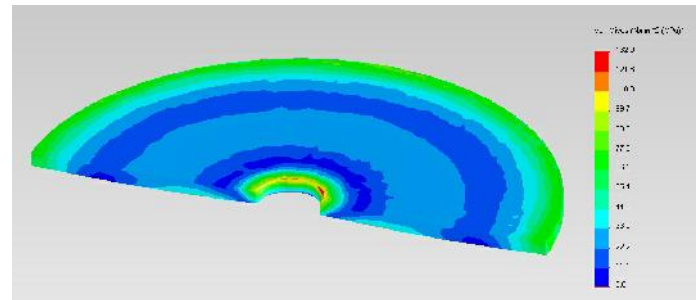




# Thermal Stress Component Analysis

## Description

A number of thermal stress analyses were required on semi conductor processing equipment, to establish thermal distribution. The aim was to establish the maximum operating temperature of the equipment and its distribution, taking into account conduction, convection and radiation. The results were then imported into a linear stress FE analysis and combined with various pressure loads, to determine induced stress levels, maximum deflections and fatigue life for a number of different load cases. The geometry was imported into a 3D Cad system using client geometry and analysed using the 3D "solid" elements.

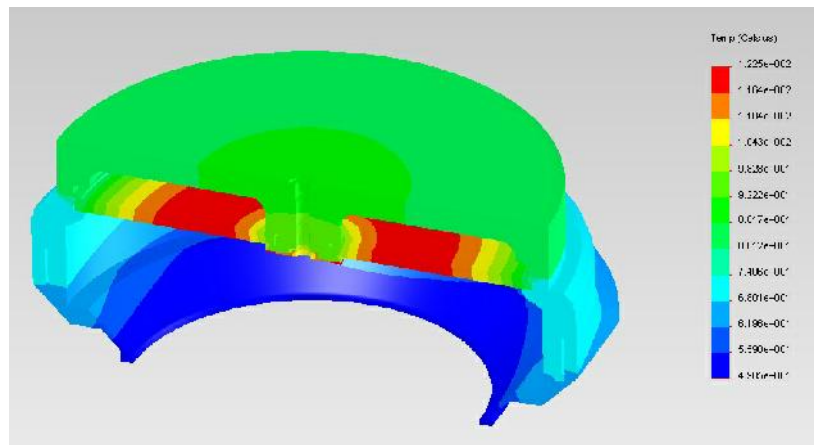


*FE Model Stress Distributions*

The component was mounted using fixed position restraints to reproduce the correct mounting configuration and stiffness, with bonded and sliding contacts used to model component interfaces, as appropriate. Thermal resistances were also applied, to model seal to housing interfaces. Varying pressure and thermal loads were applied, along with calculated heat transfer coefficients and material emissivity.

## Specification Summary

- Main structure materials - aluminium, alloy and ceramic
- 3D solid finite element models
- Thermal and pressure stress analysis
- Max stress = 133 MPa
- Max temp = 140 deg C
- Max pressure = 1 bar



*FE Model Temperature Distribution*

## Disciplines Used

- Heat transfer analysis, steady state & transient FE thermal analysis, linear FE stress analysis and fatigue life analysis.