

### RamSeries Introduction

RamSeries is a complete finite element (FEM) environment for structural analysis providing the ability to simulate every structural aspect of a product, including linear statics, modal analysis, linear and nonlinear transient simulations, impact, coupled fluid-structure analyses, thermo-mechanical studies, composite materials, fatigue assessment and much more. RamSeries is the simulation solution to rapidly solve complex structural problems with ease.

RamSeries features a full range of 3D element models, including beams, cables, shells, membranes and solid elements, as well as a full support for nonlinear analysis, including nonlinear material laws, and large displacement models. Furthermore, RamSeries offers the latest technology for solving structural dynamics analyses, including robust contact-impact algorithms required for many complex design problems in engineering.

RamSeries is part of the simulation solution Tdyn, featuring a comprehensive analysis suite that provides solution to nearly any multiphysics engineering simulation need. The solution is fully integrated in a unique Graphical User Interface (GUI) for easy management of the geometry and analysis data, featuring a versatile tree-like interface and completely automatic mesh generation and tools for visualizing results. The package can also be adapted for specific requirements, allowing the user to perform automated tasks and simplify the analysis process.

### Modules Information

#### Basic: linear static analysis

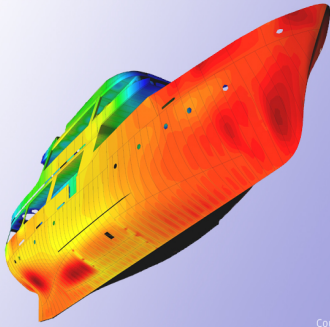
This module features a 3D Finite Element (FEM) analysis package in linear static conditions. The advanced FEM technology provides rich functionality, allowing combination of beams, shells and solid elements, advanced constraints definition, disconnection of degrees of freedom (ball/cylindrical-joint), flexible load definition and combination of load cases.

Model definition process is accelerated thanks to the different available tools, including predefined and customizable libraries, containing common material properties and standard beam sections. Furthermore, the most advanced solution techniques, including parallel sparse direct and iterative solvers, reduce the solution time required for large models.

### CAPABILITIES CHART

Fully integrated graphic user interface (GUI)	✓
Line models: 3D beams and cables	✓
Shell element models: 4-8 nodes quads, 6 nodes triangles, DKT, 3 nodes triangles with drilling rotation, and stiffened shells, 3-noded membrane element	✓
3D solid elements: 4-10 nodes tetras and 8-20 nodes hexas	✓
Advanced automatic structured and unstructured meshes	✓
Linear statics, Dynamic, modal, seismic and direct time integration analysis	✓
Calculation of main vibration modes for free or constrained models with participation mass assessment	✓
Surface to surface contact-impact algorithm, featuring sticky and frictionless contacts	✓
Nonlinear tools: large displacements, material plasticity J2 and Non linear constraints	✓
P DELTA buckling-analysis method	✓
Composite laminated shells and beams	✓
Material libraries including a collection of concrete, steels and non-linear materials	✓
Metallic profiles database and user-defined sections	✓
Various local axes definition techniques for anisotropic and laminate composite materials including geometric line collections	✓
Fatigue assessment (DNV-RP-C203, API RP 2A-WSD)	✓
One-way and two-way implicit fluid-structure interaction solvers (Ransol module required), seakeeping-structure coupled solvers (SeaFEM package required) and Thermo-mechanical solver (Heatrans module required)	✓
Cutting-edge parallel CPU-SMP direct/iterative sparse-matrix solvers	✓
Scripting programming interface based on Tcl-Tk	✓

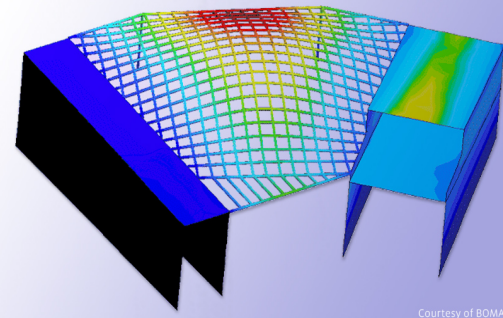
RamSeries pre-processing tools allow the generation of any CAD geometry and support importation of standard CAD files by using standard IGES, PARASOLID and DXF formats, among others. RamSeries meshing technology includes a suite of tools to create automatically, high quality unstructured, structured and semi-structured meshes of different element types.



Courtesy of Isonaval



Courtesy of CBMC



Courtesy of BOMA

### Dynsol: dynamic (modal and direct) analysis

Dynsol module includes the latest technology for dynamic analysis of structures, featuring different types of dynamic analyses including step-by-step direct integration, modal and seismic analysis. The module supports static and dynamic load definition, either by analytical functions or data tables, response spectra and several modes of damping definition.

The program can calculate any number of main vibration modes of the structure for either a free or a constrained model. As a result, the frequency, deformation and mass participation for the mode are provided.

### Composite: laminated composite calculation tools

RamSeries offers advanced utilities to analyze structures made of beam or shell laminated composites, and provides an intuitive interface to define materials, laminate sequences and ply direction, based on geometric lines orientations or local or global frames of reference.

The package provides a library of commonly used materials as well as of pre-defined laminates. RamSeries can manage a global database of predefined sequences and materials to be shared by a design team.

It is possible to display stress and deformation results as well as failure criteria for either the global composite or per ply. LaRC04 failure criteria, including six failure modes, and Tsai-Wu criteria are available.

### Non-linear: non-linear, impact and advanced tools

RamSeries features different material constitutive laws in order to analyze complex model behavior. Among others, it is possible to characterize material behavior with J2 plasticity for shell elements, including Isotropic or Kinematic hardening. The geometry can be analyzed assuming large displacements for all the element types. Furthermore, non-linear effects can be also applied to boundary conditions.

Contact-impact algorithms use the most advanced element to element approach. This method facilitates the definition of the contact regions, as collections of surfaces in the geometry. The contact is calculated without the need of any penalty value estimation and can be combined with the large displacements approach. Contacts can be modelled either as frictionless or sticking.

Furthermore, RamSeries includes specific tools for marine and offshore engineering applications, including stiffened shells, static and dynamic wave loads, panel buckling and fatigue assessment tools. These tools are adapted to the Classification Societies rules for naval architecture and offshore assessment.

### Coupling: implicit fluid-structure interaction solver

RamSeries offers seamless communication with Tdyn's Heatrans and Ransol modules for thermo-mechanical and coupled fluid-structure analyses.

Furthermore, RamSeries features one-way and two-way interaction links to the Tdyn's seakeeping package SeaFEM, allowing to perform the most advanced wave-structure interaction studies.

Typical coupled problems that can be solved with RamSeries include: thermo-mechanical, aeroelasticity, hydroelasticity and waves-structure interaction analysis.

### INFORMATION

For further information, please visit:

<http://www.compassis.com/ramseries>

A free version of RamSeries, limited in the number of mesh nodes, can be downloaded from the Compass website. A one month trial password for activating all the capabilities for demo purposes can be obtained at the same site.

### System requirements

Windows XP / XP64 / Vista / Vista64 / 7 / 8 / 8 64-bit or Linux 32/64.

Please ask us for RamSeries versions on other platforms.

Minimum requirements: 2.0 GB RAM (4 GB for 64 bits editions) and 500 MB of free hard disk space.

Supports any graphics card with OpenGL acceleration.  
Supports multiple processor parallel technology.

For information about conditions and prices of commercial licences, please contact [info@compassis.com](mailto:info@compassis.com)