Finite element analysis (FEA) techniques were used to assess the mechanical performance due to thermal loading of a high power, wide band, helix traveling wave tube's interaction (slow wave) circuit. This work was sponsored in part by the Air Force Office of Scientific Research. A steady state heat transfer analysis was performed using calculated heat dissipations and boundary temperatures that were obtained through data supplied by the TWT's manufacturer. Although the absolute amount of power dissipated by the interaction circuit is relatively small compared to the total heat dissipation of the TWT, it is locally concentrated over small areas which creates large heat fluxes. The resulting temperatures from this analysis were then used as loading conditions in a linear static analysis of a smaller model which represents a 12 turn section of the circuit. The failure modes investigated in this study were cracking of the helix tape and fracture of the support rods due to excessive thermal stresses. Cracking of the helix tape would cause an open-circuit to occur while fracture of the support rods could cause small mechanical perturbations in the slow wave structure which may reflect the RF signal. Both cases could possibly lead to electrical failure of the TWT. Static stress analysis of the attenuation section indicated that the stress levels in the helix and support rods due to this particular temperature gradient were within acceptable limits and would not fracture if these components were free of initial cracks or flaws. Stresses through the helix were sufficient to nucleate cracks, however the length of these cracks would be minute and would not affect the useful life of the helix.

The unique combination of bandwidth and power provided by TWT's has yet to be matched by any other device. The operation of a TWT depends on the continuous interaction of a beam of electrons with an electromagnetic wave. The interaction circuit analyzed in this study is from a helix TWT used in an electronic countermeasures (ECM) system. A cross-section of this circuit is shown in Figure 1. The circuit employs a rectangular cross-section helix tape which a radio frequency (RF) signal is propagated down at a speed near that of light. However, the axial velocity of the wave is reduced by the pitch of the helix. When an electron beam is injected along the axis of the helix, the axial electric field accelerate some electrons and decelerate others [7]. This causes electrons in the beam to form "bunches" which interact with the helix RF wave, surrendering energy to it. By the time the RF signal reaches the output coupler, it has been amplified exponentially.

The helix from this TWT is constructed of tungsten, has a rectangular cross-section and variable pitch. Three dielectric support rods equispaced around the helix act to hold the helix in place and also provide a path by which heat is removed from the helix. The rods are coated with a thin film of carbon over a length of about twenty helix turns starting approximately half way between the RF input and RF output. This carbon coating dissipates any waves that may be reflected back through the circuit from the output coupler in the form of heat. The helix and support rods are enclosed in a vacuum environment by a thin-wall barrel constructed of 304 stainless steel.

When the TWT is powered up, heating of the helix occurs from heat dissipations due to ohmic losses and electron beam impingement. These heat dissipations induce a temperature field through the circuit, which in turn, induces a thermal stress field. If stresses reach a critical value, cracking of the helix tape and/or support rods can occur, which could lead to electrical failure of the TWT. Whether or not these thermal stresses lie within the region of safe operation is a good indication of the interaction circuit's structural reliability.

The FEA code used for this study is Numerically Integrated Elements for System Analysis (NISA). NISA's steady state heat transfer code was used to simulate the circuit's response to a constant heat load and previously measured boundary temperatures. The resulting temperature distribution was used as a thermal load into NISA's static stress code. Results from the stress analysis were then used with the

* This work was supported in part by the Air Force Office of Scientific Research.
maximum normal stress theory to determine if the calculated stress levels were high enough to cause failure of the helix and support rods. Fracture mechanism maps were then used to determine the fracture behavior for both tungsten and beryllia. These maps take into account the temperatures and stresses experienced by these materials as well as the presence of pre-existing cracks or flaws.

This paper is organized in the following manner. First, the finite element model is described and attributes of this model are defined. Next, all input variables associated with the steady state thermal analysis are described and results of this analysis are presented. These results are then input into the stress analysis code and resulting stresses are presented. Finally, these results are interpreted using appropriate material failure and fracture theories.

**Finite Element Model**
A portion of the three-dimensional finite element developed for this study is shown in Figure 2. A 3-D model was constructed because it was felt that the heat flow through this structure would not be restricted to two dimensions. The model was constructed interactively using NISA's DISPLAY pre-processor. The entire model consists of 89 helix turns, support rods and vacuum envelope barrel. It contains approximately 18,000 elements and 45,000 nodes.

After the entire geometry was defined, finite elements were generated. The element type used for the heat transfer analysis was NISA type 104, a 3-D solid, hexahedron (brick) element suited for modeling 3-D heat flow. This element contains eight nodes, one per corner. For the static stress analysis, NISA element type 4 was employed. This element is based on a 3-D state of stress and is suited for modeling 3-D solids subject to 3-D loading. These two element types are identical in shape and orientation with their only difference being that type 104 has one degree of freedom per node (temperature), while type 4 has three degrees of freedom per node (x, y and z translations).

**Steady State Heat Transfer Analysis**
The steady state heat transfer analysis performed in this study simulated the interaction circuit's thermal performance under constant (non-time-dependent) heat dissipations and boundary temperatures. Generally, the typical temperature differences between the helix and the cooling jacket are such that the effect of radiation from the helix can be neglected and, with the helix in a vacuum environment, there is no convection [4]. Therefore, we can assume that heat is transferred only by conduction from the helix, through the support rods, out through the barrel.

**Heat Dissipation**
The interaction between the dc electron beam and RF wave is the most complicated process in the TWT. In calculating the heat dissipations through the interaction circuit, a modeling approach was adopted to work backwards from a given TWT RF output power and forwards from a given RF input power to determine an "effective" TWT gain parameter for the forward wave and an "effective" attenuation parameter for the reflected wave. To simplify this procedure, the following assumptions were made.

1. The interaction structure was modeled as a lossy coaxial transmission line. Ohmic losses are assumed to occur on both the helix and inner surface of the barrel.

2. The regions of the dielectric support rods which contain the attenuation coating partially dissipate the forward power and fully dissipate the reflected power.

3. The forward power gain is an exponential function of the interaction length.

4. TWT operation occurs in the small signal region only.

Using the aforementioned assumptions, heat dissipations through the circuit were calculated for forward, reverse, and combined forward/reverse power. The values for combined forward/reverse power were used as inputs to the finite element model because it was felt that this would represent a "worst case" heat dissipation. Also, since the heat dissipated at the surface of the barrel was an order of magnitude less than the heat dissipated by the helix, it would have little or no effect on the overall temperature distribution and therefore was not included in this analysis. Heat generation was simulated on the inside surface of the helix, which represented most of the forward wave loss, and on the attenuation portion of the dielectric support rods, which accounts for a small percentage of the forward wave loss and the entire reflected wave loss.

**Boundary Temperatures**
Boundary temperatures were designated over the entire outside surface of the barrel. The manufacturer of this particular TWT ran a test on
the same band TWT which provided realistic boundary temperatures. The TWT was first stripped down to the barrel and modified to route three thermocouple wires. Three thermocouples were then spotwelded to the barrel. The TWT was then re-assembled, the gun re-potted, and the test was performed. Temperatures were then recorded at several operating frequencies.

The case which most closely represented the conditions under which the heat dissipations were calculated, as well as a worst case condition was hot oil, 2:1 VSWR mismatch and RF at saturation. Using temperature values at the same frequency for which heat dissipations were calculated, temperatures were assigned at each node on the outside surface of the barrel. Linear interpolation was used to determine temperatures at node points that lie in between the thermocouple coordinates and it was assumed that these temperatures remained constant around the circumference of the barrel.

**Coupled Nodes**

To account for the helix-rods and rods-barrel conduction path, appropriate nodes on these structures were thermally coupled, that is, forced to exist at the same temperature. This coupling of nodes causes the calculated helix temperatures to be lower than the actual temperatures because the thermal interface resistances for the helix-rods and rods-barrel interfaces are not accounted for. These thermal resistances are a function of the contact pressures between the components, since there are no bonding materials, brazes or welds associated with the structure. This data was not available for these materials, therefore the thermal resistances could not be determined.

**Material Properties**

The NISA FEA code has the capability to handle temperature and/or time dependent material properties as well as directional dependent (anisotropic) material properties. For this analysis, all thermal conductivities were assumed to independent of crystallographic direction (isotropic). The thermal conductivities of all three materials were, however, temperature dependent. The nature of this dependency is illustrated in Figure 3 [10]. Each data point shown in Figure 3 was entered into the NISA input file. The NISA code interpolated between these points to determine a material's thermal conductivity at any temperature.

**Results**

NISA's steady state heat transfer was executed using all of the aforementioned input data. Figure 4 shows temperature contours through the helix and support rods at attenuation section. This contour plot indicates the interaction circuit's temperature distribution for combined forward/reverse power under the prescribed boundary conditions. The TWT's manufacturer indicated that these temperatures are a good lower bound for the problem but the maximum temperatures should be about 30% higher than the maximum temperatures determined in this analysis. This discrepancy was due to not accounting for the thermal interface resistances between the helix-rods and rods-barrel. These resistances are dependent upon the contact pressures between the components and could not be determined because of lack of information concerning the assembly process of this interaction circuit.

**Thermal Stress Analysis**

Since the size of the finite element model used in the heat transfer analysis was very large, a stress analysis, which involves more degrees of freedom than does a heat transfer analysis and therefore requires a much greater amount of computer memory, could not be performed on the entire circuit. A smaller portion of the interaction circuit (12 helix turns in length) was therefore selected to perform a linear static stress analysis. This section corresponds to the attenuation section of the circuit where the largest temperature gradient was found, and therefore is the location where the highest thermal stresses will occur. This smaller model consisted of 2,578 elements and 6,476 nodes.

**Thermal Loading**

The results of the NISA steady state heat transfer analysis gave temperature values at each node within the model. The calculated nodal temperatures were written to an output text file. A FORTRAN 77 program was written that read in this file and wrote a revised text file that contained temperatures only for those nodes present in the attenuation section finite element model. This updated file was then used as a thermal loading condition for the subsequent stress analysis.

**Boundary Conditions**

When constraining a finite element model, one must be careful not to over constrain or inadequately constrain the model since numerical inaccuracies would result. Constraints must be
placed on all degrees of freedom associated with the model. If the model were allowed unlimited displacement in any direction, an incomplete solution would arise. When applying these boundary conditions, precautions were taken not to over constrain the model and to apply proper, realistic displacement constraints.

First, all barrel and rod nodes lying on the z=0 plane were constrained in the axial direction. Movement in the axial direction was permitted for all remaining nodes. Secondly, a strip of nodes lying along the axial length of the barrel were constrained in the radial direction. Movement in the radial direction was permitted for the remaining nodes. Finally, as a rotational constraint, one node lying on the outside surface of the barrel was constrained, rotation for remaining nodes was permitted.

To account for interfaces between the interaction circuit components, appropriate nodes lying on the helix-rods and rods-barrel interface were coupled together. These nodes were coupled in the radial direction only. This allowed the nodes to move relative to each other in the rotational and axial directions.

**Material Properties**

The material properties necessary for the stress analysis are shown in Table 1 [1,4,11]. It was assumed that all materials exhibited isotropic behavior. All properties exhibited little or no change with increasing temperature with the exception of the thermal expansion coefficient for tungsten.

**Results**

At the onset of this study, the failure modes to be investigated in this study were cracking of the tungsten helix and beryllia support rods. Since there have been no documented TWT failures due to failure of the barrel, stresses through the barrel were ignored.

Before any failure theory could be selected to determine of the calculated stress levels would cause failure, the mechanical behavior of tungsten and beryllia in the operating temperature range had to be identified. Beryllia, a ceramic material exhibits brittle behavior as do most ceramic materials. Tungsten, a metal, exhibits ductile behavior at very high temperatures. However, for temperatures below 1000 degrees C, tungsten behaves in a brittle manner [4,11].

For isotropic materials that fail by brittle fracture, the maximum normal stress theory is the best theory to use [5]. According to this theory, failure will occur in tension if the maximum calculated principal stress exceeds the material's ultimate tensile strength, and failure in compression will occur if the minimum calculated principal stress exceeds the material's ultimate compressive strength.

Stress contours through the helix and support rods are shown in Figures 5 and 6. The contours shown in Figure 5 are maximum principal stresses (maximum normal tensile stresses), while Figure 6 shows minimum principal stress (maximum normal compressive stress) contours. These results are summarized in Table 2. The ultimate tensile and compressive strengths for tungsten and beryllia are given below [1,4,11]:

**Tungsten**

Ultimate tensile strength = 198-215 ksi
Ultimate compressive strength = 186-210 ksi

**Beryllia**

Ultimate tensile strength = 23 ksi
Ultimate compressive strength = 18,800 ksi

The maximum normal tensile and compressive stresses through the helix and support rods given in Table 2 lie within the safe range specified by the maximum normal stress theory. Therefore, according to this theory, and the fact that brittle materials do not experience fatigue failure due to accumulation of damage, it can be concluded that mechanical failure of the helix and support rods will not occur when the TWT is operating at full forward power and 1000 watts reflected power.

**Fracture Analysis**

The fracture behavior of a structure depends on several factors. Stress level, material properties, type of fracture mechanism and presence of pre-existing flaws all impact fracture behavior. If a pre-existing crack or flaw is present, failure can occur over a period of time even if the overall component stress is below its critical failure stress. For a crystalline solid loaded in tension, several fracture mechanisms exist. However, for materials that exhibit brittle behavior at sufficiently low temperatures, fracture occurs by cleavage and brittle intergranular fracture [2].

The fracture behavior for tungsten and beryllia was obtained through the use of fracture mechanism maps. These maps are shown in Figures 7 and 8 for tungsten and alumina, respectively [2]. The fracture mechanism map for alumina is typical of the maps for many oxides, and therefore was used to analyze beryllia's fracture behavior. The ordinate on these maps is the material's homologous temperature (T/Tm). T represents the material's temperature and Tm the...
material's melting point. From the temperatures determined in the steady state heat transfer analysis, \( T/T_M \) for both tungsten and beryllia is approximately 0.1. The abscissa on these maps is the normalized tensile stress, which is defined as the maximum normal tensile stress (Table 2) divided by the elastic modulus (both of these variables are dimensionless). Since \( T/T_M \) for both materials is around 0.1, the only fracture mechanisms present will be Cleavage 1 and Cleavage 2. Cleavage 1 occurs when pre-existing cracks or flaws are present which can propagate at stresses below the critical failure stress. When a crack reaches a critical length, fracture occurs. If pre-existing cracks or flaws are small or absent, stresses can reach a level which can nucleate cracks. This regime of nucleation of cracks is called Cleavage 2.

For tungsten, the normalized tensile stress was calculated to be 0.00159. The point (0.1, 0.00159) in Figure 7 lies on the boundary between Cleavage 1 and Cleavage 2. This indicates that fracture can occur if there are pre-existing cracks or flaws and it is possible for cracks to nucleate if pre-existing cracks or flaws are small or absent. The normalized tensile stress for beryllia was calculated to be 0.00029. The point (0.1, 0.00029) in Figure 8 lies within the Cleavage 1 regime. Therefore, if there are no pre-existing cracks or flaws present, fracture of the support rods will not occur.

**Conclusions**

Results from the finite element analyses indicated that the interaction circuit's mechanical design is sound and would not exhibit any mechanical failure mechanisms as long as there were no pre-existing cracks or flaws in the helix or support rods. If thorough inspection techniques are used on these components prior to TWT assembly, there would be no mechanical failures associated with the interaction circuit. The results of this study have demonstrated that finite element element analysis is a valuable tool that can be used to assess interaction circuit thermal/mechanical performance. The heat transfer capability of the slow wave structure is one of the most important factors that limit the output power capability of helix TWT's. Tube designers must be able to examine thermal behavior of the circuit, especially the dielectric support rods which play a critical role in removing heat from the helix [4]. By employing finite element techniques, tube designers can be able to estimate the feasibility of a projected design by predicting influences of various physical and geometrical parameters of the interaction circuit on helix temperatures for various power dissipations.

The results of this effort have significantly extended the fundamental understanding of the material behavior within helix interaction circuits subjected to forward and worst case reflected power. A cost effective analysis technique has been developed for modeling helix interaction circuits and analytically predicting dynamic behavior while operating in their system environments. This technique can be used to determine the short and long-term effects on materials used within TWT interaction circuits and the long-term mechanical structural instabilities that exist within a TWT's vacuum envelope.

**References**

Table 1 Mechanical Properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Modulus of Elasticity (x10^6 psi)</th>
<th>Poisson's Ratio</th>
<th>Density (g/cm³)</th>
<th>Thermal Expansion Coefficient (x10^-5/C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tungsten</td>
<td>50.0</td>
<td>.28</td>
<td>.697</td>
<td>4.44 @ 25 C, 4.48 @ 150 C, 4.55 @ 300 C</td>
</tr>
<tr>
<td>Beryllia</td>
<td>48.0</td>
<td>.23</td>
<td>.105</td>
<td>8.0</td>
</tr>
<tr>
<td>304 Stainless Steel</td>
<td>29.0</td>
<td>.30</td>
<td>.286</td>
<td>1.7</td>
</tr>
</tbody>
</table>

Table 2 Component Stresses

<table>
<thead>
<tr>
<th>Component</th>
<th>Maximum Principal Stress (ksi)</th>
<th>Minimum Principal Stress (ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helix</td>
<td>79.7</td>
<td>102.6</td>
</tr>
<tr>
<td>Support Rods</td>
<td>13.92</td>
<td>43.6</td>
</tr>
</tbody>
</table>
TWT Interaction Circuit
Finite Element Model

Helix

Support Rods

Barrel

Figure 2

Steady State Thermal Contours – Attenuation Section
(Helix and Support Rods Only)

Figure 4

Units in Degrees C

Maximum Principal Stresses
Attenuation Section

Figure 5

Minimum Principal Stresses
Attenuation Section

Figure 6

Units in psi