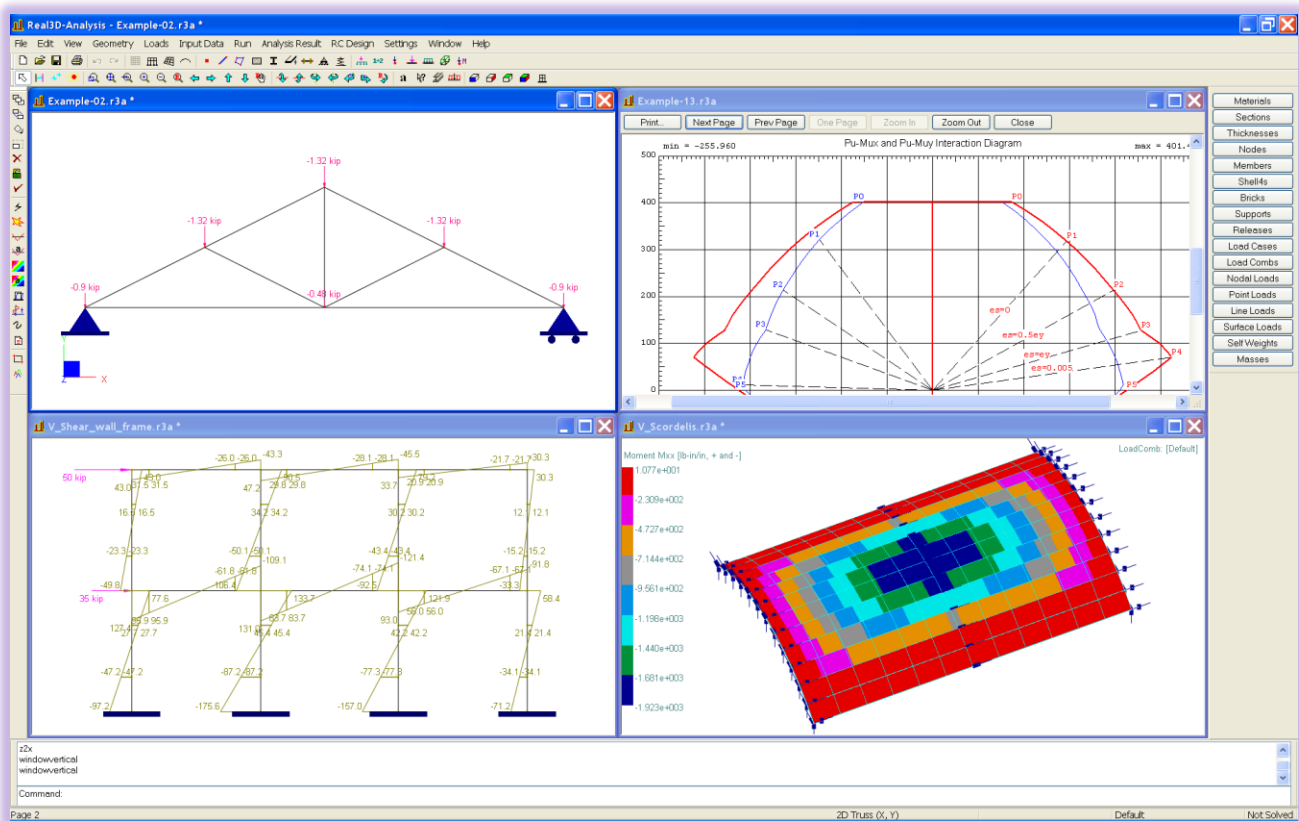


Real3D-Analysis

Real3D-Analysis is a powerful frame-finite element analysis and design program built from ground up, with latest technologies from the fields of finite element analysis, numerical computation and computer graphics. It brings accuracy, reliability, and ease of use to average structural engineers to perform static and dynamic analyses and design of general truss, frame, plate and shell structures. It features unique 128-bit floating point solver, multiple document interface, spreadsheet input and output as well as compelling graphics built upon industry standard OpenGL(R).



Frame & Finite Elements

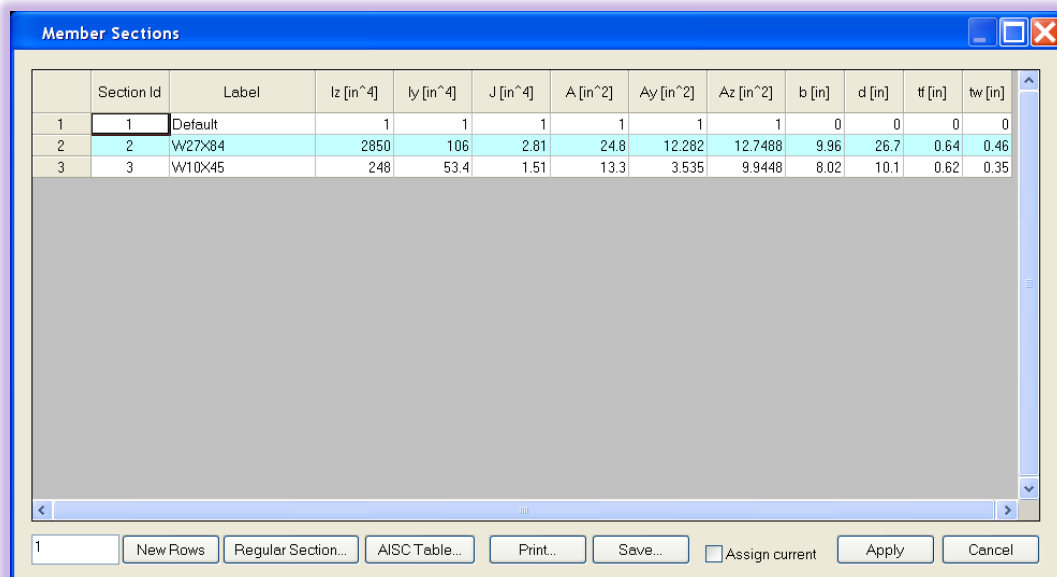
- 2D and 3D frame (beam and truss) elements
- 2D and 3D four-node plate/shell element, with thick or thin plate bending element (for slab modeling) and compatible/incompatible in-plane stress element (for shear wall modeling)
- 3D eight-node solid (brick) element with compatible and incompatible formulations
- Linear, compression-only, tension-only nodal, line and surface spring elements
- Moment releases on frame element
- Forced displacements on supports
- Nodal, point forces and moments, line, true area load (on members), surface forces (on shells) and self weight

Analysis and Design Options

- Static linear analysis
- Geometric nonlinear (P-Delta) analysis
- Frequency analysis
- Standard 64-bit Skyline and Sparse Solver and unique 128-bit floating point Solver (extremely accurate)
- Concrete design for beams, columns and slabs according to ACI 318-05, -02. It includes such as features as exact biaxial column interaction diagrams, automatic moment magnification, cracked section properties, Wood-Armer moments, color coded plots for member capacity and reinforcement contour plots for plates

User Interface Features

- Multiple documents may be opened at the same time; each document may have multiple views with different display settings.
- Spreadsheets for input data and results
- Graphically drawing nodes, frames and finite elements via mouse click or keyboard in command window
- Quality 3D graphical rendering built on OpenGL(R) with hidden line or surface removal. Graphics display includes loading diagram, moment and shear diagrams for beams, contours for shells and solids, deflection diagram and annotations for input and results. Structural response animation is available for result diagrams and contours
- Powerful automatic model generations for continuous beams, 2D and 3D frames, 2D and 3D shells, arc beams and non-prismatic beams. Include AISC steel shape database, ASTM rebar database, regular sections and standard load combinations

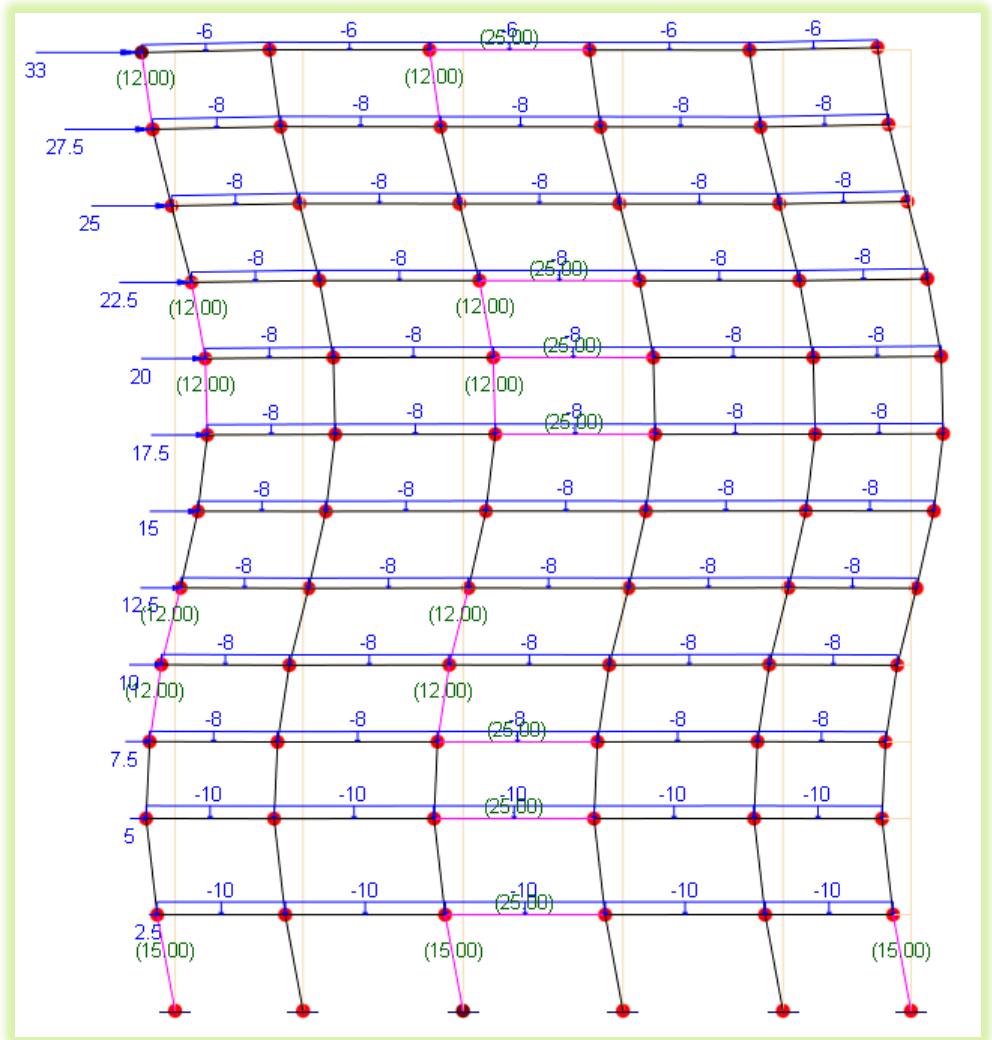


The screenshot shows a software window titled "Member Sections" with a table of section properties. The table has 13 columns: Section Id, Label, Iz [in^4], Iy [in^4], J [in^4], A [in^2], Ay [in^2], Az [in^2], b [in], d [in], tf [in], tw [in], and a final unlabeled column. There are three rows of data. Row 1 is the 'Default' section. Row 2 is a W27x84 section. Row 3 is a W10x45 section. Below the table is a large grey area for a graphical view. At the bottom are buttons for 'New Rows', 'Regular Section...', 'AISC Table...', 'Print...', 'Save...', 'Assign current' (checkbox), 'Apply', and 'Cancel'.

	Section Id	Label	Iz [in ⁴]	Iy [in ⁴]	J [in ⁴]	A [in ²]	Ay [in ²]	Az [in ²]	b [in]	d [in]	tf [in]	tw [in]
1	1	Default	1	1	1	1	1	1	0	0	0	0
2	2	W27x84	2850	106	2.81	24.8	12.282	12.7488	9.96	26.7	0.64	0.46
3	3	W10x45	248	53.4	1.51	13.3	3.535	9.9448	8.02	10.1	0.62	0.35

- Flexible editing features such as undo/redo, duplicate, move, scale, delete, revolve, extrude, splitting beams, sub-mesh shells, node and element merging

- Many different selection methods such as window/point select, select by IDs, select by properties etc, with options to freeze or thaw parts of a model. Ability to pan, zoom and rotate in real time
- Text and graphic report in html format. Graphic report may contain multiple images. Text report may be saved in plain text format.
- Print previews for graphics and text reports
- Importing from and exporting to DXF files
- English or Metric units or mixture of the two
- True 32-bit, fully integrated Windows application written from ground up entirely (graphical user interface and solver) in standardized, object-oriented C++ programming language, resulting in a truly robust software for engineering community

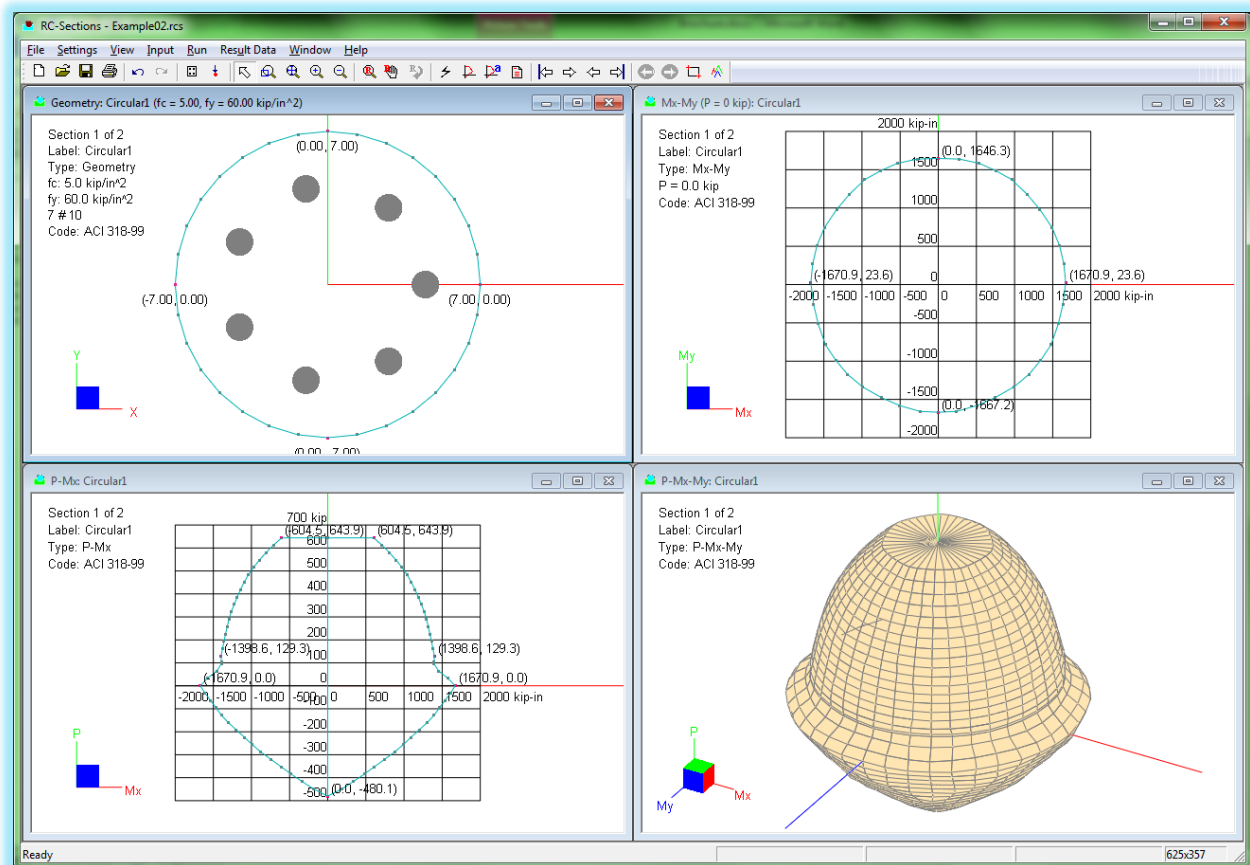


System Requirements

- Computer CPU: Intel Pentium based; 450 MHz or faster is recommended
- Memory (RAM): Minimum 64 MB; 128 MB or more is recommended
- Operating System: Windows 2000, XP, Vista or Windows 7
- Video Card: Minimum 4MB, OpenGL(R) hardware acceleration essential

cColumn

cColumn is a powerful 32-bit Windows program designed specifically for structural engineers to perform axial-flexural analysis and design of concrete sections (beams, columns or walls) according to ACI 318-11/08/05/02 and ACI 318-99. It analyzes the uniaxial- or biaxial-bending capacity of multiple sections of various regular shapes (rectangular, round, Tee, I, L) and generic shapes (with openings) simultaneously in a fast and user-friendly manner. Multiple load sets may be input, magnified and checked against capacities of the sections. It features tabulated strain-P-M at every user defined neutral axis step, 2D P-Mx and P-My interaction diagrams, and incredible 3D P-Mx-My interaction surface built upon OpenGL(R)!

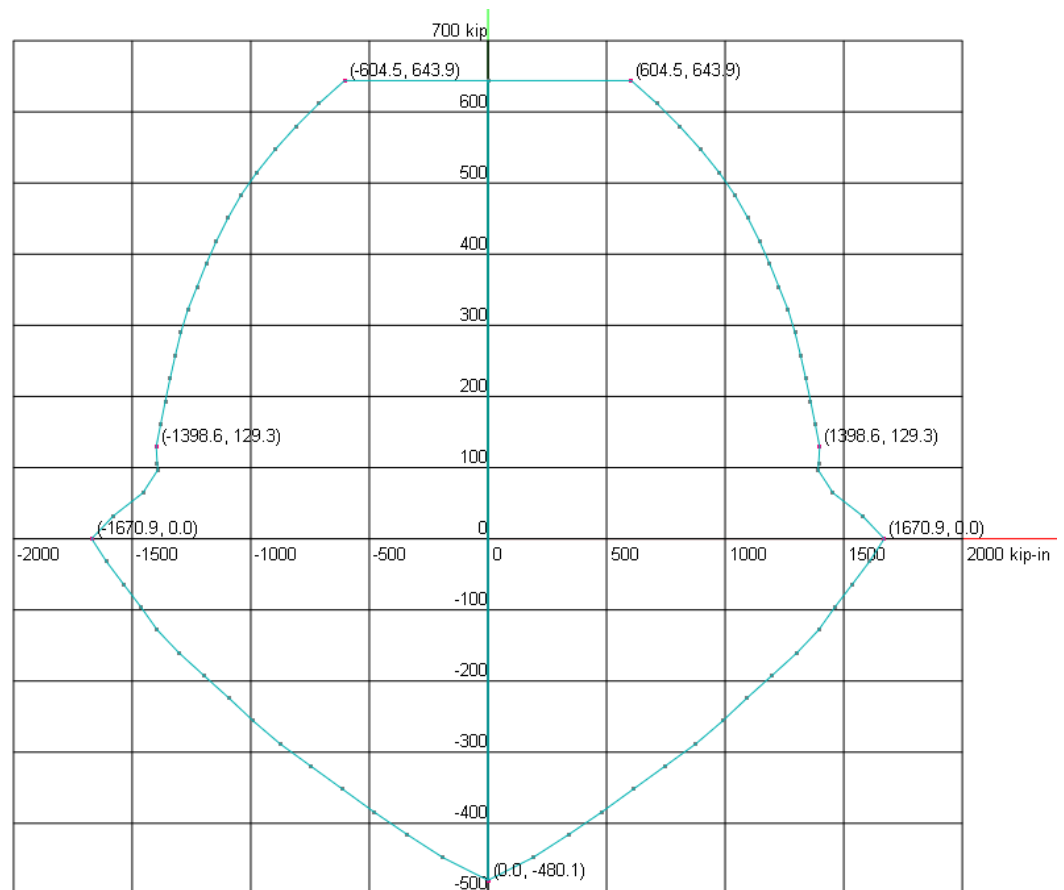


Key Features

- Codes: ACI 318-11, -08, -05, -02 and -99
- Unit: English, Metric or Mixed
- Section Shapes: Unlimited number of rectangular, tee, circular, I, inverse L and generic (with up to 3 openings) sections. Ability to parametrically generate multiple rectangular or circular sections with different sizes and reinforcement configurations. Great tool for shortening the trial-and-error design process
- Loads: Unlimited number of load sets (P_u , M_{ux} , M_{uy}). Moments can be magnified by considering column slenderness. Adequacy of each section to carry the loads is computed in terms of simple capacity ratio

- Solver: Unique, exact and blazingly fast solution algorithms that can accurately solve hundreds even thousands of sections uniaxially or biaxially simultaneously in seconds. Solution options include strength reduction factor, tied or spiral confinement, variable number of neutral axis steps, variable biaxial angle steps, variable axial steps for display and inclusion or exclusion of displaced concrete
- Result Data: Very detailed result data tabulated in spreadsheets. P-M result includes neutral axis depth, eccentricity, maximum tensile strain and strength reduction factor, as well as axial capacity P and moment capacity M

- Diagrams: 2D P-Mx, P-My and Mx-My interaction diagram and 3D P-Mx-My interaction surface or wire-frame. Key points shown automatically on the interaction diagrams. Input loads shown with different colors on the 2D/3D diagrams or surface denoting the adequacy of the section for each load



- Printing: Automatic printing of very impressive interaction diagrams for multiple sections. If you have installed PDF-making software, you will be able to create your very own design handbook similar to CRSI Handbook!

System Requirements

- Computer CPU: Intel Pentium based; 450 MHz or faster is recommended
- Memory (RAM): Minimum 64 MB; 128 MB or more is recommended
- Operating System: Windows XP, Vista or Windows 7
- Video Card: Minimum 4MB, OpenGL(R) hardware acceleration essential