Minimum: Supporting 1024 by 768 resolution and 16 bits colors for standard (GDI+) graphics mode
- Recommended: Discrete video card with NVIDIA GPU or equivalent and dedicated graphics RAM (512 Mb or larger) for DirectX graphics mode. The card must be DirectX 9.0c compatible (DirectX SDK Aug 2009 - Build 9.27.1734.0).
- DirectX graphics mode fully utilizes the hardware acceleration provided by a GPU and dedicated graphics RAM.
- For better graphics quality in terms of anti-aliasing and line thickness, the device raster drawing capabilities should support legacy depth bias.

SAP2000 Levels and Features

<table>
<thead>
<tr>
<th>Physical Modeling Features</th>
<th>Basic</th>
<th>Plus</th>
<th>Advanced</th>
<th>Ultimate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model joint size limit</td>
<td>1500</td>
<td>Unlimited</td>
<td>Unlimited</td>
<td>Unlimited</td>
</tr>
<tr>
<td>All drafting and model generation features</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>3D line and area objects</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>3D solid objects</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Database of all standard hot rolled steel, aluminum, and cold formed sections</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Section Designer for specialized sections</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Interactive database spreadsheet editing</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Analytical Modeling Features</th>
<th>Basic</th>
<th>Plus</th>
<th>Advanced</th>
<th>Ultimate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Meshing tools</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Automatic mesh generator</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Loading Features</td>
<td>Basic</td>
<td>Plus</td>
<td>Advanced</td>
<td>Ultimate</td>
</tr>
<tr>
<td>---------------------------------------------------------------------------------</td>
<td>-------</td>
<td>------</td>
<td>----------</td>
<td>----------</td>
</tr>
<tr>
<td>Point, line, trapezoidal, and area loads</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Tributary area load distribution to frames</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Automatic code-based wind loading</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Open structure wind loading (multiple codes)</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Automatic code-based seismic loading</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Pattern loading</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Applied displacement loading</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
</tbody>
</table>
Moving loads on frame elements
Gravity, pressure and thermal loading
Strain loads, deformation loads, target force
Prestress loads
Wave loading

<table>
<thead>
<tr>
<th>Analysis Features</th>
<th>Basic</th>
<th>Plus</th>
<th>Advanced</th>
<th>Ultimate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fast advanced solver technology with SAPFireTM analysis engine</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Multiple 64-bit solvers for analysis optimization with unmatched solution capacity</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Generalized joint constraints including: diaphragms, plates, rods and beams</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Eigen analysis with auto-shifting for ill-conditioned problems</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Ritz analysis for fast predominant mode evaluation with missing mass</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Multiple response spectrum cases in single run</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Modal combination by the CQC, SRSS, GMC or double sum methods</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Directional combination by the ABS, SRSS, or CQC3 methods</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Linear dynamic modal time history analysis</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Linear dynamic direct integration time history analysis</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Frequency domain analysis</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Power spectral density analysis</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Buckling analysis</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Nonlinear analysis</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Tension/compression-only frame elements</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Tension/compression only soil (area)</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>P-delta analysis - both small and large</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
</tbody>
</table>
P-delta

Static pushover analysis

Wilson FNA (Fast Nonlinear Analysis) method

Direct integration time history

Line and surface multi-linear springs (P-y curves)

Material nonlinearity - frame hinges and links

Geometric nonlinearity - large displacement

Target final geometry iterations

Creep and shrinkage

Staged construction

Staged construction – change property modifiers for frames, tendons, and shells

Staged construction – change section properties for frames, tendons, and shells

Hyperstatic analysis for secondary effects of prestressing forces

Static and dynamic load combos - linear, envelope, absolute, SRSS, range

<table>
<thead>
<tr>
<th>Design Features</th>
<th>Basic</th>
<th>Plus</th>
<th>Advanced</th>
<th>Ultimate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel frame design - American and international design codes (19+)</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
<tr>
<td>Aluminum frame design</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
<tr>
<td>Cold formed frame design</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
<tr>
<td>Concrete shell required reinforcement display</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
<tr>
<td>Load combinations can be converted to nonlinear load cases for Advanced design</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Output and Display Features</th>
<th>Basic</th>
<th>Plus</th>
<th>Advanced</th>
<th>Ultimate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformed and undeformed geometry in 3D</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
<tr>
<td>Loading diagrams</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
<td>✔️</td>
</tr>
<tr>
<td>Feature</td>
<td>Basic</td>
<td>Plus</td>
<td>Advanced</td>
<td>Ultimate</td>
</tr>
<tr>
<td>------------------------------------------------------------------------</td>
<td>-------</td>
<td>------</td>
<td>----------</td>
<td>----------</td>
</tr>
<tr>
<td>Moment, shear and axial force diagrams</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Stress contours for area and solid objects</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Resultant forces displayed along section cuts</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Detailed results with right button click</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Virtual work plots</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Time history displays of function vs. time</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Force vs. deformation plots</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Response spectrum curves from time history response</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Video of animations and time varying results displays</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Capture of graphics to .emf, .jpg, .bmp, .tif</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Customized report generation</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Quick generation of final printed reports with complete user control</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Cover sheet identifying client, project, user and company</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Formatted tables of model definition, analysis and design results</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Graphical displays of the model</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>All Import &amp; Export Formats</th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Microsoft Access</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Microsoft Excel</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>SAP2000 text file (.s2k)</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Export to Perform-3D</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>CIS/2 - STEP</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Steel detailing neutral file (SDNF)</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>AutoCAD (.dxf/.dwg)</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
<tr>
<td>Frameworks Plus</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
<td>✔</td>
</tr>
</tbody>
</table>
IGES

IFC

Prosteel

Other analysis software text files (Import only)

SAP2000 Watch and Learn

<table>
<thead>
<tr>
<th>Title</th>
<th>Length</th>
<th>Click to Play</th>
</tr>
</thead>
<tbody>
<tr>
<td>SAP2000 - 01 Introductory Tutorial</td>
<td>30 min</td>
<td><img src="http://www.youtube.com/watch?v=2povDdw2pTQ" alt="SAP2000 - 01 Introductory Tutorial" /></td>
</tr>
<tr>
<td>SAP2000 - 02 Select Commands</td>
<td>6 min</td>
<td><img src="http://www.youtube.com/watch?v=Ck1-a4asRhA" alt="SAP2000 - 02 Select Commands" /></td>
</tr>
<tr>
<td>SAP2000 - 03 Draw Commands</td>
<td>9 min</td>
<td><img src="http://www.youtube.com/watch?v=aLCMg0P6pmw" alt="SAP2000 - 03 Draw Commands" /></td>
</tr>
<tr>
<td>SAP2000 - 04 Mass and Modal Analysis</td>
<td>5 min</td>
<td><img src="http://www.youtube.com/watch?v=3ZbKg-oVC5I" alt="SAP2000 - 04 Mass and Modal Analysis" /></td>
</tr>
<tr>
<td>SAP2000 - 05 P-Delta Analysis</td>
<td>10 min</td>
<td><img src="http://www.youtube.com/watch?v=YzWocAtK03I" alt="SAP2000 - 05 P-Delta Analysis" /></td>
</tr>
<tr>
<td>Course Title</td>
<td>Duration</td>
<td></td>
</tr>
<tr>
<td>------------------------------------------</td>
<td>----------</td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 06 Nonprismatic Sections</td>
<td>6 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=ihltIW9AaSU">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 07 Interactive Database Editing</td>
<td>7 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=YIrv_W0xlT0">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 08 Area Edge Constraints</td>
<td>3 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=nbotSbo1odU">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 09 Tension-only Bracing</td>
<td>4 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=JkhYzTtwseKE">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 10 Response Spectrum Analysis</td>
<td>5 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=mryI8Jij7F0">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 11 Modal Time History Analysis</td>
<td>14 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=VjkqA4_hLMk">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 12 Section Designer</td>
<td>13 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=1a-0Vbuejcs">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 13 Cardinal Points</td>
<td>9 min</td>
<td></td>
</tr>
<tr>
<td><a href="http://www.youtube.com/watch?v=EHkm8TRywn0">Video Link</a></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAP2000 - 14 Displaying Tabular Data</td>
<td><a href="http://www.youtube.com/watch?v=nda5Mov-82k">http://www.youtube.com/watch?v=nda5Mov-82k</a></td>
<td>9 min</td>
</tr>
<tr>
<td>--------------------------------------</td>
<td>---------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>SAP2000 - 15 Creating Reports</td>
<td><a href="http://www.youtube.com/watch?v=SFNS22hUObk">http://www.youtube.com/watch?v=SFNS22hUObk</a></td>
<td>11 min</td>
</tr>
<tr>
<td>SAP2000 - 16 Cable Objects</td>
<td><a href="http://www.youtube.com/watch?v=xXZ77VoCZO8">http://www.youtube.com/watch?v=xXZ77VoCZO8</a></td>
<td>18 min</td>
</tr>
<tr>
<td>SAP2000 - 17 Tendons</td>
<td><a href="http://www.youtube.com/watch?v=0vHvYa4qPWQ">http://www.youtube.com/watch?v=0vHvYa4qPWQ</a></td>
<td>10 min</td>
</tr>
<tr>
<td>SAP2000 - 18 Gap Elements</td>
<td><a href="http://www.youtube.com/watch?v=2xh4gRX9_A0">http://www.youtube.com/watch?v=2xh4gRX9_A0</a></td>
<td>17 min</td>
</tr>
<tr>
<td>SAP2000 - 19 Nonlinear Staged Construction</td>
<td><a href="http://www.youtube.com/watch?v=L-HT3KTUa4w">http://www.youtube.com/watch?v=L-HT3KTUa4w</a></td>
<td>7 min</td>
</tr>
<tr>
<td>SAP2000 - 20 Nonlinear Shear Walls</td>
<td><a href="http://www.youtube.com/watch?v=Y7z6IYxF-eQ">http://www.youtube.com/watch?v=Y7z6IYxF-eQ</a></td>
<td>29 min</td>
</tr>
<tr>
<td>SAP2000 - 21 Static Pushover Analysis</td>
<td><a href="http://www.youtube.com/watch?v=CzqZVDse-eI">http://www.youtube.com/watch?v=CzqZVDse-eI</a></td>
<td>10 min</td>
</tr>
<tr>
<td>-------------------------------------</td>
<td>-----------------------------------</td>
<td>--------</td>
</tr>
<tr>
<td>SAP2000 - 23 Wave Loading</td>
<td><a href="http://www.youtube.com/watch?v=GCDj25WeCOI">http://www.youtube.com/watch?v=GCDj25WeCOI</a></td>
<td>5 min</td>
</tr>
<tr>
<td>SAP2000 - 24 Model Alive</td>
<td><a href="http://www.youtube.com/watch?v=QyJCmZ0cziU">http://www.youtube.com/watch?v=QyJCmZ0cziU</a></td>
<td>4 min</td>
</tr>
<tr>
<td>SAP2000 - 25 Open Application Programming Interface</td>
<td><a href="http://www.youtube.com/watch?v=QRdXGi0g0BE">http://www.youtube.com/watch?v=QRdXGi0g0BE</a></td>
<td>13 min</td>
</tr>
<tr>
<td>SAP2000 - 26 CSiLoadOptimizer</td>
<td><a href="http://www.youtube.com/watch?v=2iGsJMY_-84">http://www.youtube.com/watch?v=2iGsJMY_-84</a></td>
<td>10 min</td>
</tr>
<tr>
<td>SAP2000 - 27 Buckling Factors and Modes</td>
<td><a href="http://www.youtube.com/watch?v=qXoPMkNmwrQ">http://www.youtube.com/watch?v=qXoPMkNmwrQ</a></td>
<td>9 min</td>
</tr>
</tbody>
</table>

Disclaimer: In an effort to highlight certain key capabilities, artistic license may have been taken during the making of these videos. Some input may have been skipped for the sake of brevity, resulting in movies that do not portray all of the steps necessary to create a complete analysis or design. Users should consult the associated program documentation to make sure that they understand all input and modeling assumptions.

Note: If you would like to download the Watch and Learn videos for offline viewing, please click [here](http://www.youtube.com/watch?v=MmdPgA-RLdE).
SAP2000 V16 News

SAP2000 V16.1.0 Enhancements

Significant enhancements include:

- Steel frame design has been added according to the Norsok N-004 2013 code.
- Steel frame design according to the API RP 2A-WSD 21st edition code has been updated for supplements 1, 2 and 3.
- Auto lateral wind load has been added according to the API 4F 2013 code.
- The Chinese section-property database files have been enhanced by adding new sections.
- A bilinear Maxwell damper has been implemented as a new link property. This device is a linear spring in series with a dashpot whose force-velocity relationship exhibits bilinear viscous behavior, typical of certain oil dampers having a relief valve.
- A friction-spring hysteretic damper has been implemented as a new link property. The force-displacement relationship exhibits linear slipping stiffness when loading, but unloads with a smaller slipping stiffness. A pre-compression displacement and a displacement stop-limit may be specified.
- The specification of the notional size used for time-dependent creep and shrinkage analysis has been enhanced to now be specified with the frame and shell section properties instead of with the material property.
- A new staged-construction operation “Change Section & Age” is now available allowing specification of the “Age at Add” when changing frame sections or shell sections.
- Table output can now request the correspondence between the response components at a single location for additive-, range-, and enveloping-type load combinations. Results that admit correspondence includes most displacement, force, and stress response quantities.
- The import of primary data from SACS data files is now available.
SAP2000 V16.0.0, V16.0.1 and V16.0.2 Enhancements

Significant enhancements include:

- Improved graphics performance
- Display of multiple grid systems at the same time
- New steel frame design code AISC 360-10
- New steel frame design code NTC 2008
- New concrete frame design code ACI 318-11
- New concrete frame design code NTC 2008
- New concrete frame design code NZS 3101-06
- New concrete frame design code RCDF 2004
- New concrete frame design code Hong Kong CP 2013
- Enhanced beam-column capacity check for concrete frame design using Eurocode 2-2004
- Stiffness modifiers from the AISC 360 steel frame design direct analysis method are now available in the database table for assigned frame property modifiers
- New wind, seismic, and response-spectrum lateral loads for code IBC 2012
- New wind lateral loads for code AS/NZS 1170.2-2011
- New wind, seismic, and response-spectrum lateral loads for code NTC 2008
- New wind, seismic, and response-spectrum lateral loads for codes TSC 2007 and TS 498-97
- New response-spectrum lateral loads code AASHTO 2012
- New assignment to control area loads transferred to frames
- New hybrid U-girder frame section property
- New built-in ASTM bridge steel materials
- Chinese concrete material and design updated to use characteristic strength rather than grade
- New link property for modeling triple-pendulum bearings
- External tendons can now be modeled under user control
- Plotting of tendon loads, losses, and axial-force response has been enhanced
- Ground displacement loading now applies to single-joint links, including distributed spring supports, as well as to springs and restraints
- Multiple mass sources can be defined for modeling alternative dynamic behavior
- The undeformed shape can now be viewed for various stages of staged construction load case without running the analysis
- Response output for staged-construction load cases now is labeled with the stage and step within the stage
- Shear stress, von Mises stress, and principal stress response is now available for certain frame sections
• Shell stresses can now be displayed normalized by material strength
• Bearing pressure on shells from area springs can now be displayed
• Display of frame-design D/C ratios can be limited by a specified threshold value
• Verification suites are now provided for steel and concrete frame design by most codes
• New Open API functions to define section-cuts
• New Open API functions to define external load cases for frame results
• New Open API functions to get detailed steel frame design results for certain codes
• Customizable keyboard shortcuts for menu commands
• Support for the import and export of files using the IFC 4 format
• Licensing will now support web-activation and virtual servers
• Simplified installation using the CSI Installation Wizard
• Other minor enhancements