SAMCEF Field

RESULTS ANALYZE

As soon as the analysis is completed, the results are easily accessible from a simple click in the navigator.

Display function have been improved. In addition to standard graphical tools allowing to display results on the geometry following color cards, by symbols, on the initial or deformed geometry or post-processing on selected portions, you can also display the discontinuity of the stress fields and display "stress tensor" on the model to visualize the stresses in 3D.

Among the new features, you have the possibility to export directly your result files to EXCEL. More specifically, for composite analyses, the "composite viewer" allows to access to results in the different laminate plies in any place in the structure.

Lastly, for your report creation or your internal presentations, navigation and video functionalities (zoom, replay, etc.) allow to scan a shape, display isosurfaces…, and then, save these representations under an image or a video format.

RESULTS

- Direct access to the results through the data tree, then, according to the units to the different criteria…;
- Visualization in shaded formats or using symbols;
- Display on all the model or on a part of it;
- Display by iso-surface or shape;
- Result listing for the selected nodes or element which can be exported in EXCEL;
- Internal results in mechanical assemblies;
- Results on composite layers;
- Function of one point;
- Image (.gif) and animation (.avi) generation procedure.
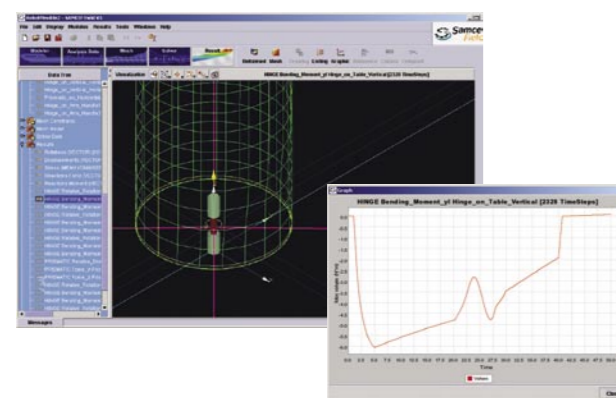
DOCUMENTATION

For direct access to information, the Users Guide and Help manual are available via your favorite navigator (HTML).
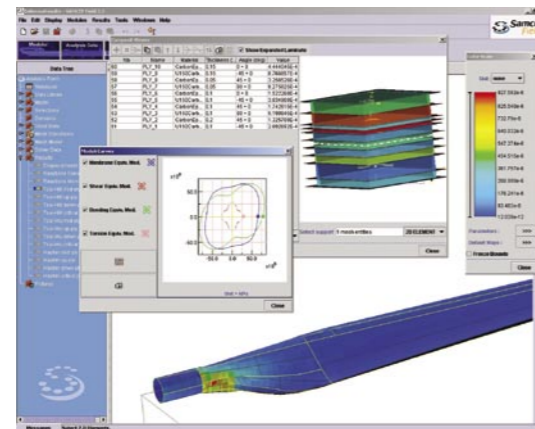
COUPLING SAMCEF FIELD WITH OTHER SAMCEF PRODUCTS FAMILY

The file compatibility between SAMCEF software makes it possible to perform linear and non-linear analysis on the same model.

It is also possible to perform a modal analysis or a linear buckling analysis after a non-linear calculation (SAMCEF Mecano).
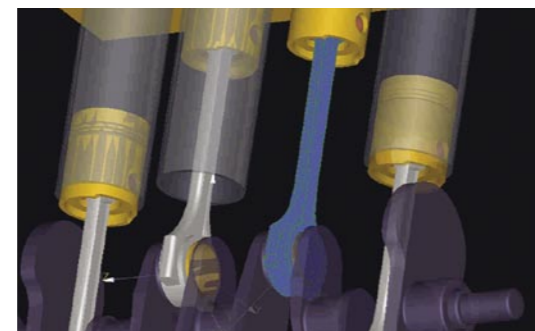
- SAMCEF Asef, Dynam, Stabi: linear static, modal and buckling analyses;
- SAMCEF Mecano: unique integrated software that solves non-linear structures and mechanisms problems. The software is declined in different modules to provide a more specific answer to following analyse;
  - o MECANO Structure: dedicated to the non-linear analysis of structures;
  - o MECANO Motion: dedicated to the static, kinematical and dynamic analyses of flexible mechanisms;
- SAMCEF Thermal: stationary thermal analyses.

PLATFORMS

SAMCEF Field is available on Window NT, 2000, XP Pro, SUN and Linux.

About SAMTECH

Founded in 1986 from the Aerospace Laboratory of the University of Liège to allow the development and commercialisation of the general purpose Finite Element Analysis package SAMCEF, SAMTECH is now the leading European company for the development of Integrated CAE Solutions. SAMTECH provides its expertise to industries like SNECMA, AIRBUS, EADS, ALENIA, ESA, SONACA, ABB, PSA, ... willing to increase their competitiveness by decreasing design time and costs and optimizing their methodologies. In order to answer this objective, SAMTECH has structured its competences around two main complementary activities:

- The "**Software edition**" activity, which includes the development of scientific software tools like the general FEA package SAMCEF for linear or non-linear analysis (Field, Linear, Mecano, Thermal…), BOSS quattro (application manager and optimization platform), EUROPLEXUS (explicit analysis and rapid dynamic) and TEA CAA V5 Based (TEA Mecano and TEA Thermal); the edition of "professional solutions" like SAMCEF Rotor (rotor dynamics) SAMCEF for Machine Tools (design and analysis of Machine Tools), SAMCEF Bolt (analysis of composite assemblies) and SAMCEF Hvs (Analysis of High Voltage Substations)… answering to specific industrial needs; the edition of "third party tools and customized solutions"; the edition of "customized multi-physics solutions".

- The "**Engineering services**" activity, which provides customized solutions such as consultancy, intellectual services and on-the-road projects. It includes also assistance to prime contractors and high-tech subcontracts to engineering offices, which belongs to industries, as well.

Some References

Spatial aeronautics

EADS-LV, EADS AIRBUS, EADS-CCR, EUROCOPTER, SNECMA, SNECMADMF, MESSIER DOWTY, MTU, HUREL DUBOIS, HISPANO, LATECOÈRE, CRYOSPACE, ALTAL, SABCA, SONACA, TECHSPACE AERO, ESTEC, ASC, BOEING HELICOPTER, DASA, IBERESPACIO, BOMBARDIER AEROSPACE, FIAT AVIO, ALENIA AERO.

Transport

PSA, RENAULT, FORD, VALEO, DAIMLER-CHRYSLER, AUDI, LOHR, VW, DELPHI AUTOMOTIVE SYSTEMS, FIAT AUTO, RENAULT SPORT, TUC RAIL, ALSTOM, ADTRANZ.

Energy

EDF, ABB, ENEL, SCHNEIDER, GE ENERGY PRODUCT, ALSTOM, SEHV.

Mechanics

PICANOL, MAK, SOLAC, PONT-à-MOUSSON, GLAVERBEL, COMAU/RENAULT AUTOMATION.

Defence

AÉROSPATIALE MATRA MISSILE, FN NH, GIAT INDUSTRIES, CAP, DCN, DGA.

Research Centers

CENAERO, CNRS/IN2P3, CEA DAPNIA.

SAMTECH Headquarters
Parc Scientifique du Sart-Tilman
Rue des Chasseurs-Ardennais 8
B-4031 Angleur-Liège, Belgium
Tel. : +32 4 361 69 69
Fax : +32 4 361 69 80
http://www.samcef.com

SAMTECH France
Paris
14, avenue du Québec
Bât. K 2.1, SILIC 618
Villebon-sur-Yvette
F-91945 Courtaboeuf Cedex, France
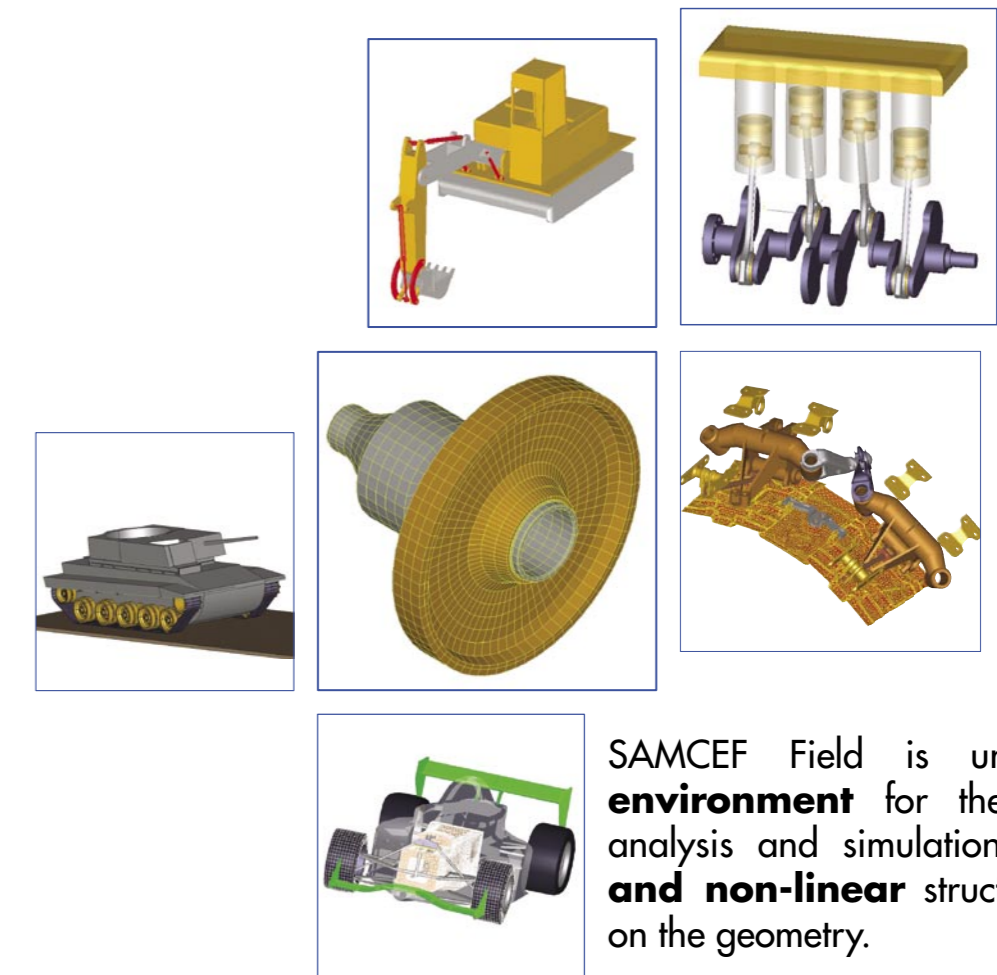Tel. : +33 1 69 59 22 80
Fax : +33 1 64 46 29 65

Toulouse
11, rue Marius Terce
F-31300 Toulouse, France
Tel. : +33 5 34 55 20 99
Fax : +33 5 34 55 15 00

SAMTECH Germany GmbH
Oskar-Kalbfell-Platz 8
D-72764 Reutlingen, Germany
Tel. : +49 7121 92 20 0
Fax : +49 7121 92 20 90

SAMTECH Italia
Via Guido d'Arezzo, 4
I-20145 Milano, Italy
Tel. : +39 02 48 559 407
Fax : +39 02 46 94 998

SAMCEF Field is unique **CAE environment** for the modeling, analysis and simulation of **linear and non-linear** structures, based on the geometry.

SAMCEF Field allows you to design your systems with a simple click. Tasks relating to the different process steps are managed through menus, icon bars, shortcuts or keyboard commands, all being contextual.

GENERAL CHARACTERISTICS

The geometry is either performed in the SAMCEF Field modeler or directly imported from most CAD software of the market. The user stays in the same environment from the design to the analyses of the results for the linear analyses as well as for non-linear dynamic analyses of complex mechatronic systems, thanks to its numerous data setting tools based on FEA method.

The SAMCEF Field integrated reports generation system allows you to make up very comprehensive analysis files in HTML format (modeling and analysis tasks summary, data and properties, photos, list of pre-selected values in Excel format…).

The SAMCEF Field documentation is directly accessible from the graphical interface. The help function offers two research levels depending on the requested information detail: tutorial and user manual.

Make analysis an integral part of your design process

USER INTERFACE AND GENERAL FEATURES

- Pull-down menus and toolbars;
- Contextual menus and dialog boxes;
- Navigational system structured in directories;
- Possibility to create "user" sub-directories for an optimized management of the model;
- Interactive graphical manipulation with different kinds of visualization;
- Model building history recorded in the navigational system;
- User preference management for units and symbols;
- Management of groups of homogeneous entities;
- Graphical treatment of results coming from SAMCEF and other solvers from the market;
- Automatic generation of a customizable HTML analysis report;
- Help and documentation on-line.
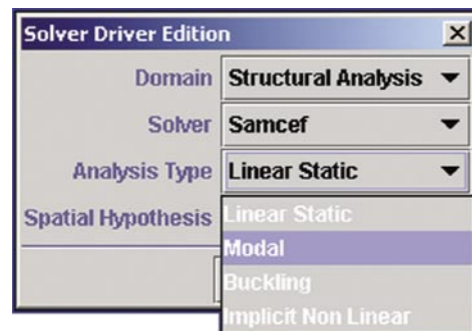
CREATION OF CAD GEOMETRIES

SAMCEF Field integrates simulation in the design process. It includes 2D and 3D design tools making it easy to create simple or complex parts in a very intuitive way, but also to rework existing mechanical systems starting from imported geometries.

SAMCEF Field provides all necessary tools for the design or the correction of imported geometries, but also tools dedicated to the creation of basic geometries (lineic, surfacic, volumic), a sketcher and tools for the assembling of existing geometries.

The complete history of your analysis is directly accessible via the navigational system. Easy-to-use and user-friendly, this feature allows you to re-use either all or parts of your geometries components or assemblies, or to introduce modifications, immediately passed on to your geometry.

To design or complete a CAD Geometry automatic correction tools
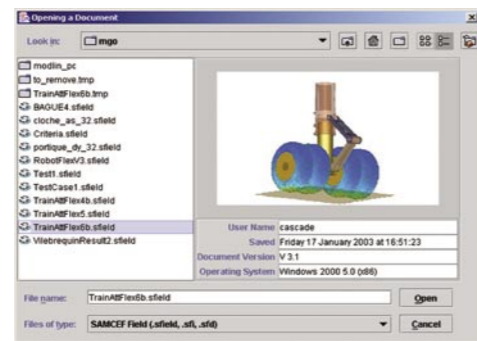
GEOMETRICAL MODELER

- Lineic, surfacic and volumic geometries hierarchical modeler;
- Sketcher to create 2D geometries under constraints;
- Geometry parameterization and visualization of entities concerned by the parameterization;
- Direct import of CATIA V5 & CATIA V4 models;
- Import and generation of standard formats STEP, BREP and IGES (import and export);
- Model analysis to verify the entity quality and validity;
- Geometry correction tools: faces sewing to create shells, filling holes in surfaces, treatment of micro-curves, design detail deletion;
- Boolean operations on entities;
- Assemblies of non-manifold geometries;
- Footprints;
- Numerous tools of measurement; angles, distances, coordinates…

DATA MANAGEMENT

This analysis process step lets you introduce and define all material and physical data of your mechanical system (material behavior, shell thicknesses, beam or rod properties…), as well as boundary conditions, loads and constraints between parts the system (i.e. local stiffness, springs, gaps).

Data definition uses the mechanical engineer vocabulary. SAMCEF Field enables you to use familiar tools of your trade and contributes to make the data input process much easier. A key feature is that you can thus focus on your problem and not on the software. When changes are made to the grid, all related items are easily updated. You can also assign data on the grid.

The Super Element management is a new feature of this version. You can very easily use and manipulate parts, which are pre-calculated Super Elements, in order to create or improve a standard geometry.

SAMCEF Field provides you with intuitive tools for data definition and pre-visualization, as well as import and use of existing data libraries.

Reliability and quality of your analysis data definition on the geometry
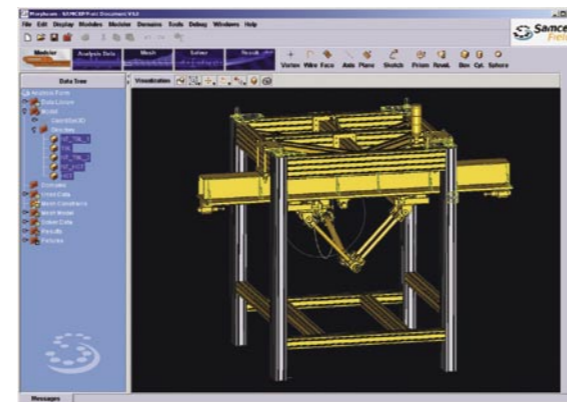
ANALYSIS DATA

- Assignment of the analysis data to the geometry or the F.E. mesh;
- Linear and non-linear isotropic and orthotropic materials (elastoplastic, hyperelastic…) for thermo-mechanical analysis;
- Composite layers of materials (plies, laminates, moduli curves display);
- Physical data for volume, shell, membrane, beam and rod elements;
- Flexible or Rigid behaviour;
- Mass & inertial moments in Rigid Behaviour;
- Boundary conditions and loads changing according to time step;
- Functions of one variable;
- Joints (fixed cylindric, prismatic, hinge, slider, sensor distance, etc.);
- Verification of the unit system coherence;
- Creation or import of data libraries;
- Super Element management;
- Automatic update after edition of the geometry.

MESHERS

Within SAMCEF Field, data are assigned when launching the solver. This characteristic is essential since it allows in particular to easily re-mesh the model during the process, while preserving a coherent data definition.

It is also possible to directly import a mesh either in SAMCEF, PATRAN or ANSYS formats.

The range of mesh fonctionalities in SAMCEF Field corresponds to the four meshing steps:

- First, definition of constraints on the mesh (number of elements on a line, element size…);
- Then, choice of the type of element (lineic, surfacic, volumic…) and the meshing type (setted mesh, by projection, by extrusion, free mesh…);
- Mesh generation;
- And finallly, verification of the obtained mesh according to quality criteria as well as of the possible mesh modification.

The free tetrahedral mesher (for volumic parts) can operate equally well on either a solid geometry or by filling in an existing skin mesh.

Among the new features available in this version, SAMCEF Field provides you with a tetrahedric mesher which can be used by extrusion, revolution or projection of a starting face. You can then obtain very regular volumic meshes. The check functions of a mesh allow to locate the areas where the mesh quality has been damaged. While editing elements, they can be corrected: i.e. by node fusion, by relocating nodes…

From fast simple simulations to most complex analyses
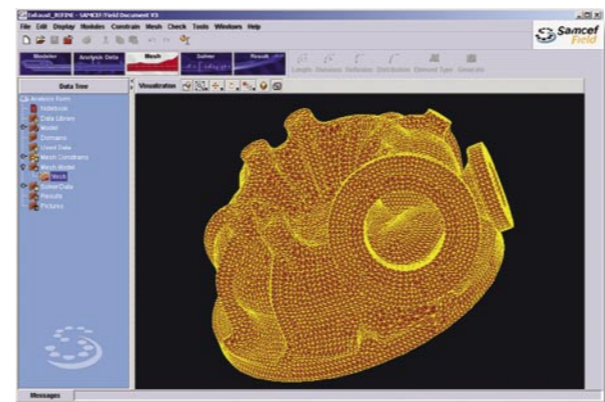
GRID

- Possibility to impose global or local mesh constraints;
- Lineic meshers for rod and beam models;
- Triangular and quadrangular surfacic meshers;
- Trans-patch meshing;
- Free tetrahedral mesher;
- Hexahedric and pentahedric meshers with prismatic solids;
- Rigid body mesher;
- Generation of cells with curved or straight edges;
- Sticking between different meshes;
- Graphic checking of mesh quality using topological criteria (user management);
- Detection of free edges and filing up of holes;
- Sewing of meshes: nodes close to each other are merged;
- Automatic correction of cells violating selected topological criteria;
- Interactive relocation of nodes and cells recombination;
- Mesh orientation visualization;
- Import of SAMCEF, NASTRAN and SDRC formatted meshes;
- Displaying of nodes or elements selected by their numbers;
- Selection of colors for the meshing objects as done for geometries.
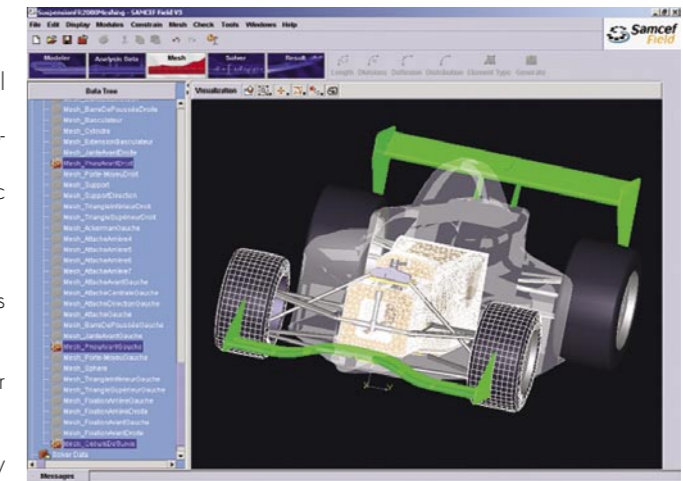
SOLVERS

Before launching the solver, and in order to avoid any waste of time, a data coherence checking is performed. This functionality verifies that the analysis is relevant and that elementary data are not missing. If data are missing, you are immediately informed.

A new feature is the possibility for the user to select the results to store for the post-processing. By default, you can access the main files generated by SAMCEF and organized in directories. You can also complete these available results by selecting complementary files to be run.

In order to exploit the whole power of integrated design, the access to SAMCEF family solvers are transparent.

SAMCEF Field only prompts you to define data that are relevant to the application field. It detects any incoherent information and informs you of possible corrective actions.

The monitoring interactive window allows to follow up the evolution of the calculation steps and interrupt if a problem is detected.

Integrated calculation in the design and simulation processus

CALCULATION

- Analysis data and meshing check before calculation;
- Choice of the result files to be saved;
- Specific parameters for non-linear SAMCEF analyses (they can be dynamically modified);
- Newmark and HHT time-step algorithm parameters;
- Linear solvers (linear analysis available in SAMCEF: linear static, buckling, modal extraction…);
- Non-linear solver (SAMCEF Mecano);
- External solvers.