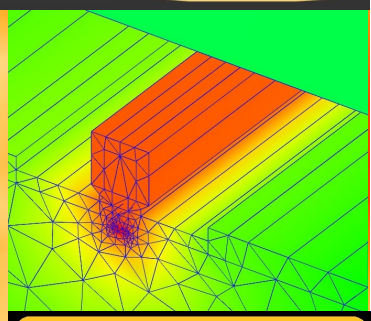
SOLARIA THERMAL

Solaria Thermal is a system designed to computer simulate nearly any thermal problem



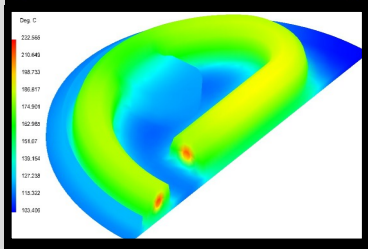
Problems are simulated geometrically using 2D Plate and 3D Brick and Tetrahedron elements. **Solaria** can accurately represent skewed elements.

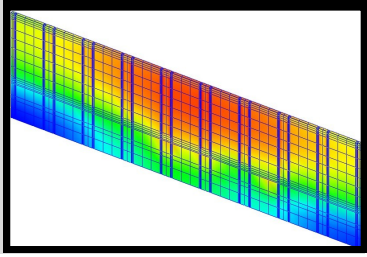
Solaria uses a Finite Element method to convert geometry into an accurate resistor/capacitor representation which is then accurately and quickly solved using a finite difference method. This combination gives accurate results like other Finite Element (FEA) tools like NASTRAN and ANSYS but solves in a fraction of the time. It also provides you with unsurpassed flexibility in model generation, not possible with a FEA solution.

Model generation, solving and post processing in a single easy to use environment. Low cost with all the features needed by the professional.

EASY TO USE

- Complete, no other tools required
- Complete model generation, solving and post processing in one environment
- Built-in steady state and transient solvers
- Model size only limited by available memory



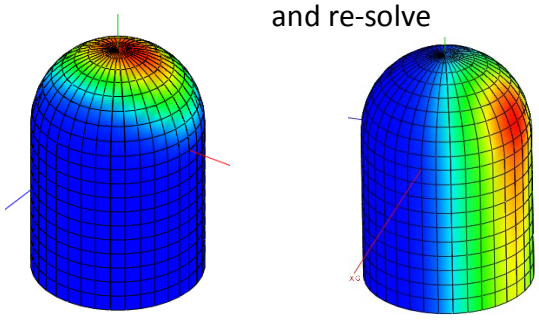


Elements supported

- Geometric – 2D Plates, 3D Bricks and Tetrahedrons
- Heat loads – Nodal, Surface and Volumetric
- Radiation – Node-to-Node and Node-to-Surface, Surface-to-Surface using the interface with TRASYS
- Flow, Convection, Interface, Nodal Capacitance, Resistors, Thermostats, Boundary nodes
- IntelliMesh allows you to thermally attached different parts of the model that are meshed very differently
- Most everything can be temperature and time dependent
- Complete Sinda/G and Sinda/FLUINT files exported
- Imports and exports TAS neutral file
- Built-in 2D automesher
- Super fast finite difference and conjugate gradient steady state and transient solvers

Just move the sun

and re-solve



Go to www.SolariaThermal.com for more information and videos

SOLARIA THERMAL

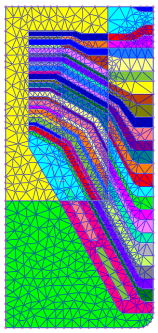
843.564.1229

info@solariathermal.com

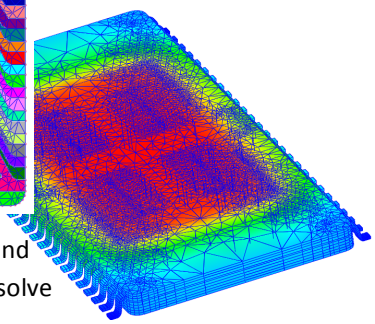
AutoMesher

Import geometry DXF file

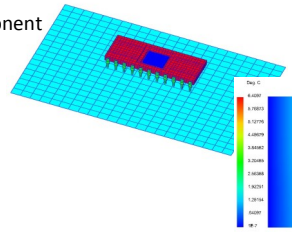
AutoMesh



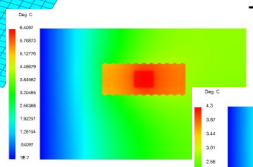
Extrude, define materials and boundary conditions then solve



Complex electronic component on a simple PCB

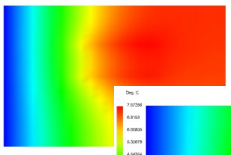


Resultant temperatures, top view, PCB temperatures

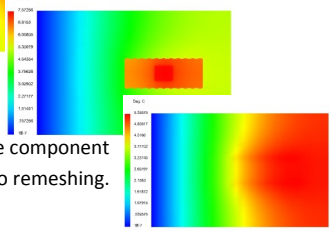


IntelliMesh

Thermally connect dis-similar meshes



Just move the component then solve. No remeshing.



Capabilities

- Steady state and transient analysis
- Two dimensional plate elements.
  - Elements can be four or three sided
  - Thermal conductivity, thickness, density and specific heat is defined in the property table
  - Anisotropic thermal conductivity, (different in X and Y directions based on user defined coordinate system)
  - Thermal conductivity, density and specific heat can be temperature, temperature difference, time and time cyclic dependent
- Three dimensional brick elements
  - Elements can be six or five sided
  - Thermal conductivity, density and specific heat is defined in the property table
  - Properties can be temperature, temperature difference, time and time cyclic dependent
  - Anisotropic thermal conductivity, (different in the X,Y and Z directions based on user defined coordinate system)
- Tetrahedral elements
- IntelliMesh thermally attaches parts of the model that are meshed differently
- Variables can be defined that can be referenced by any item in the property table.
- Three types of radiation elements can be generated, node-to-node, node-to-surface and surface-to-surface.
- Convection elements can be temperature, temperature difference, time or time cyclic dependent.
- Scalar resistor elements can reference a property and be a function of geometry and can add capacitance to the model
- Fluid flow elements can be generated to account for the fluid temperature rise and fall.
- Nodal, surface and volumetric heat loads can be fixed or temperature, temperature difference, time or time cyclic dependent
- Thermostats can be on/Off and proportional type.
- Boundary or fixed temperature nodes can be time or time cyclic dependent.
- Documentation on the model can be saved with the model.
- Any consistent units can be used.
- The model is displayed using colors to represent element type, properties and temperatures.
- Groups of elements can be defined to built complex models.
- Hidden line and property plots can be viewed on the screen.
- Element information can be queried and displayed.
- Solaria can export ready to run SINDA/G and SINDA/FLUINT models.
- Solaria contains a complete interface with FEMAP
- MSC/NASTRAN and ANSYS files can be imported
- Solaria has a complete interface with TRASYS.

843.564.1229
info@solariathermal.com