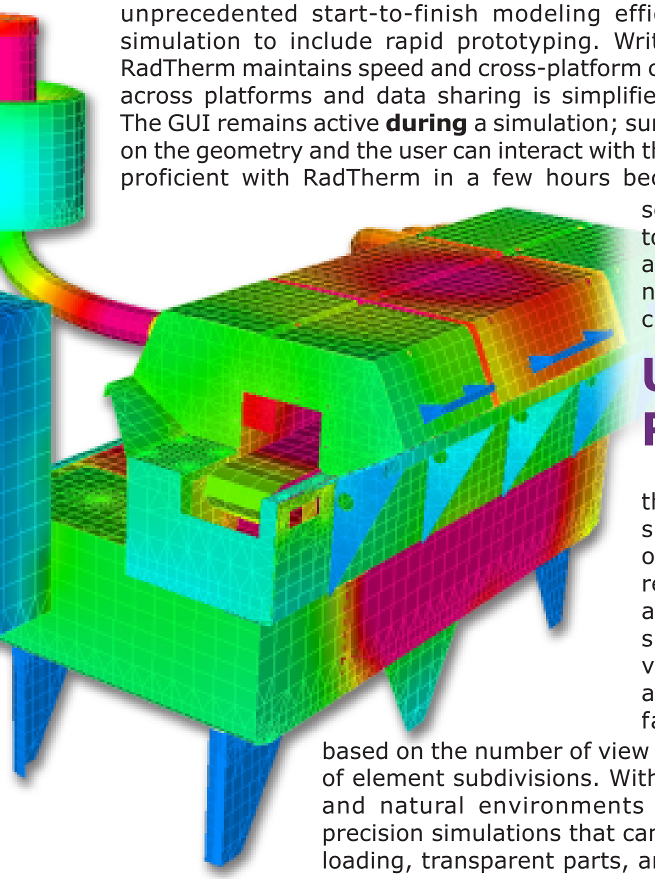
RadTherm™ Software

Advanced Thermal Modeling

RadTherm is full-featured, cross-platform, thermal analysis software for advanced CAE applications. RadTherm employs a state-of-the-art radiation module and an extremely user friendly Graphical User Interface (GUI) for simulating multimode heat transfer (radiation, conduction, and convection) and one-dimensional fluid flow.

Integrated preprocessing, solution, and post-processing capabilities offer unprecedented start-to-finish modeling efficiency, expanding the role of simulation to include rapid prototyping. Written entirely in portable C++, RadTherm maintains speed and cross-platform compatibility: the GUI is identical across platforms and data sharing is simplified with single-file data storage. The GUI remains active **during** a simulation; surface temperatures are displayed on the geometry and the user can interact with the model. Users report becoming proficient with RadTherm in a few hours because the GUI is spatially and sequentially arranged according to engineering intuition, with tabs and icons providing transparent navigation and a short learning curve.



Unlimited Resolution

RadTherm can solve thermal models of virtually any size - the model size is limited only by available system resources. Radiation, conduction and convection are solved simultaneously after radiation view factors are generated using a voxel-based ray tracer. The view factor accuracy is adjustable, based on the number of view factor rays cast and the number of element subdivisions. With an array of convection options and natural environments available, RadTherm delivers precision simulations that can account for the effects of solar loading, transparent parts, and advective fluid flow.

Extreme Accuracy

The accuracy of RadTherm's predictions has been independently verified against theoretical solutions and 3rd party measurements on systems operating in natural and controlled environments. Because of RadTherm's unlimited model resolution, mesh densities can be raised higher than ever before, ensuring that the simulation converges to the governing equations both spatially and temporally. RadTherm software is available on CD or via download, and is backed by ThermoAnalytics' 30-day unconditional money back guarantee.

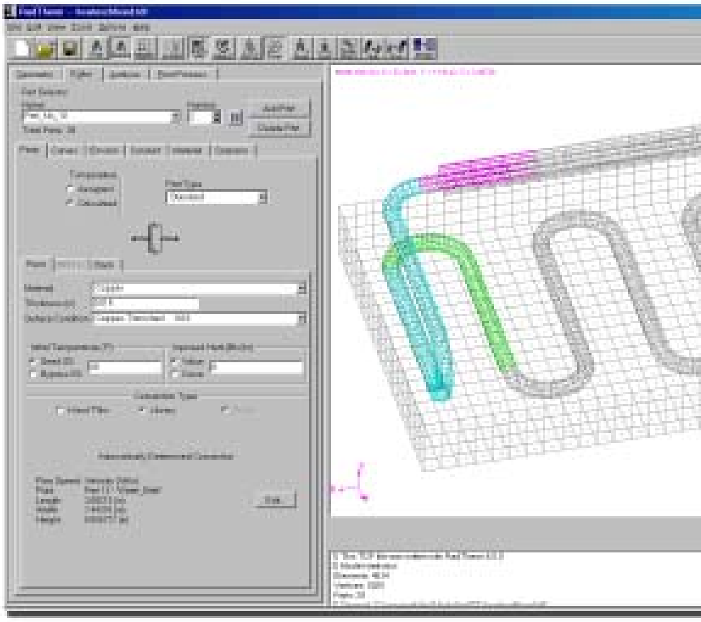
Efficient Model Setup

Geometry

RadTherm software requires only a surface mesh of user geometry, and although RadTherm comes equipped with the ability to create geometry, most users import meshes from common data formats, including:

Patran Neutral (ntl)	Wavefront (obj)
Nastran (nas)	Digital Elevation Maps (asc)
Stereolithography (stl)	Pro-E (Plug-in)
Autocad (dxf)	IGES (via Eclectic)
Rhino3D (3dm)	STEP (via Eclectic)

ThermoAnalytics is a developmental partner with many CAD, CAE, and CFD software companies, and continues to expand RadTherm's data exchange functionality to maximize end user productivity.



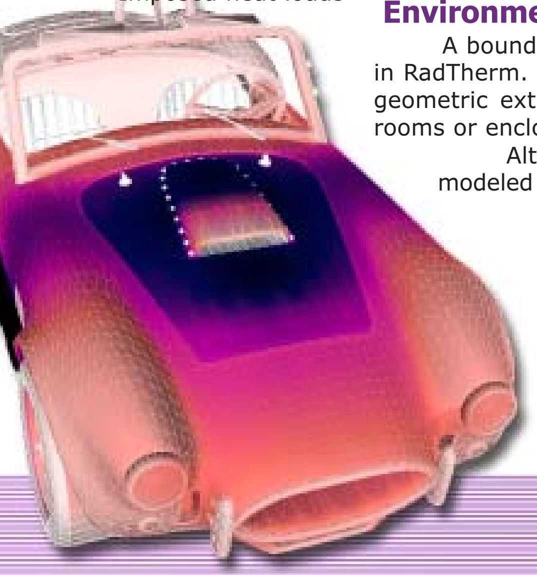
RadTherm has basic geometry creation and editing tools, such as copy, translate, etc. The user can quickly modify designs, e.g. shield position, without returning to the original CAD file.

Boundary Conditions

RadTherm supports several types of parts, which can accept various boundary conditions. Assigned temperature parts can have a uniform temperature on each side (can be different) or they can be assigned point temperatures (from thermocouple data, including transient data), and RadTherm will spatially interpolate the temperatures of the remaining elements. Curves for a wide range of parameters can be easily created in the GUI or imported.

All geometry parts are based on the shell mesh, so the thickness of each layer is assigned in the user interface. 3-layer part types allow the use of single-layer shell geometry to model composites or double-wall heat shields. The middle layer can be vacuum, air, solid, or mixed air/solid. The user specifies additional boundary conditions, including:

- Material properties, including anisotropic conduction
- Imported or specified convection
- Fluid advection links
- Imposed heat loads



Environmental Boundary Conditions

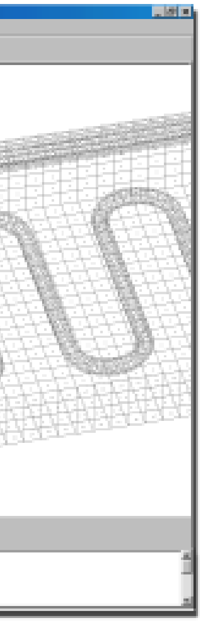
A bounding cube environment is the default in RadTherm. Wall temperatures and offsets from geometric extremities can be varied to simulate rooms or enclosures.

Alternately, natural environments are modeled as a terrain with sophisticated foliage and moisture settings and a hemispherical sky with built-in solar load. Terrains are modeled according to first principles, including core temperature, moisture content, etc. Comprehensive environmental data can be utilized to capture thermal effects of natural environments, such as rate of rainfall, wind vector, sky temperature, and solar radiation (direct & diffuse).

Solar band multi-bounce radiation calculations are performed to capture effects of part-to-part and part-to-terrain reflectance. Additionally, glass part types include solar transmissivity of direct and reflected solar radiation, independent of thermal-band radiation calculations. Radiation view factors to the environment, geometry, and solar (apparent) areas are automatically calculated using RadTherm's voxel-based ray tracer.

Model Output

Model Interaction During Solution



RadTherm displays both numerical and graphical results to the user during the simulation,. During the solution, the graphics window must be updated to reflect updated results at regular intervals (counted in terms of iterations). The graphics update frequency setting allows the user to update the displayed temperatures frequently, if the solution will be closely monitored, or infrequently (variable up to once in 200 iterations) for faster completion of the simulation.

Built-in Post-Processing

Viewing, animating, plotting and exporting results are all available in RadTherm's built-in post-processor. The front and back surface temperatures are displayed on the geometry by color (as they are during the simulation) and the geometry can be displayed as shaded surfaces, wireframes, or points. Individual elements can be selected in the graphics window and detailed data for that element is automatically displayed: node temperatures, flux rates for each mode of heat transfer, boundary conditions, and environmental parameters, e.g. solar radiation. The data can be displayed as flux or absolute units. Transient simulations can be animated, paused, and results at each time step reviewed.

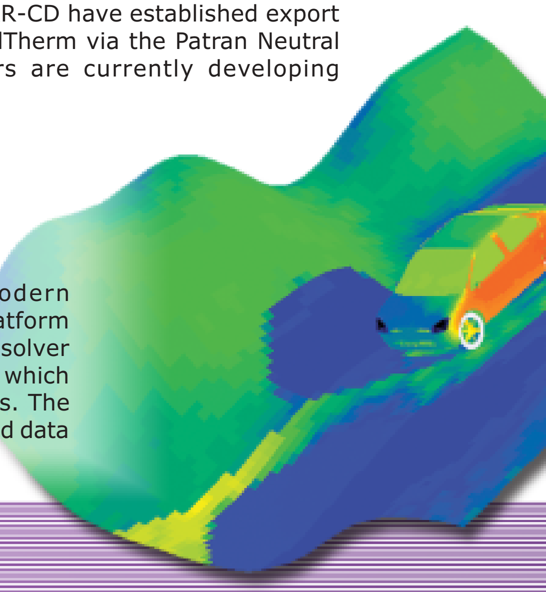
Results for parts are averaged from the part elements, and both part and element-level results can be plotted or exported. The plot window retains data from previous runs, so that results from sequential runs can be plotted together for comparison.

Data Exchange with other Software

In addition to exporting thermal results, RadTherm's single-file data can be easily converted to multi-file collections for loading into the popular EnSight™ data visualization software. Export and conversion routines are in development for other data and post-processing software.

CFD Import

RadTherm's speed and accuracy allows thermal systems with complex radiation to be solved much faster than by CFD alone: run times are often shortened by orders of magnitude. For maximal accuracy, CFD convection results can be imported with surface geometry into RadTherm and part surface temperature results can be exported from RadTherm back to CFD codes as boundary conditions. Fluent and STAR-CD have established export functions to convey this data to RadTherm via the Patran Neutral file format. Other CFD developers are currently developing automated export functions.



Solver Characteristics

Software Structure

RadTherm is written in modern object-oriented C++ with cross-platform functionality and file exchange. The solver employs an optimized architecture in which sub-modules exist as library routines. The library routines operate on well-defined data structures with documented inputs, outputs, and side effects.

RadTherm includes a C++ API, called the TDF Library. The TDF Library is cross-platform compatible and can be used to export results to other programs, create models, setup boundary conditions, and directly interface with other programs.

RadTherm also supports running in batch mode, which will automatically calculate view factors and solve thermal models without any needed interaction from the user interface.

Solution Initialization

Part initial temperatures are set as a boundary condition during model setup. Alternatively, RadTherm can assign initial element temperatures to the results from the previous run (self-seeding) or from a different model file using the same geometry. Self-seeding yields very rapid convergence of new steady state models where only minor changes to boundary conditions have been made. RadTherm can restart transient solutions to continue a run further in time (using the same or different boundary conditions).

Solution Convergence

Two different methods (tolerance and tolerance slope) are available for determining when the iterative solution has converged. Both methods make use of the solution tolerance, which is defined to be the maximum change in any node temperature between two iterations. The tolerance method specifies that a solution is converged when the solution tolerance is less than the specified tolerance value.

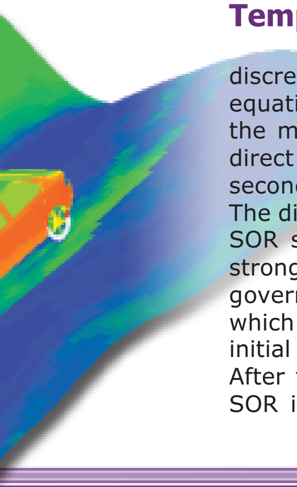
The tolerance slope method keeps a record of the solution tolerance for each iteration. A line is fitted to the last portion of the tolerance data using a linear least squares curve fit. The tolerance slope method specifies that the solution is converged when the absolute value of the slope of the fitted line is less than the specified tolerance slope value for 2 consecutive iterations.

Numerical Method

RadTherm solves the energy balance equation simultaneously for convection, conduction, and radiation. All temperature and heat load data are input as constants or as functions of time at user discretion. RadTherm utilizes the time-averaging Crank-Nicholson implicit finite difference scheme to discretize the governing equations. The Crank-Nicholson method is used because it is one of the most versatile numerical methods available for parabolic differential equations. The method is unconditionally stable and is second order accurate in time and space.

Adaptive Relaxation

RadTherm's main iterative solution method is Successive Over-Relaxation (SOR). The SOR method uses a relaxation parameter which can be adjusted to accelerate convergence. Since the optimal relaxation parameter is different for each model and is difficult to determine, it can be a burden for the user to have to set this parameter. RadTherm provides the adaptive relaxation technique which monitors the solution process and adjusts the relaxation parameter as necessary. In the rare cases where the adaptive relaxation technique prevents convergence, it can be disabled and a fixed relaxation value can be specified.



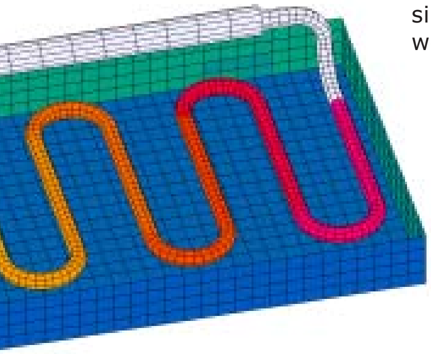
Temperature Distribution

The temperature distribution is determined by applying the discretized governing equation to each node. The resulting system of equations is solved using a partially direct solution method in which the main iterative method is used in conjunction with a secondary direct method. The main iterative method is the SOR method and the secondary direct method is Gauss elimination with partial pivoting. The direct solution is invoked at the beginning of each iteration of the SOR solution as a preconditioner for nodes that are known to be strongly dependent upon one another. The direct solution solves the governing equations of two or more of these nodes simultaneously, which allows the node temperatures to converge quickly from the initial values to the temperatures that satisfy the governing equations. After the direct solution is complete, the solution proceeds with an SOR iteration, in which the governing equations are evaluated for each node. The iterations continue until all node temperatures have converged. Then the solution advances to the next time step and repeats the above procedure.

Radiation Exchange

RadTherm uses the net-radiation equation for computing radiation exchange between diffuse-gray surfaces in an enclosure. The net-radiation equation is based on node emissivity, surface area, the Stephan-Boltzmann constant, the view factor from node k to node j, and the temperatures of nodes k and j. This equation is combined with the energy equation prior to discretization using the Crank-Nicholson numerical method. Since the net-radiation solution is integrated into the thermal and radiometric solutions, RadTherm can account for time or temperature varying surface conditions.

Solar Radiation View Factors



RadTherm includes a built-in solar and sky model for rapidly setting up natural environment simulations. RadTherm can also use detailed weather data for more accurate simulations. When using a natural environment, the solar azimuth and zenith are automatically updated at each time step, and then the solar apparent areas are recalculated with RadTherm's voxel-based ray tracer.

Radiation Patching

Patching accelerates the solution of large models by averaging the view factors of the elements from the patches and solving for radiation on a patch-by-patch basis, rather than on an element-by-element basis. For an average model, patching cuts solution run time by 50% and substantially reduces view-factor file size.

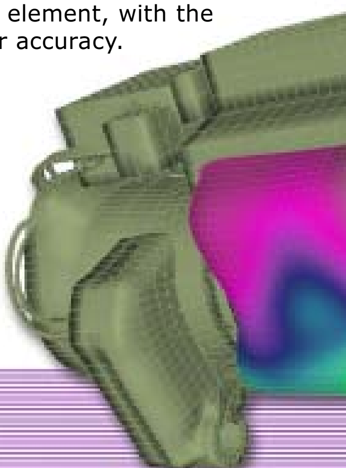
RadTherm features an automatic patch generator which groups elements based on the following criteria, specified by the user: maximum angular deviation of any elements in the patch, maximum aspect ratio of the patch, the maximum number of elements in a patch, and a check to insure that all elements in a patch are assigned to the same part.

Ray Tracing

ThermoAnalytics has pioneered the use of a voxel-based ray tracing algorithm which computes both radiation view factors and solar projected (apparent) areas. The voxel method is exceptionally fast and accurate, and is done within the software for both speed and cross platform compatibility. This technique, in which the bounding box of the entire scene to be ray-traced is divided into cubic volume elements (or voxels), is also known as spatial subdivision. The ray-tracer uses uniform spatial subdivision, in which all voxels are the same size. Before ray-tracing, the geometry is preprocessed to create a list of all elements that occupy each voxel. As each ray is cast, the voxels are visited in the sequence in which the ray passes through them, and only those faces that occupy these voxels are tested for intersection.

A unit hemisphere is placed over the centroid of each element surface. The unit hemisphere is then uniformly subdivided based on the number of rays to be cast. This subdivisions are a function of the angle, so the density of rays cast will be highest at the element normal. A variable number of rays are thus cast from the centroid of each side of each geometry element, with the number of rays controlled by the user-assigned view factor accuracy.

View Factor Accuracy



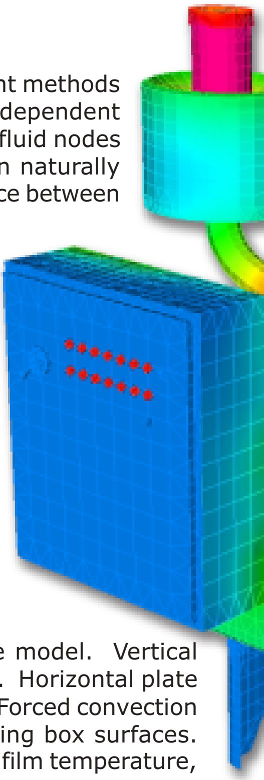
By adjusting the number of rays cast from each element, the speed and accuracy of the view factor calculation is determined. RadTherm also allows the elements to be subdivided and rays cast from each element subdivision centroid. The combination of number of rays and element subdivisions thus determine the overall view factor accuracy and calculation time, with minimum settings of 512 rays and maximum of approximately 250,000 rays from each side of each element.

This range of settings provides the user with the ability to perform rapid view factor calculations early in the design process when the product geometry is in development. Later in the design process, the view factors can be recalculated at a higher density to provide a more accurate radiation solution.

RadTherm improves model analysis speed for multiple iterations or parametric analysis by reusing view factors. On subsequent analysis of the same model, RadTherm performs a view factor validation and then begins the thermal analysis without recalculating the view factors. Reusing the view factors from a model can significantly improve run time and allow for increased analysis of a model.

Conduction

RadTherm computes both planar conduction between elements and conduction through the thickness of elements. The thickness is independent of the model geometry, as RadTherm requires only a surface mesh. The user specifies the material thicknesses and thermal conductivities. Planar conduction linkages are created by examining shared element edges and computing conductors between adjacent elements based on the thickness of each element, the thermal conductivity of each element, and the distance between each element centroid and the midpoint of the shared edge. Conduction through an element (from the front surface node to the back surface node) is based on the element surface area, thickness, and thermal conductivity.

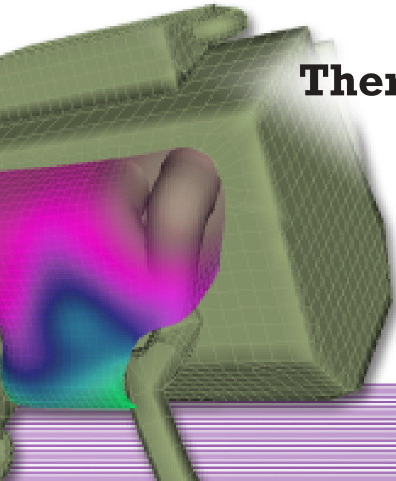


Convection

RadTherm allows the user to choose from several different methods of convection calculation on each surface: a fixed or time-dependent convection coefficient, a convection library option for use with fluid nodes and automatically calculated convection coefficients based on naturally induced fluid motion (flows induced by the temperature difference between the fluid and surface). Natural convection can be combined with forced convection based on fluid flow values or curves. For the highest level of convection accuracy, RadTherm can import node-level convection and fluid temperature data from CFD codes as described above. Where natural environments play a role in the analysis, RadTherm can utilize weather data for wind speed, direction, solar, and other data, and automatically calculate convection values based on the planar orientation of each element.

Wind Convection

The wind convection algorithm accounts for wind speed, wind direction, element orientation, laminar flow, and turbulent flow. Both natural (free) and forced convection is considered in the determination of the convective heat transfer due to wind. The characteristic lengths used in wind convection are derived from the overall bounding box of the model. Vertical plate natural convection uses the height (z-axis) of the model. Horizontal plate natural convection uses the top area divided by the perimeter. Forced convection uses the square root of the largest area found on the bounding box surfaces. All fluid properties used in wind convection are evaluated at the film temperature, which is defined to be the average of the surface and fluid temperatures.



ThermoAnalytics, Inc.

23440 Airpark Blvd.
PO Box 66
Calumet, MI 49913
USA
info@ThermoAnalytics.com
http://www.ThermoAnalytics.com

Tel. 800-214-3766
Tel. 906-482-9560
Fax 906-482-9755