ModelChecker™ for ANSYS® 5.7 Finite Element Analysis Model

Examination and Documentation Software

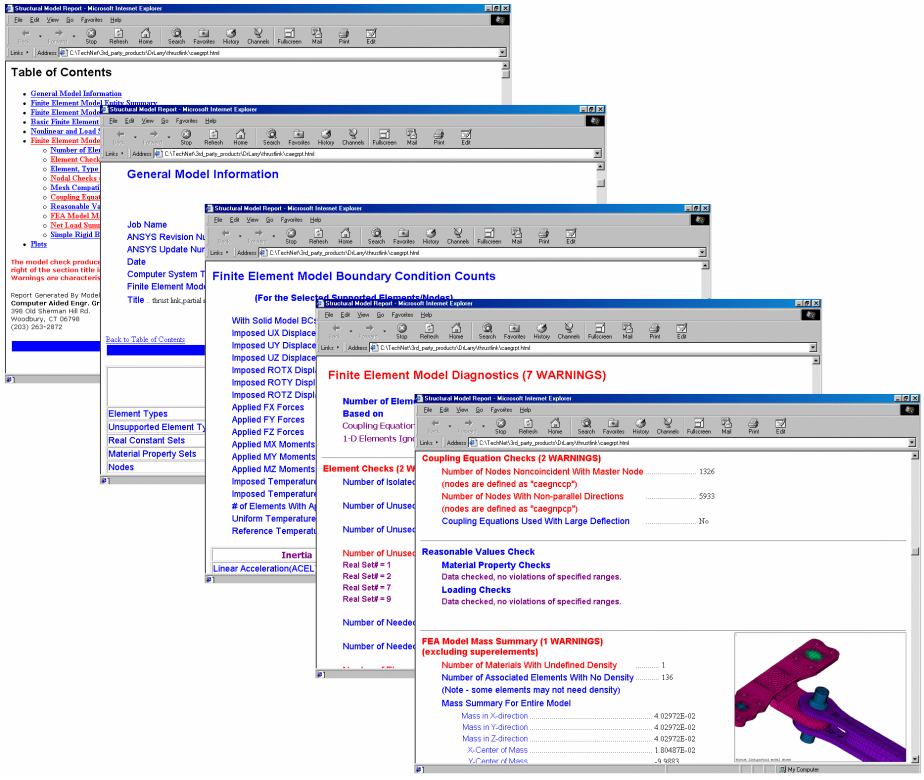
ModelChecker is a productivity tool providing detailed examination and documentation of any ANSYS 5.7 structural finite element model. Thermal, and other field models, can be subjected to most of the checks by switching the physics environment to structural.

Designed to identify problems in the model to minimize unnecessary solutions, ModelChecker helps reduce model-debugging time and eliminates analysis runs that may produce incorrect solutions.

ModelChecker is a collaborative tool that is a must for engineering organizations using ANSYS software. It is used for communication, review, and support of finite element models. As many companies distribute or outsource their computer-aided engineering simulation, ModelChecker can provide critical finite element analysis information to anyone in your organization -- anywhere in the world -- with Internet access.

Developed by Computer Aided Engineering Group, Inc. (CAEG), ModelChecker was written by the practicing finite element specialists of CAEG -- with over fifty years of experience providing ANSYS customer support, solving practical problems and providing ANSYS-specific and general finite element best practices education.

ModelChecker performs a comprehensive examination and documentation of your model attributes, boundary conditions, loads, and virtually all nonlinear and load step settings that exist. And, ModelChecker can examine these settings for appropriateness based on your own customized requirements.



ModelChecker *automatically produces an HTML report containing the complete model description and assessment, as well as multiple view plots of the solid model and/or the finite element model with, or without, boundary conditions.*

ModelChecker will be available when ANSYS 5.7 ships in December 2000.

What does ModelChecker *do?*

ModelChecker provides a menu-driven (or batch-processed), automatic, comprehensive examination and documentation of any ANSYS structural model before the model is solved:

- ✓ Thoroughly documents your model.
- ✓ Increases the reliability of your models.
- ✓ Decreases model debugging time/costs.
- ✓ Helps minimize solutions that may produce incorrect results.
- ✓ Ensures there are no modeling mistakes.
- ✓ Facilitates evaluating outsourced models.
- ✓ Exchanges FEA model information over the Internet in standard HTTP formats.
- ✓ Minimizes the support burden inexperienced users may place on expert users.
- ✓ Helps educate inexperienced users in the creation of correct FEA models.

Use ModelChecker*!*

You can use ModelChecker at any time during the modeling process. You may want to use it sequentially to provide information about the solid model, the element and attribute settings, the quality of the finite element mesh, and the applicability of the types of loadings and solver settings. In general, you will obtain the most benefit by using ModelChecker throughout your FEA model building process.

If you would like to learn more about ModelChecker *software, please send your email inquiry to:*

durocher@caeai.com
or call:
203-263-2872 (USA)

Developed by
Computer Aided Engineering Group, Inc.
398 Old Sherman Hill Road
Woodbury, CT 06798

COMPUTER AIDED ENGINEERING ASSOCIATES, INC.

Supported by
Computer Aided Engineering Associates, Inc.

CAEWORLD™
Distributed by CAEWORLD Corporation
www.CAEworld.com

© 2000 CAEG, Inc. All Rights Reserved. ModelChecker is a trademark of Computer Aided Engineering Group Inc.
ANSYS is a registered trademark of ANSYS, Inc. All other trademarks and registered trademarks are the property of their respective owners. Printed in USA.

ModelChecker™

Version 1.0 for ANSYS 5.7

Technical Specifications

- ✓ **Checks model statistics.** In addition to standard information, the software provides:
  - System date
  - ANSYS Revision #
  - ANSYS Update number
  - ANSYS agreement number
  - Title
  - Job name
  - System type
  - Analysis type
  - Equation solver
  - Detailed load and displacement BC summary for solid model and FEA model
  - Listing of element types used , # of elements associated with each type
  - List of material properties used with # of associated elements
  - # of super elements
  - # of contact elements
  - # of pretension elements
  - # of surface effect elements
  - # of shell elements
  - # of explicit elements
  - # of 1-D elements
  - # of lumped mass elements
  - # of p-elements
  - # of parts, solid model and FEA model
  - Model units
  - Model dimensionality (2-D, axisymmetric, harmonic, 3-D)
  - List of actual, active DOF in model (e.g., UX,UY,ROTZ), ANSYS listings are simply based on defined element types
  - Rotated nodal coordinate system flag
  - Nonlinear model flag
  - Sources of nonlinearity flagged such as plasticity or viscoplasticity, creep, hyperelasticity, large deformation, stress stiffening, nonlinear element types, contact element types
  - Conservative or non-conservative, (users manual discusses implications)
  - Current setting of nonlinear controls and options
- ✓ **Number of FEM parts** (based on FEM topology), options to include or exclude coupling and constraint equations and one-dimensional elements in determining FEM parts
- ✓ **Element checks**
  - Isolated elements (not connected to other elements)
  - Unused (count of number and list of unused numbers)
    - Element types, Material, Real sets
- ✓ **Element Checks** (continued)
  - Needed but undefined (count of number and list of needed numbers)
    - Materials
    - Real sets
  - element shape checks with concise summaries
  - reversed (inconsistent) shell normals
- ✓ **Nodal checks**
  - Unused nodes
  - DOF mismatch (elements with different DOF sharing common node)
  - Applied forces/displacements that will be ignored since model does not have the corresponding DOF
- ✓ **Mesh compatibility** (cracks), including mid-side nodes shared with corner nodes
- ✓ **Temperature checks** – wide variety of node and element temperature checks, reference value(s), temp-dependent properties, coefficient of thermal expansion, etc. Portions of model where temperatures are undefined and hence TUNIF will be used.
- ✓ **Coupling equations checks** (non-coincident nodes, non-parallel directions)
- ✓ **Constraint equation check** (simulate uniform temperature rise), look for equations that inadvertently induce stress
- ✓ **Constraint and/or coupling sets** used in large deformation analysis
- ✓ **CPU-intensive options**(require element mass formulation)
  - Net load summary
  - Net load and moment in each direction. Overall model, or part by part, broken down by:
    - Applied nodal forces/moments
    - Inertia loads
    - Pressures
  - FEM mass summary, over-all model or part by part.
- ✓ **Preliminary rigid-body motion check** (types and directions)
  - Entire model or part by part
- ✓ **Element/load consistency checks**
  - Mixtures of axisymmetric and nonaxisymmetric elements
  - Mixtures of harmonic and nonharmonic elements
  - Use of harmonic elements in nonlinear analysis
  - Axisymmetric elements subjected to inertia loads(spin, G-loads) in inappropriate directions(e.g., spin about the x-axis)
  - Harmonic elements subjected to inertia loads in inappropriate directions(e.g., spin about the y-axis for MODE >0)
- ✓ **Nonlinear summary** and consistency checks
- ✓ **Solid Model Checks**
  - Simple model statistics
  - # of parts (based on solid model topology); volume parts, area parts, line parts
  - Small/large line, area, volume summary
  - Area/volume/mass and center of mass location
  - All of the above (default)

ModelChecker gives the user the option to include multiple view plots of the solid model and/or the finite element model with, or without, boundary conditions, in the HTML report.

If the model produces a warning in any check, the number of entities producing such a warning is noted and the entities are grouped into components for later review.

For companies that want to provide their users with a customized report and additional checking, the software provides the following options:

- ✓ The information gathered by *ModelChecker* can be stored in the ANSYS database. This allows companies to create company-specific reports via the use of the data, which is fully documented and easily accessible via the ANSYS APDL language or a formatted external file.

- ✓ The company or user can provide a file (or input values) containing company- specific "reasonable" values of the following:
  - Young's Modulus
  - Poisson's ratio
  - Density
  - Coefficient of thermal expansion
  - Angular velocity
  - Linear acceleration
  - Temperature

  The program can then check the database for reasonable values and report quantities "out of range". This will catch many errors associated with typos or incorrect units.

Hardware platforms supported:

ModelChecker appears in the ANSYS graphical user interface as a Main Menu selection and runs on all the ANSYS supported hardware platforms.