Genberg, V. “Thermal and Thermoelastic Analysis of Optics”
Optomechanical Engineering Handbook
Ed. Anees Ahmad
Boca Raton: CRC Press LLC, 1999

Thermal and Thermoelastic Analysis of Optics

Victor Genberg

Notations

1D = 1 dimensional
2D = 2 dimensional
3D = 3 dimensional
BC = boundary conditions
CFD = computational fluid dynamics
CTE = coefficient of thermal expansion
DOF = degrees of freedom
FD = finite difference
FE = finite element
FEA = finite element analysis

9.1 Introduction

The goal of most thermal analyses of optics is to provide temperature profiles for subsequent thermoelastic analyses which in turn provide surface distortions for optical performance predictions. The flow of data for a typical design is shown in Figure 9.1. There have been many problems involving the interaction and data flow between the steps in Figure 9.1. This chapter will address some of those issues. Other applications of thermal analysis include the design of thermal control systems and the analysis of temperature-dependent properties such as the index of refraction. Analogies for adhesive curing and hygroscopicity are discussed in the final section.

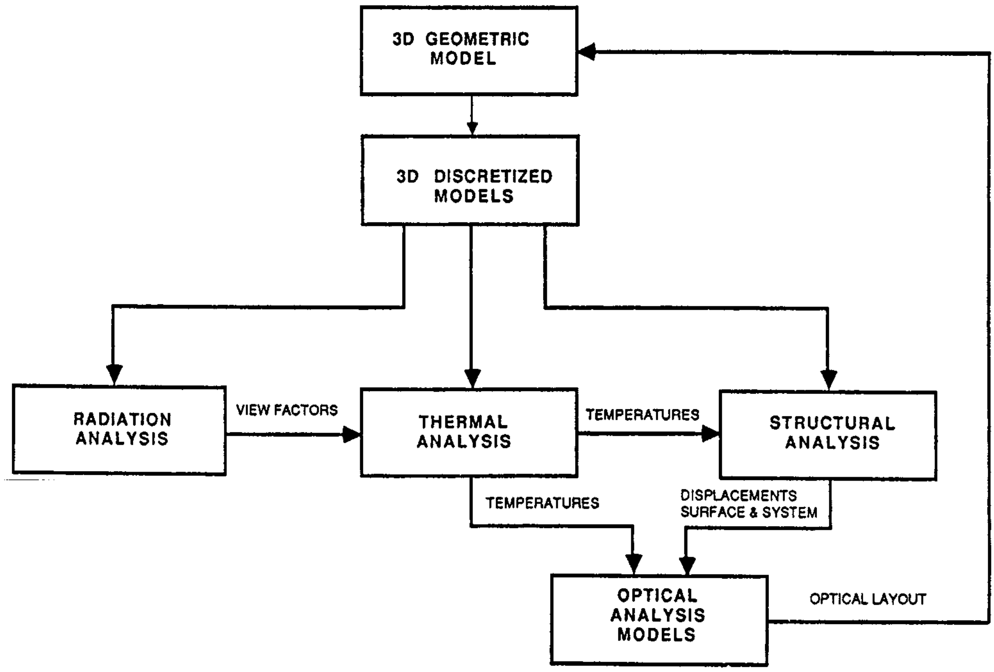


FIGURE 9.1 Data flow in optomechanical design.

9.2 Heat Transfer Analysis

Modes of Heat Transfer

In the analysis of precision optical systems, conduction, convection, and radiation must all be considered. Since small temperature gradients may be important in the resulting optical performance, the analyst should check for all three modes of heat transfer.

The Fourier heat conduction equation in 1D is

$$q = -k(dT/dx)$$

where q = heat flux
k = thermal conductivity
dT/dx = spatial thermal gradient

For an optical structure, conduction must be considered as the most important factor in the determination of gradients in the optic and the support structure. The conductivity of most materials is fairly constant over a broad range of temperatures. The heat balance equation for conduction in a homogeneous and isotropic material is

$$k\,\Delta T + Q = \rho c(dT/dt)$$

where Q = volume heat generation
ρc = heat capacity/unit volume
dT/dt = change of temperature with time

Other terms may be added to the energy balance equation. For convection,

$$q = -hA(T - T_a)$$

where h = convection film coefficient
A = surface area
T_a = ambient temperature

Convection is a surface effect requiring a fluid such as air. The convection coefficient is difficult to predict and is temperature dependent. For some applications, the net heat flow due to convection is small so an approximate value of h is sufficient.

Radiation is another surface effect:

$$q_{ij} = \sigma F_{ij}\varepsilon\left(T_i^4 - T_j^4\right)$$

where q_{ij} = heat flow surface i to surface j
σ = Stefan-Boltzman constant
ε = emissivity
F_{ij} = radiation view factor from surface i to surface j
T_i = temperature of surface i in absolute units

Note that the radiation term is to the fourth power, requiring nonlinear solution algorithms. Radiation is especially important in spaceborne applications since there is no air present for convection, and the view to deep space is at absolute zero. Unfortunately, the calculation of view factors is very expensive in computer resources and will be addressed in a later section.

Conduction

For most real problems, a numerical solution of the conduction equation is required. Most heat transfer books teach the use of a finite difference approach to heat conduction as in Krieth.[5] However, finite difference lacks the advantage of finite elements in analyzing complex geometries. Almost every finite element text addresses both structural and thermal derivations of finite elements as in Hubner[4] or Segerlind.[6]

The finite difference (FD) approach is very straightforward for rectangular geometries with a regular mesh pattern. The FD approach can also be used on optics with a polar mesh pattern effectively. However, when additional detail around mounts in required, or nonregular geometry is required as in most real real optical systems, then the FD approach has difficulty.

A simple example will show the advantage of using finite elements in real geometries requiring irregular geometries. In the simple problem of Figure 9.2, the resulting thermal contours should be uniformly spaced horizontal lines, because the heat flow is one dimensional. Using an irregular mesh required by most real problems, the finite difference mesh provides erroneous answers, whereas the finite element mesh provides the correct results. The conventional finite difference

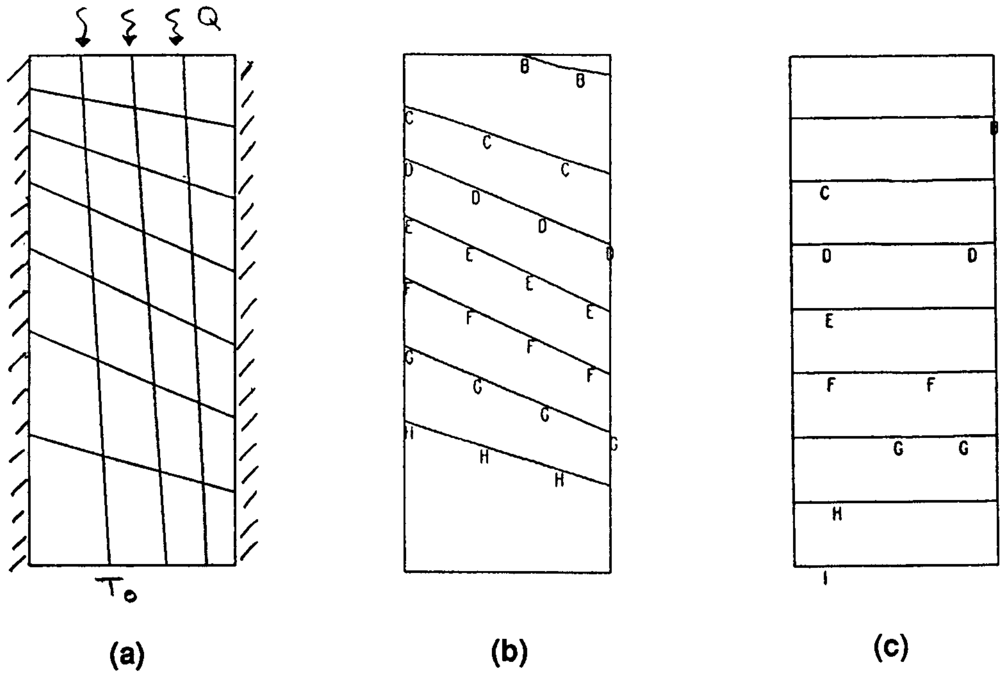


FIGURE 9.2 Simple heat conduction example. (a) Skewed mesh pattern; (b) finite difference results; and (c) finite element results.

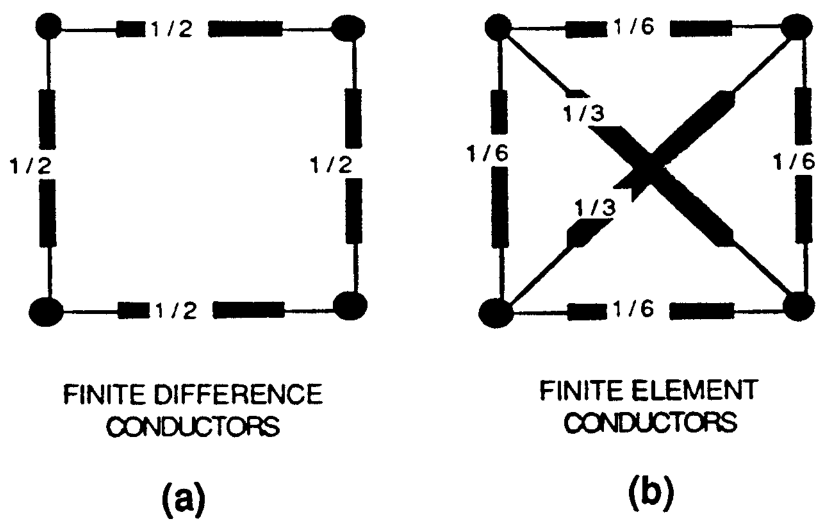


FIGURE 9.3 Conduction links for a square mesh. (a) Finite difference conductors; (b) finite element conductors.

conduction links are compared to the finite element conduction links in Figure 9.3. The presence of the diagonal links in the FE method allows for the accurate solution of distorted meshes.

Many thermal analysts use programs such as SINDA to solve heat transfer problems. SINDA is often referred to as a finite difference program, however, it is more accurately described as a matrix iterative solver. The calculation of the input coefficient matrix can be done using either finite difference or finite elements. It is possible to use an FE program such as NASTRAN to calculate the conduction and capacitance matrices, output the matrices from NASTRAN, reformat the matrices as SINDA input, then solve the system in SINDA. The resulting solution will be a finite element solution from SINDA. This approach has the advantage of the highly automated, graphics-oriented FE model generation programs, yet allows the analyst the use of traditional solution tools in the traditional thermal solvers.

Another problem associated with the traditional matrix solvers is that there is no graphical display of thermal contours since there is no associated geometry. If the FE approach described above is used, then the temperature contours can be displayed on the FE model. An additional step may be required to convert the solver output to a format which can be read into the FE postprocessor. These approaches are illustrated in flowchart form in Figure 9.4.

The finite element derivation for thermal analysis parallels the structural derivation leading to an element conductivity matrix:

$$[k] = \int [B]^T [E][B] dV$$

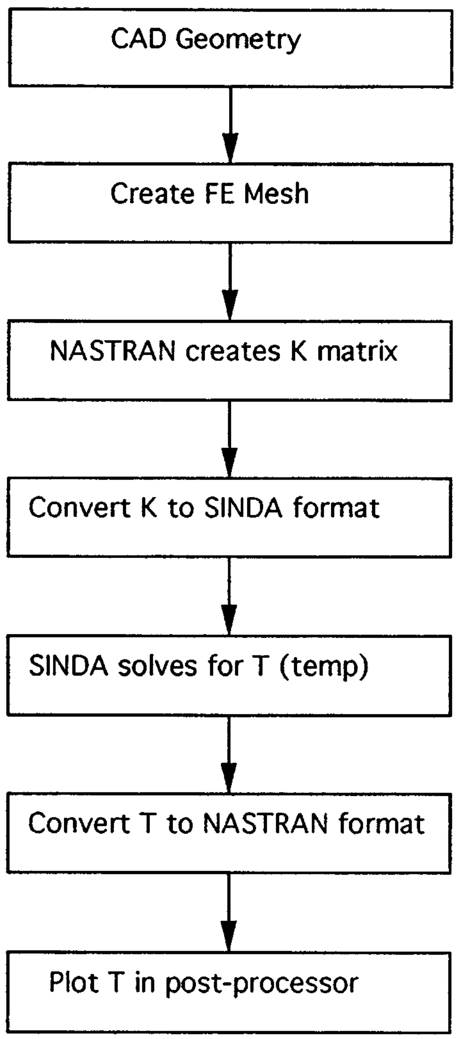


FIGURE 9.4 Thermal analysis flowchart.

where the [E] matrix is a diagonal matrix of material thermal conductivity and the [B} matrix is matrix of derivatives of assumed shape functions as described in Section 8.2 ("Derivation of Stiffness Matrix"). Assembly into a system level equation leads to:

$$[K]\{T\} = \{Q\}$$

where [K] is a symmetric system conduction matrix, {T} is the vector of nodal temperatures, and {Q} is the vector of applied nodal heat loads. Once boundary conditions (fixed temperatures) are applied, the system can be solved by many standard techniques.

Convection

Convection is a common heat transfer mechanism whenever fluids are present. Optics in such common machines as the office copier or a laser printer are affected by convection heat transfer, especially when fans are used to cool the machine. Although convection is not a consideration for orbiting telescopes in space, it must be included in the analysis of ground-based tests of such systems.

The most accurate calculation of convection requires the use of computational fluid dynamics (CFD) codes, which may be finite difference or finite element based. In these techniques the fluid is modeled in 2D or 3D space. Boundary conditions of pressure and temperature are applied, causing fluid motion and the resulting heat flow. These techniques can be very computationally intensive for transient 3D analysis, but are necessary when highly accurate results are required. A typical limitation of CFD analysis is that it is not coupled to the conduction analysis of the solids which thermally interact with the fluid. The CFD boundary conditions at the solids are either constant or prescribed in a time-varying manner rather than coupled to the conduction model. A common technique is to use CFD to find the effective convection coefficient or time-varying thermal boundary condition (BC) for a subsequent thermal analysis of the optical system.

A less expensive alternative to the CFD calculation is to use handbook values for that convection coefficients. For most applications a range of h values is given. By running an analysis at each end of the given range, the sensitivity of the model to the value of h can be determined. If the sensitivity is small, then the handbook value can be used with confidence; otherwise a CFD analysis or an experiment is required to determine a proper convection value.

Radiation

Radiation is a highly nonlinear effect that is especially important in the analysis of spaceborne optical systems. Most surfaces are usually treated as "gray body" such that they absorb a fraction of all incident radiation, and then reflect or re-emit radiation according to Lambert's cosine law independent of wavelength or incident direction. This simplifying assumption allows the use of the radiation equation in Section 9.2 ("Modes of Heat Transfer") to be used with view factors (F) calculated from the geometry. For very simple geometries and small models, simple equations are available for view factors. However, for most geometries found in detailed models, computer-based calculations are required. In NASTRAN, either a finite difference method or a contour integral method is available to calculate view factors between elements. As in most analyses, higher accuracy by using contour integrals or a small finite difference mesh requires more computer time. Typically, the calculation of view factors takes much longer than the solution of the nonlinear thermal problem.

Most optical systems include highly polished mirrors which reflect the incident radiation just as they reflect light. To analyze the heat transfer in systems with highly specular (reflective) surfaces, a ray-trace approach is needed to calculate the effective view factors. To get reasonable accuracy, many rays (>10,000) must be traced through the system, requiring significant computer time. Ray tracing can easily take ten times as long to calculate view factors as the finite difference method for gray bodies.

The radiation properties of real surfaces tend to be wavelength dependent. If high accuracy is required, the analysis should account for the wavelength effects. This complication is usually ignored in most cases, however, some modern programs (i.e., NASTRAN) allow wavelength-dependent effects.

Symmetry

Optical systems often contain high degrees of symmetry in the geometrical design. Just as in structural analysis, symmetry may be used to reduce the size of the analysis model. Symmetry techniques rely on the use of linear superposition to get the final answer. Nonlinearities such as radiation may prevent the use of model reduction in an otherwise symmetric system.

A symmetric body with symmetric thermal boundary conditions but with a generally nonsymmetric thermal load can be solved as a linear combination of two half-models. The half-model is solved once with symmetric BC and loads (Q_s) to give T_s and once with antisymmetric BC and loads (Q_A) to give T_A as in Figure 9.5. Assuming the right side of the body is modeled, then the submodel loads can be found from:

$$Q_s = 0.5 \times (Q_R + Q_L)$$

$$Q_A = 0.5 \times (Q_R - Q_L)$$

where Q_R represents the actual load on the modeled (right) half and Q_L is the load on the unmodeled (left) half. If Q_R and Q_L were equal, then there would be no heat flow across the cut boundary, making the symmetric boundary insulated. If Q_R was the negative of Q_L, then there would be no temperature change on the cut, making the antisymmetric boundary fixed. The resulting solution on the full model is T_R on the right side and T_L on the left side, where

$$T_R = T_S + T_A$$

$$T_L = T_S - T_A$$

A common special case is when the applied load is symmetric ($Q_R = Q_L$), in which Q_A is zero, so only the symmetric BC solution needs to be run. Note that in this case, $Q_S = Q_R$, so no new load calculation is required.

Just as in structures (Section 8.3), multiple planes of symmetry can reduce the model even more. This is generally useful only when the loads have the same symmetry, so the antisymmetric solutions are not required. The most common submodels in optical systems are 1/2 and 1/6 models.

Since symmetry can be thought of as a conventional mirror, the radiation view factors can be found for a half-model by placing a reflecting surface at the cut in the ray trace approach. Since the radiation is nonlinear, superposition does not apply. However, a purely symmetric solution is possible without superposition.

Solution Methods

A linear steady-state heat transfer problem has the form:

$$(K + H)T = Q$$

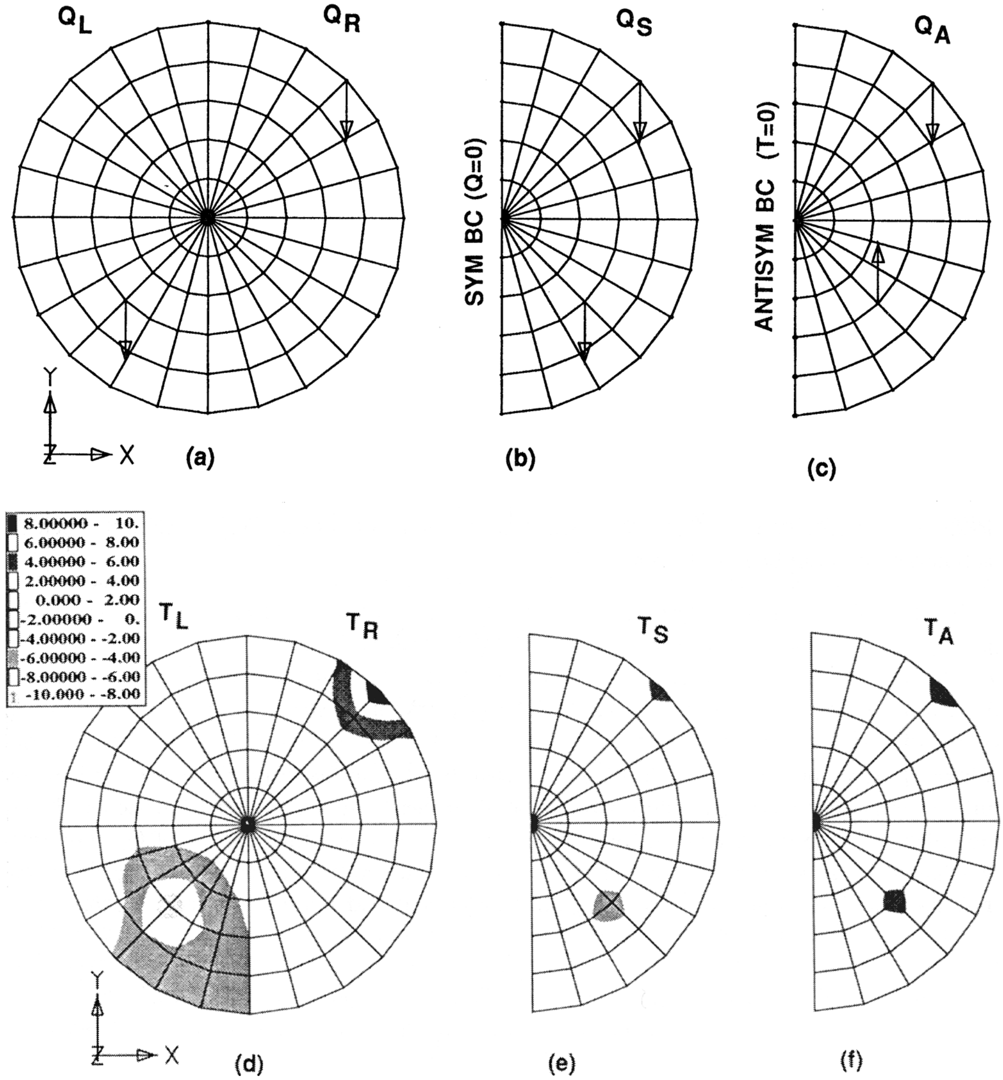


FIGURE 9.5 Thermal analysis of a symmetric structure with asymmetric loads. (a) Full model with general loads (Q_R and Q_L); (b) half-model with symmetric loads (Q_S) and boundary conditions; (c) half-model with antisymmetric loads (Q_A) and boundary conditions; (d) full model with temperature results (T_R and T_L); (e) half-model with symmetric temperatures (T_S); (f) half-model with antisymmetric temperature (T_A).

where the conduction (K) and convection (H) are temperature independent and the applied load (Q) is constant with time. A variety of solution algorithms are possible for large linear systems. A single pass solution such as Gauss elimination or Cholesky decomposition (NASTRAN), or an iterative scheme such as Jacobi (SINDA) are common techniques. The best method is problem (size, accuracy, conditioning) and computer resource (memory, disk space, speed) dependent.

When the problem becomes nonlinear due to radiation, temperature-dependent properties, or mass flow elements:

$$[K(T) + H(T)]T + [R(T)]T^4 = Q(T)$$

then variations of Newton's method are used. Lack of convergence can become a difficulty if the initial starting guess is not accurate enough. Convergence can also be improved by breaking the load into smaller increments so the algorithm can track the solution more closely.

Transient problems require numerical integration to track the solution through time:

$$[C]T' + [K + H]T + [R]T^4 = Q(t) + N(T,T')$$

where T' is the derivative of temperature (T) with respect to time (t) and the coefficient matrix [C] is the thermal capacitance matrix. Nonlinear effects require Newton-like iterations at each time step to stay on a converged solution path. For the best combination of accuracy and efficiency, an automatic time step adjustment algorithm will reduce the step size in periods of rapid change or lengthen the step size in periods of slow thermal change.

The $N(T,T')$ term on the right-hand side represents nonlinear load term which can be used to model thermal control systems. The temperature is sensed at a control node, then, based on its value, a heating or cooling load will be applied to other points in the structure.

9.3 Model Types

The choice of model types depends on the goals of the analysis. A lumped parameter model may be sufficient to determine the net power requirements in an optical system. If thermoelastic distortions are required, then a 2D or 3D continuum model is required.

Lumped Parameter Models

In a lumped model, each optic and major component may be treated as an individual thermal node. Conduction, convection, and radiation links to the other nodes are required. In transient analysis, the total capacitance of each node is also required. The capacitance calculation requires only the net volume or mass of each component which is often available from mass property tables. The thermal links, on the other hand, can be difficult to calculate. Simple geometric calculations or handbook values may be sufficient for many geometries at this level of approximation. Many heat transfer texts use the electrical circuit analogy for lumped parameter models.

One example of a lumped parameter model would be a system level model of an orbiting telescope which is used to determine the net power requirements to maintain the operating temperature when subjected to solar heating while also radiating to deep space. Often a thermal control system must provide heating, cooling, shading, or insulation to maintain the optics at operating temperatures.

A more common example would be a high volume office copy machine. The heat output from several sources in the machine must be controlled for it to function properly. The size and location of fans can be studied using a lumped parameter model.

Even in a more detailed 3D model, small optics may be treated as a single lumped node, creating a mixed model. Physical size is relative and is not always the governing criterion. If a small optic is critical to the overall performance of the system, then a distributed model is warranted.

In a general sense, a lumped model may provide enough information to calculate the despace or tilt in a system, but cannot provide any surface distortion predictions.

Two-Dimensional Models

When thermoelastic distortions are to be studied, the thermal gradient information is required within the optic. A 2D model will provide these results in a plane.

The most common 2D model for optics would be an axisymmetric model of a lens barrel (Figure 8.4). An axisymmetric model assumes that the structure, the boundary conditions (BC), and the thermal loading are axisymmetric.

If thermal gradients through the thickness of the optic are negligible, but the radial or circumferential gradients are not, a 2D shell model of an optic is possible. In Figure 9.6, the in-plane contours are shown for a thin optic with three edge supports subjected to laser heating. Another application would be the nonsymmetric temperature profiles in a thin pelicle due to an off-axis laser beam. Remember, these 2D models cannot account for the temperature gradients in the third direction.

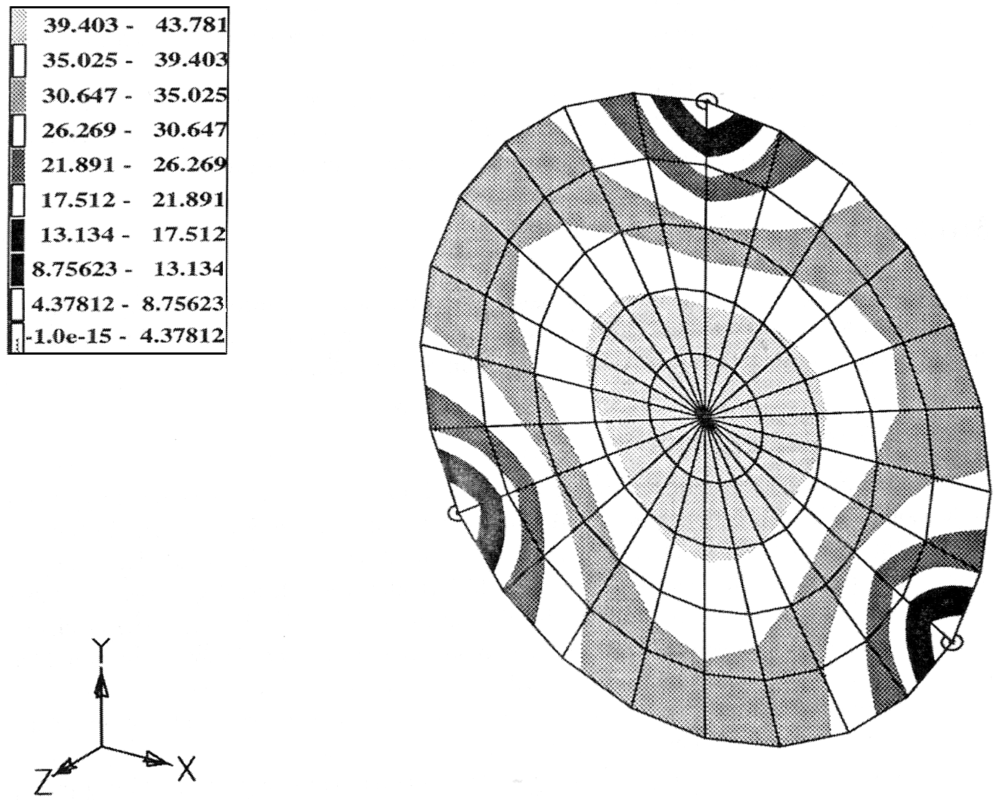


FIGURE 9.6 2D model with in-plane temperature contours.

Three-Dimensional Models

For most thermal problems the temperature varies in all spatial directions requiring a 3D model. If the geometry is very regular, finite difference techniques may be used to create the coefficient matrices. For most real geometries, computer-based modeling schemes such as finite elements are required. The retro-reflector (corner cube) in Figure 9.7 is an example of geometry requiring finite elements to get an accurate heat transfer model. The conduction matrix would be impractical to calculate without a computer-based geometry processor.

In some problems, the models for thermal and structural may be of different order. A thin optic may be represented as a 2D plate model for structural analysis, yet require a 3D thermal model to obtain gradients through the thickness. Most structural FE programs will allow specification of a thermal gradient through the thickness of plate elements. The reason that a 3D thermal model is required is because the temperature on the top and bottom surface are independent variables. A

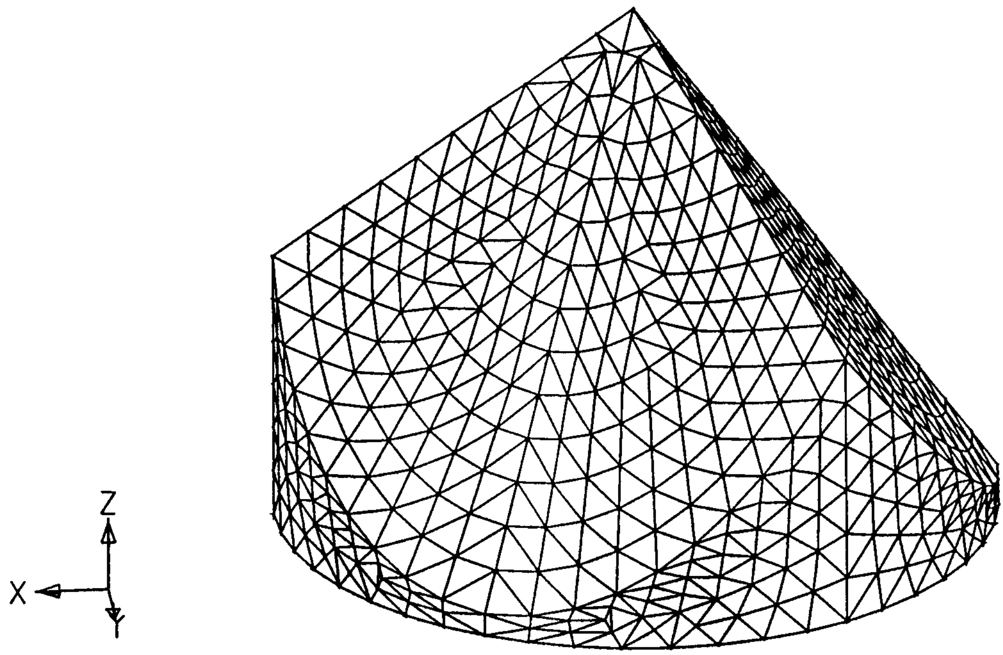


FIGURE 9.7 3D model of a corner cube.

midplane plate model has only a single variable (midplane temperature) to represent the response. In the 2D structural model, the midplane node has both displacement and rotations to represent the structural response. Thus, the out-of-plane bending of plate can be predicted by the equivalent moment loads caused through the thickness thermal gradients. However, the thickness growth is not accounted for by a plate model. If thickness change is important, a 3D solid model is required.

9.4 Interpolation of Temperature Field

In some analyses, the thermal and structural model are identical, allowing an easy transfer of temperatures to the structural model for thermoelastic response. The models are of the same geometrical order (i.e., 2D or 3D) and the node points have the same numbering and spatial location. In this case, some finite element programs allow for the solution of the thermal and structural problems in a single execution. If the thermal and structural model are solved in different executions, then the temperatures must be passed between the models. If thermal and structural analyses are run in the same program, this is usually automated. If different programs are used for thermal and structural, then the user may have to translate the output format of the thermal results to the input of the structural program.

A much more complicated interaction occurs when the models do not use a common mesh. This occurs in practical problems because the models are created to study different phenomena. Typically, the thermal model has a coarser mesh within an optic, because the thermal model must also include the exterior surroundings to accurately model convection and radiation effects. The structural model, on the other hand, may include only the optic being studied, but may have much higher refinement around mount points because of the stress gradients. Now the analyst must take temperatures from a thermal model, which has different node locations and numbering, and map those onto the structural model. Three techniques are discussed for the mapping of temperatures on to the structural model.

Nodal Averages

One approach is to search over all thermal nodes to find the N closest nodes to a given structural node. Then average the temperatures (T_j) of the N closest nodes, usually weighting by their inverse distance (d_j) from the structural node.

$$T_{avg} = (1/D)\Sigma(T_j/d_j)$$

$$D = \Sigma(1/dj)$$

This technique is limited to the interior of continuous and uniform structures, because it ignores the gaps, changes of materials, and boundaries. This technique is not accurate at important points like an optic mount location. Also, this technique always underpredicts the temperatures at the curved boundaries since it cannot extrapolate.

Interpolation via Conduction Models

An improvement on the modal averaging is a conduction analysis in the structural model. In this technique, the analyst maps the modal temperatures from the thermal model onto the single closest structural nodes. This is usually done 1 for 1, so that there are only NT (NT = number of thermal nodes) specified temperatures in the structural model containing Ns nodes (Ns = number of structural nodes). To obtain the remaining structural node temperatures, the structural model is converted to a thermal conduction model with the specified temperatures as fixed boundary conditions (BC). This model is then run to obtain the temperatures at the remaining structural mode points. The output from this model is then input to the structural model with a convenient interface since it has the same node numbers.

The advantage of this conduction interpolation is that no other software is required. Changes of material properties and gaps are accounted for in this approach, so that the mounts can be accurately described.

This technique has the major disadvantage of requiring point-to-point mapping of thermal node to structural node which is very time consuming and error prone. Interpolation errors are introduced because the thermal BC are not accurately described in the conduction run. The simple model in Figure 9.8 shows the type of error introduced. The bottom edge is fixed to T = 0, the sides are insulated, and a uniform heat input across the top edge causes a temperature of 100. A coarse thermal model (a) shows the resulting temperature contours. In (b), a detailed structural model is converted to a conduction model with the nine common nodes fixed to the temperatures from part (a). The resulting contours show the lack of correlation at the boundaries because the intermediate boundary nodes do not have any boundary effects enforced. In a 2D model the boundaries are lines, but in a 3D model the boundaries are every interior surface. This is a major flaw in the technique, because the most important structural behavior, such as the highest stress or optical surface distortion, occurs at the boundaries. Similar errors occur around the interior fixed points as well, which can cause unrealistic higher order surface distortions on optical elements.

Interpolation via Shape Functions

This technique[3] was developed to overcome the flaws in the above two methods. In this approach, a finite element representation is required for the thermal model since FE shape functions are used to interpolate temperatures to the structural nodes. The actual temperatures may be computed from any thermal analysis technique, but during interpolation the thermal nodes must be connected with finite elements. A general 3D approach will be described, but this could be specialized to 2D by replacing tetrahedrons with triangles.

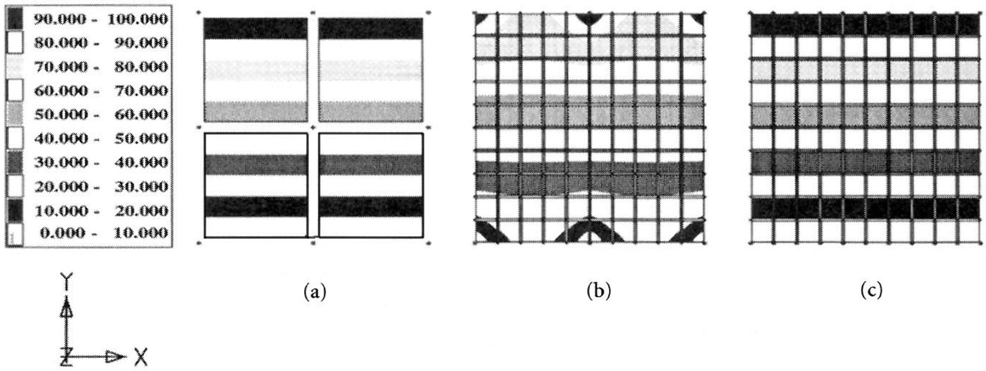


FIGURE 9.8 Interpolation of temperatures from coarse to detailed model. (a) Temperatures determined from coarse model; (b) interpolated temperatures by conduction model; and (c) interpolated temperatures by shape functions.

In step 1, the thermal FE model is converted to a solid model using two primitives, either a parametric tetrahedron or a parametric cylinder. ID finite elements such as beams or rods are converted to cylinders, whereas all 2D and 3D finite elements are converted to tetrahedra. Only the tetrahedra are described in this section since they are the most general. The subdivision of any solid hexahedron into 5 tetrahedron or the wedge into 3 tetrahedron is obvious. 2D plates are converted to solids by extruding normal to the surface 1/2 of the thickness in each direction. In this manner 4-noded quadrilateral become 8-noded hexahedron and 3-noded triangles become 6-noded wedges, which are then subdivided into tetrahedra. Note that this model is not used for thermal analysis, but only for the postprocessing interpolation step, so the aspect ratios are not important. Only the first-order elements are used because of the math involved. For first-order tetrahedra, the Jacobian matrix is constant throughout the element, making the geometric search for the node points reasonable. Also, most thermal models use linear elements because the temperature tends to behave more smoothly than stress.

In step 2, for any given structural node, all thermal elements are searched to see if the structural node is inside. This requires that the structural node $\{x_p\}$ be converted to an element's parametric coordinates by:

$$\{\xi_p\} = [J]^{-1}\{x_p\} + \{\xi_0\}$$

where [J] is the Jacobian matrix of the element and $\{\xi_0\}$ is the spatial center in element coordinates. The structural node is inside this element if the following conditions are satisfied:

$$0 \le \xi_{pj} \le 1 \text{ for } j = 1,3$$

$$0 \le \xi_{p1} + \xi_{p2} + \xi_{p3} \le 1$$

The search is performed over all elements until the above condition is satisfied.

Once the proper element is found, the corner temperatures are interpolated to the structural node using the appropriate finite element shape functions (N).

$$T(x_p) = \Sigma N_j(\xi_p) \times T_j \qquad j = 1,4$$

This process is performed on all structural points as shown in the flowchart in Figure 9.9.

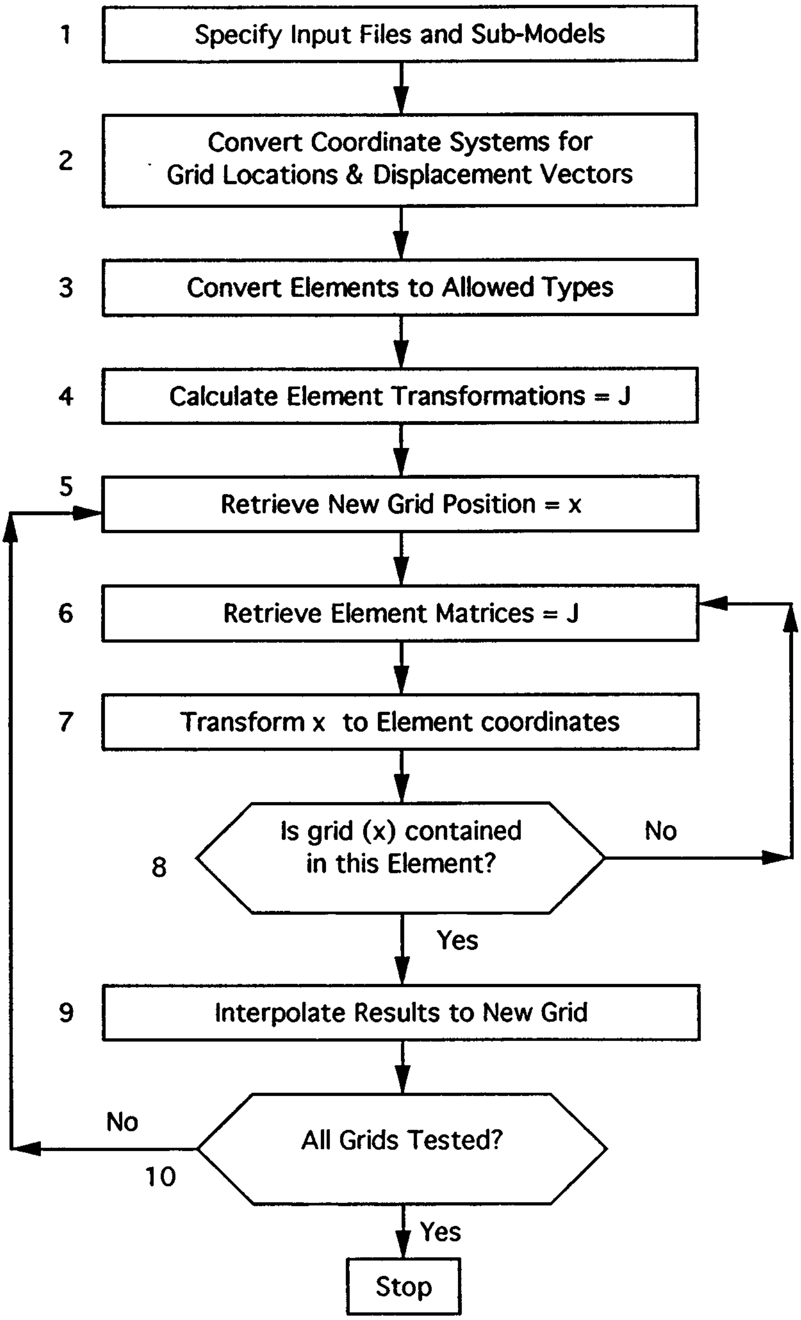


FIGURE 9.9 Flow chart for shape function interpolation.

This is a highly accurate approach, using the same finite element shape functions for interpolation as were used to solve the initial thermal model. The gaps, changes of material properties, and other geometric effects are accounted for automatically.

The first-order tetrahedron used has straight edges. If the real geometry has curved edges, then the boundary nodes on the finer detailed structural model may fall outside of all straight-sided thermal elements. This may be accounted for by taking a second pass through the search routine for any nodes not within an element. On the second pass a user-specified tolerance (ε) is used to modify the search bounds:

$$0 - \varepsilon \le \xi_{pj} \le 1 + \varepsilon \quad \text{for} \quad j = 1,3$$

$$0 - \varepsilon \le \xi_{p1} + \xi_{p2} + \xi_{p3} \le 1 + \varepsilon$$

This will catch nodes slightly outside an element and the shape functions will extrapolate a nodal temperature.

An example of a circular mirror with a mount ring (Figure 9.10) shows a thermal finite element model with temperature contours from an applied load. The finer detail structural model with its interpolated temperature contours shows the accuracy of this automated technique.

9.5 Thermoelastic Analysis

In many applications the goal of a thermal analysis is to determine the resulting image motion or surface distortion effects in an optical system. Thus, the temperatures obtained must be applied to a thermoelastic structural model. The techniques described in the previous section may be used to apply the thermal loads to the structural model. This section discusses some common analyses for optical structures.

Distortions and Stress

A discussion of proper modeling techniques and model checkout for the structural analysis is given in Chapter 8. When applying a temperature load to the system, a coefficient of thermal expansion (CTE = α) is required for each material. If a strain-free configuration occurs in a simple 1D rod at temperature (T_{ref}), then the equivalent mechanical force (F_T) caused by thermal strain (ε_T) is

$$F_T = AE\varepsilon_T = AE\alpha(\Delta T) = AE\alpha(T - T_{ref})$$

where the cross-sectional area (A) and modulus of elasticity (E) are given. In the general finite element notation of Section 8.2 ("Derivation of Stiffness Matrix"), the equivalent thermal forces are

$$\{F_T\} = \int [B]^T [E] \{\varepsilon_T\} dV$$

where B contains the derivatives of the element shape functions. These loads add to any existing mechanical loads and cause distortion and stress in the structure. The calculation of stress requires the correction for stress-free thermal growth:

$$\{\sigma\} = [E]\{\varepsilon - \varepsilon_T\}$$

where ε is the total strain due to all loads. This last effect may require some user action when using a finite element program to obtain stress. For example, in NASTRAN, standard subcases with thermal loads calculate the stress correctly; however, when using subcase combinations (SUBCOM), the thermal load request must be listed again to correctly backout the free thermal growth.

As noted in Chapter 8, accurate stress models generally require more detail than accurate displacement models whether the load is mechanical or thermal.

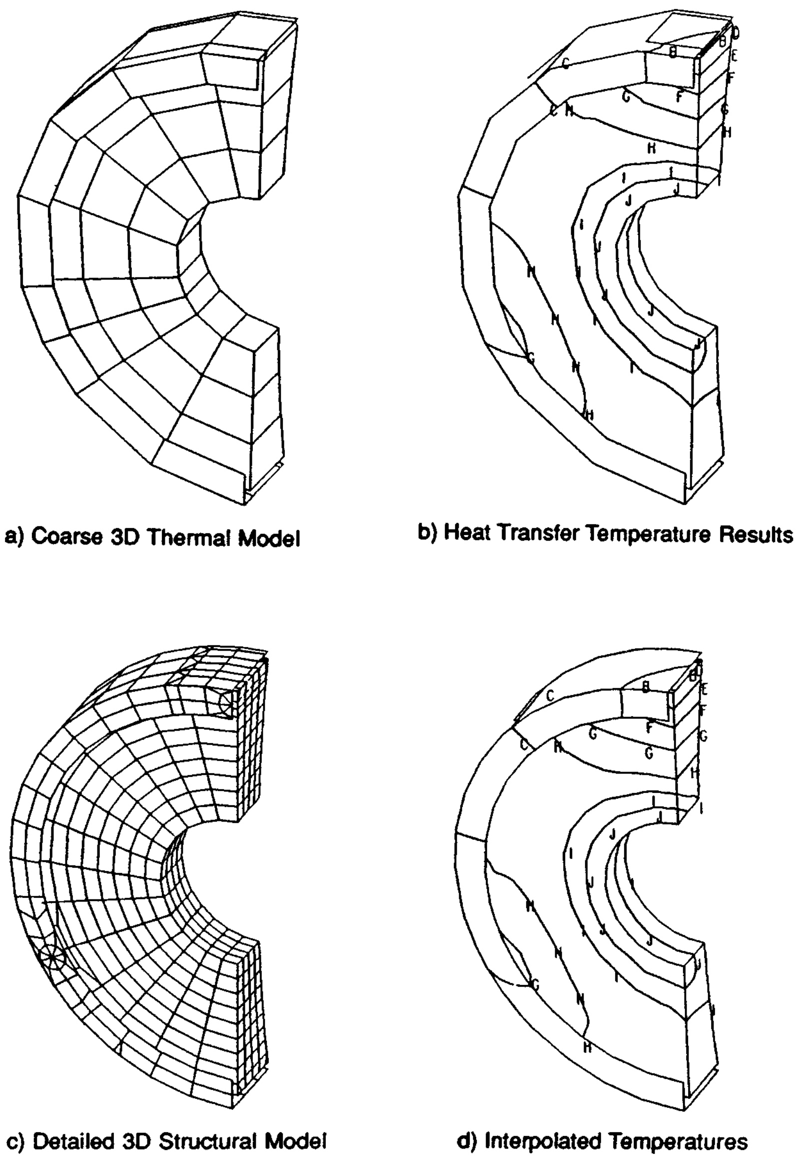


FIGURE 9.10 Shape function interpolation on a mirror and mount ring. (a) Coarse 3D thermal model; (b) heat transfer temperature results; (c) detailed 3D structural model; and (d) interpolated temperatures.

Rigid Element Problems

Rigid elements do not account for thermal growth, thereby introducing potentially large errors into the thermoelastic analysis. Often, rigid elements represent very stiff members which create very large thermal forces and dominate thermoelastic behavior, which is ignored in the rigid element formulation. The magnitude of the error introduced is dependent on the size of the rigid element used. However, even small elements used as offsets can cause large errors, as seen in the shell model with offset stiffeners in Figures 8.47 and 8.48. Offsets on element definition entries are implied rigid links and have the same effect as specifically defined rigid elements. Multipoint constraint equations (MPC) may add errors to the solution in the same manner as rigid elements. After all, the rigid elements are just an automated technique for creating MPC equations inside of the program.

To check for potential rigid body problems, the method described in the next section must be used. An alternative to rigid elements is to use very stiff structural elements which allow for thermal growth. The drawbacks to very stiff structural elements is that they may create numerical error in the solution. If the structural elements are too soft, the resulting flexibility may cause errors in the solution. Therefore, care must be used in their specifications.

Model Check via Thermal Soak

A useful model check for any thermoelastic model is the uniform thermal soak. In this check, all materials used are converted to a common CTE and T_{ref} values. Indeterminate boundary conditions are replaced by a statically determinate BC. A single load case of a uniform temperature change is applied. The magnitude of the temperature change is of the order of the maximum temperature change expected in the problem. The output is then scanned to see if the model predicts a totally stress free condition according to theory. Any nonzero stress is a measure of error created by rigid bodies, offsets, or MPC equations.

Node vs. Element Temperatures

Some finite element programs allow for the specification of either nodal temperatures or element temperatures. In NASTRAN, for example, if element temperatures are specified they take precedence over nodal temperatures. If element temperatures do not exist for a particular element, then the nodal temperatures are used to calculate them. For 3D elements, the nodal temperatures describe both bulk temperature change and gradients within the element. For 2D plate elements, the nodal temperatures can only describe midplane (membrane) thermal effects. The gradients through the plate thickness which cause bending must be specified on element temperature entries. The temperature interpolation schemes described in Section 9.4 only apply to the nodal temperatures, not element temperatures.

CTE Spatial Variation

For large optics, the coefficient of thermal expansion (CTE) may vary throughout the structure. Often an analyst is required to study the effects due to several possible spatial variations of CTE. Since the CTE appears on the material entry, separate models and runs are required to examine different CTE values. Even though CTE appears on the material entry, it has no effect on the stiffness matrix, only on the load calculation as shown in Section 9.5 ("Distortions and Stress"), above. When solving the linear system equation:

$$[K]\{\delta\} = \{F\}$$

for displacements $\{\delta\}$,

$$\{\delta\} = [K]^{-1}\{F\}$$

The major expense is the decomposition or inversion of the stiffness matrix [K]. The multiplication by {F} is very cheap computationally. If the vector {F} is replaced by a matrix of several columns, there is relatively little additional cost to obtain multiple solution vectors in $\{\delta\}$. Thus multiple load cases are much cheaper than multiple model solutions. An efficient technique is to represent each variation in CTE as a new load case. Using the equations in Section 9.5 ("Distortions and Stress"), the thermal strain caused by a variation in CTE (α^*) with a nominal temperature change (ΔT), is the same as that caused by a nominal CTE (α) with a variation of temperature (ΔT^*):

$$\varepsilon_T = (\alpha\times)(\Delta T) = (\alpha)(\Delta T\times)$$

where

$$\Delta T\times = \Delta T(\alpha\times/\alpha)$$

This method allows several variations in CTE to be studied in a single model run just by creating multiple thermal load vectors.

CTE Thermal Variation

In many materials, the CTE may vary over the temperature range of interest, especially in infrared applications which involve temperatures close to absolute zero. In thermal handbooks, the net thermal strain relative to a reference temperature (usually room temperature) is quantity plotted as a function of temperature. The instantaneous CTE is the current slope of the curve. However, when studying the net effect of a large isothermal temperature change, the net accumulated strain is the desired quantity from which the net CTE is derived:

$$\varepsilon_T = \alpha(\Delta T)$$

or

$$\alpha = \varepsilon_T/\Delta T$$

If the analysis program allows a temperature-dependent CTE, then this effect can be accounted for by providing a tabular $\alpha(T)$ input. If this feature is not included in the program, the user must input different CTE for each element, or modify the input load as shown in the above section.

Surface Coating Models

Many optical elements have a very thin surface coating applied for various optical reasons (transmission, reflectance, scratch resistance). The coating usually has different structural and thermal properties than the substrate. If the coating is very thin relative to the substrate, it stiffness may be ignored in the model; however, the loads induced by the mismatch in CTE or moisture absorption may be significant. As the relative thickness of the coating increases, it becomes more important to include its structural properties in the model. Several options for models are given below which offer varied complexity.

Model Type 1: Effective Gradient

Model the substrate as a single layer of plate elements with substrate properties, ignoring the stiffness of the coating. Apply an effective temperature gradient through the thickness to approximate the thermal moment effect. In the coating apply an effective temperature which is found by multiplying by the actual temperature (T) in the coating by the ratio of the coating CTE (α_c)-to-substrate CTE (α_s) and applying this temperature variation through the thickness via element temperature input:

$$T^* = T\alpha_c/\alpha_s$$

Model Type 2: Composite Plate

If the analysis program has a composite plate element which is typically used for graphite/epoxy structures, this element may be used for two-layered isotropic materials as well. This accounts for both stiffness and load effects of a coating in a convenient format. Interlaminar shear stress is usually provided which can be used to study layer debonding, as well as laminar stress to study cracking of the coating.

Model Type 3: Offset Plates

Without a standard composite element, the user can create a composite by using two overlapping layers of plate elements, one for the coating and one for the substrate. These must have the proper relative position by using element offsets so the proper moment is created. If element offsets are unavailable, create two planes of grids connected by rigid bars. The coating stress is available, but interlaminar shear is not.

Model Type 4: Solid-Plate

If the substrate is to be modeled with solid elements for other modeling considerations, then the coating can be added as a thin surface layer of membrane (or shell) elements with no additional node points required. Again, the coating stresses are available, but interlaminar shear is not.

Model Type 5: Solid–Solid

When the coating is of comparable thickness to the substrate, then it may be reasonable to model the coating as solid elements as well. This requires additional layers of node points resulting in larger matrices to be solved. In this case, the 3D effects and stresses are available.

Line-of-Action Requirements

Even small isothermal temperature changes can result in large internal forces in an optical system composed of a variety of materials. Using the equations in Section 9.5 ("Distortions and Stress"), a piece of BK-7 will generate a force of 46 lb for every square inch of area per degree F of temperature change. Obviously, internal forces of this magnitude can affect the performance of sensitive optical systems. For a model to accurately predict the behavior of a system, the load paths must have their proper geometric relationships. If two structural members join so that their neutral axes are offset, then internal thermal forces will cause a moment resulting in bending. An example of a mirror supported by a ring with its neutral axis aligned (a) and unaligned (b) with the mirror's neutral axis is compared in Figure 9.11. Obtaining the proper geometric relationship of member locations in a practical model requires additional effort from the analyst, often in the form of offsets, rigid links, or additional model detail.

Note that the amount of distortion caused by these offsets may be small for an automobile or an airplane, but they can be quite large compared to the wavelength of light. Thus, approximations which are valid in other industries may not be valid in optomechanics.

9.6 Analogies

Analogies are useful when an analysis technique developed in one field of engineering can be applied to solve problems in other fields which use the same type of governing equations.[1] One of the older techniques is to solve finite difference thermal problems as an electrical circuit network. Up until very recently, finite element structural or thermal codes were used to solve electrical field problems by analogy. Now several specific electric field finite element programs are available. In this section, some analogies useful in optics will be discussed.

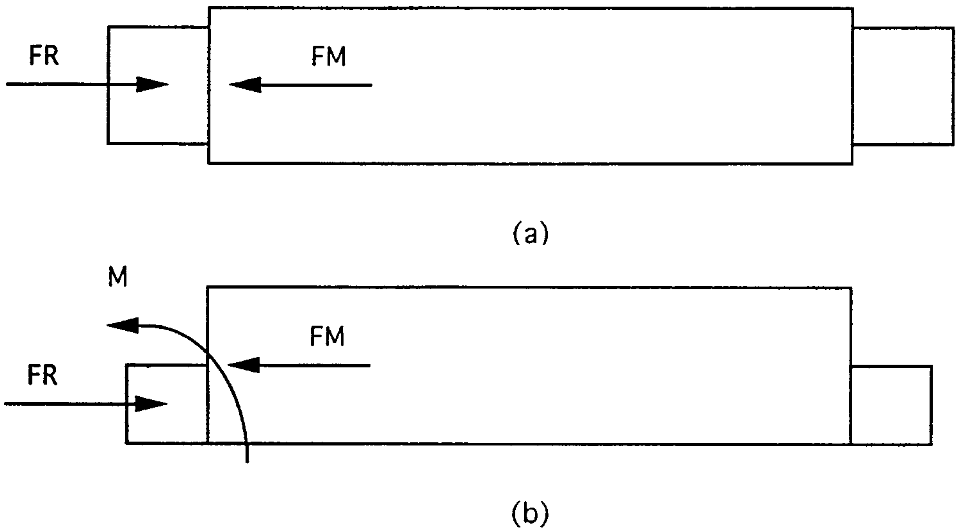


FIGURE 9.11 Line of action comparison on a mirror and mount ring. (a) Ring aligned with neutral axis and (b) ring unaligned with neutral axis.

Structural–Thermal

The analogy between structural and thermal analysis may seem unnecessary because there are many analysis codes available in both fields; however, an analyst trained in one field will appreciate the other topic more if he understands the analogy. Consider the 2D heat conduction problem in the XY plane. This problem can also be solved using a 2D structural analysis capability with the following modifications:

1. Let the X displacement = temperature.
   Set all other displacements = 0.
2. Define the material properties as:
   Young's modulus (E) = thermal conductivity (k)
   Shear modulus (G) = thermal conductivity (k)
   Poisson's ratio (ν) = 0
3. Apply fixed temperatures as fixed X displacement BC.
   Insulated boundaries = free displacement = no BC.
4. Apply point heat input as forces in the X direction.
   Apply distributed flux as distributed X traction pressure.
5. Convection can be treated as springs to the ground (ambient point)
6. Structural strain output = thermal gradient output.
   X strain (du/dx) = X gradient (dT/dx)
   Shear strain (du/dy) = Y gradient (dT/dy)
7. Structural stress = negative thermal flux.
   X stress (E × du/dx) = –X flux (k × dT/dx)
   Shear stress (G × du/dy) = –Y flux (k × dT/dx)

This analogy can be extended into 3D solids. Sometimes differences appear between the solutions because structural elements may have used advanced features in their development to improve their structural behavior. If both the structural and thermal elements use the same standard formulations the results will compare exactly.

A complete correlation table between the structural, thermal, and general field problems is given in Figure 9.12. Using this table, an analyst can solve a field problem using either a thermal or a

Field Problem	Heat Transfer	Structures
Variable (Φ)	Temperature (T)	Displacement (U)
X gradient (dΦ/dX)	X gradient (dT/dX)	X normal strain (dU/dX)
Y gradient (dΦ/dY)	Y gradient (dT/dY)	XY shear strain (dU/dY)
X flux (-k dΦ/dX)	X thermal flux (-k dT/dX)	X normal stress(E dU/dX)
Y flux (-k dF/dX)	Y thermal flux (-k dT/dY)	XY shear stress(G dU/dY)
diffusivity	thermal conductivity (k)	modulus (E = G)
1	thermal capacitance (ρc)	mass density (m)
Cauchy BC	convection coefficient (h)	elastic foundation
body force	volumn heat generation	gravity
surface force	surface flux	pressure
point force	point flux	force
Dirichlet BC	fixed temperature	fixed displacement
Neuman BC	insulated BC	free edge
Cauchy BC	surface flux	pressure

FIGURE 9.12 Analogy table for field, thermal, and structural problems.

structural analogy. Generally speaking, the thermal analogy is useful for scalar fields, whereas the structural analogy has the capability to represent vector fields.

Moisture Absorption

Plastic optics may, depending on their composition, absorb moisture and swell, causing a change in shape. The absorption of moisture follows Fick's law which is the same form as transient heat transfer. The following heat transfer analogy can be used to analyze the moisture concentration:

1. Moisture concentration = temperature.
2. Diffusivity = conductivity (capacitance = 1).
3. Moisture gradient = temperature gradient.
4. Moisture flow = thermal flux.

Once the moisture concentration has been determined, the moisture swell is analogous to thermoelastic expansion. Thus the following structural analogy applys:

1. Moisture concentration = temperature.
2. Moisture expansion coefficient = CTE.

The output of the thermoelastic analysis is the deformation due to moisture absorption which may then be added to other deformation effects for surface fitting as in Chapter 8.

Adhesive Curing

Many optics are bonded to their mounts with an adhesive. In the curing of an adhesive, the solvent evaporates according to equations of transient heat transfer. Thus the concentration of solvent is similar to moisture desorption so the above analogy holds. Shrinkage during curing is analogous to thermoelastic distortion. To apply these analogies, the proper coefficients must be obtained from test data since little or no published data are available.

Temperature-Dependent Index of Refraction

In some materials the index of refraction (n) is temperature dependent. As light passes through an optical element with a temperature distribution, the effective pathlength may vary from ray to ray. Thus a planar wave entering the optic may be nonplanar when it exists. For sensitive systems, this can have a bigger effect than the thermoelastic effect. Two modeling techniques are presented here to analyze the pathlength difference.

For windows and nearly plano lenses, the technique given in Genberg[2] is easy and efficient. In this technique, the 3D solid model used to determine the temperature distribution is converted to a modified "thermo-index" model. In this modified model, all displacements except those along the beam path are set to zero. Poisson's ratio is set to zero to eliminate any coupling of the displacements. The CTE is replaced by the index gradient (dn/dT) and the temperature distribution is the applied load. If the first surface is constrained to zero displacement, the second surface's displacement is the shape of the outgoing wave.

$$\Delta L = L\alpha\, \Delta T = L(dn/dT)\Delta T$$

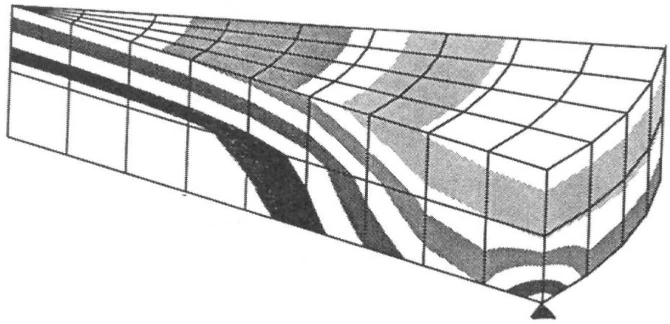
An example of this application is a window in a test chamber (Figure 9.13) subjected to thermal heating when a laser test beam is passed through. The temperature contours are shown on the 1/12 symmetric solid model. A 1/12 model was the smallest model describing the 6-point mount on the window. The apparent surface distortion from the "thermo-index" analogy model (b) represents the laser beam profile as it leaves the second surface of the window. In this application, the index effect in the window was bigger than the actual surface effects on the test article inside the chamber. With this analysis and a subsequent Zernike fit of surface 2, the window effects could be factored out of the test results to obtain the accurate test article response.

For more complex geometries, a new model is required. First, determine the beam path through the optic of many points distributed across the incoming beam. This may require an optical ray-trace program. Represent each optical ray as a string of truss (rod) elements with enough subdivisions to pick up the variations in temperatures throughout the optic. Use a temperature mapping algorithm to determine the temperatures at each node point along the ray path. Give the truss elements a CTE value of (dn/dT) and apply the temperature distribution as a load on a "thermo-index" model. If the surface 1 nodes are constrained, then the surface 2 nodal displacements represent the exiting wave's profile.

The corner cube in Figure 9.7 shows a complex application of the truss element approach in Figure 9.14. As an individual ray bounces off the multiple surfaces, the index effect must be summed over the ray segments. This is possible by writing multipoint constraint equations (MPC). If the displacement coordinate system is chosen so that δ_x is along each ray path, then at any reflective surface

$$\delta_x \text{ (outgoing ray)} = \delta_x \text{ (incoming ray)}$$

In this example, a plane of symmetry was used requiring an additional internal surface. The exiting beam from surface 6 represented the beam profile effects due to index changes. These were then fit with appropriate Zernike polynomials and added to other effects such as thermoelastic distortion.


(a)

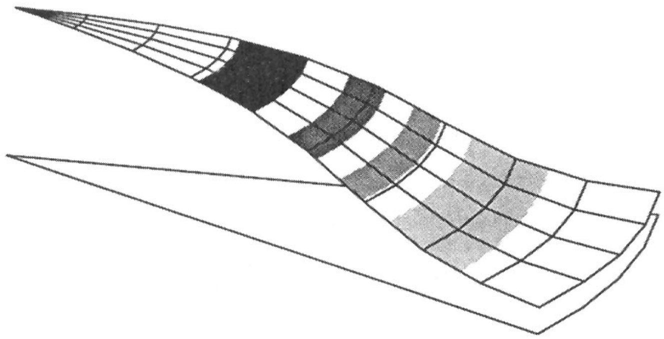

(b)

FIGURE 9.13 Optical pathlength effects due to index of refraction thermal sensitivity. (a) Temperatures of a 1/12 model of window subjected to laser heating and (b) resulting beam profile due to index changes.

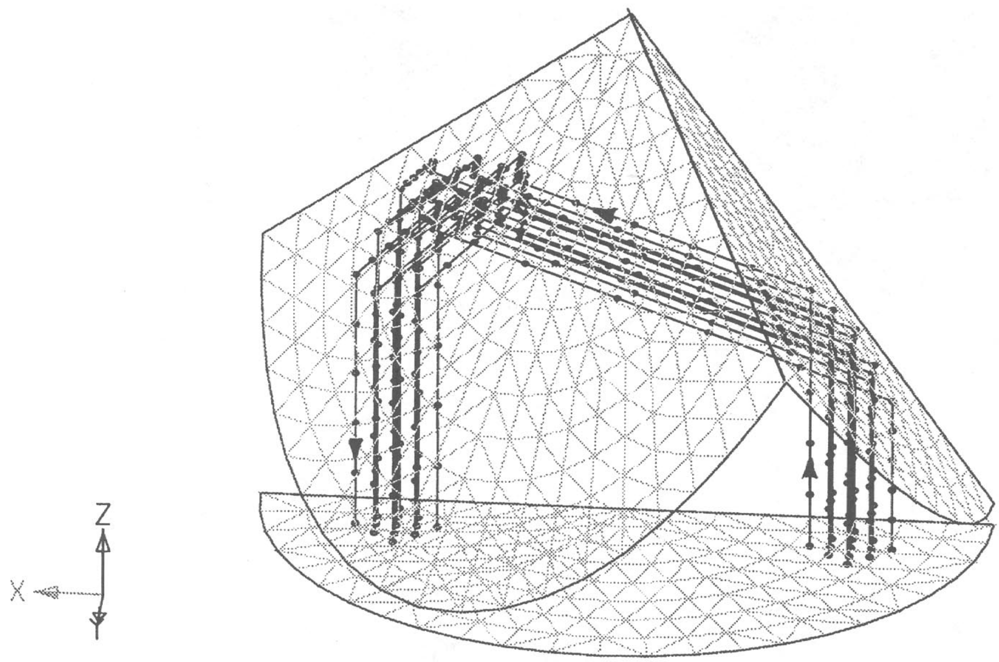


FIGURE 9.14 Individual beam paths in a corner cube represented as rod elements.

9.7 Summary

The thermal analysis of optical structures requires the accuracy of modern numerical methods such as finite elements to obtain thermal gradients throughout the system. Conduction, convection, and radiation can all be important modes of heat transfer in any system. The resulting temperature profiles are often input into a structural model to obtain the thermoelastic distortion. If the thermal and structural models are not the same, then some form of interpolation is required to apply the temperature results onto the structural model. Other problems, such as moisture absorption and swell, can be solved by analogy to thermal and structural solutions.

References

1. Genberg, V. L. 1986. Solving field problems by structural analogy. In *Proc. Western New York Finite Element User's Conference.* STI, Rochester.
2. Genberg, V. L. 1987. Optical pathlength calculations via finite elements. In *Proc. Structural Mechanics of Optics II*, Vol. 748–14. SPIE, Bellingham.
3. Genberg, V. L. 1993. Shape function interpolation of 2D and 3D finite element results. In *Proc. 1993 MSC World User's Conf.* MSC, Los Angeles.
4. Huebner, K. H. and Thorton, E. A. 1982. *The Finite Element Method for Engineers*, 2nd ed. Wiley, New York.
5. Krieth, F. 1958. *Principles of Heat Transfer*, 2nd ed. International Textbook, Scranton.
6. Segerlind, L. J. 1984. *Applied Finite Element Analysis*, 2nd ed. Wiley, New York.