TRANSIENT THERMAL ANALYSIS OF A FUEL ROD CASING

William J. Treytl, Senior Staff Engineer
FMC Engineered Systems Division, Santa Clara, CA.

Jeffery J. Bradford, Senior Engineer
FMC Central Engineering Laboratories, San Jose, CA.

ABSTRACT

The application of the NASTRAN thermal analysis capabilities to the prediction of material strength after high thermal loading is presented. The investigation into the materials' strength envolved a two-step analysis. The first step was to determine the materials transient thermal profile, which was followed by an elastic/plastic/creep analysis using the thermal profile as the structural loading. This paper presents only the thermal analysis phase of the investigation.

INTRODUCTION

A nuclear reactor composed of many uranium rods each encased in a metal housing is cooled by exterior air flow. One of the loading conditions such a nuclear reactor must be designed for is an unexpected large power pulse which produces a high temperature load on the fuel rod casings. These metal casings which contain the uranium fuel have to be designed so that their structural integrity is maintained under these adverse conditions. Given this design criteria, the problem was to determine the residual affects (i.e. the decrease in structural integrity) which the fuel rod casing would undergo during a high temperature loading cycle.

The approach taken on this problem was to use the finite element technique to solve for the temperature profile, whereupon a non-linear structural analysis could be performed using the temperature profile as the structural loading.

Since a thermal analysis cannot be performed simultaneously with a structural analysis, the project was divided into two phases;

Phase 1. Transient Thermal Analysis
Phase 2. Structural Analysis

Due to the magnitude of the problem (which will be described below) the thermal phase was divided into two tasks;

Task 1. Determine the location along the rod where the worst temperature loading occurs.
Task 2. Determine an accurate transient thermal profile of the rod cross-section at the location where the worst temperature loading occurs.

This paper presents only the thermal analysis phase of a verification project which was performed on a nuclear fuel rod casing. Due to proprietary reasons, all magnitudes and dimensions have been changed.

PROBLEM DESCRIPTION

The thermal problem consisted of the following;

1. The fuel rod was assumed to be initially at ambient temperature.
2. The ends were assumed to remain at ambient temperature during the thermal cycle, effectively becomming heat sinks.
3. The power pulse was the only heat source.
4. Heat could only be removed through conduction to the rod ends and through convection to the coolant air flow. Heat flowed by conduction through the casing to the air-cooled surface.
5. Coolant air flowed through the square channels formed by the adjacent bevelled casing corners. (see figure 1)
6. The air flow was assumed to remain constant and to be initially at ambient temperature.
7. The casing was assumed to be in contact with the fuel material except at the bevelled corners, allowing thermal conduction between the fuel and casing.
8. The casing was assumed to be in contact with adjacent casings execpt at the bevelled corners. Thus allowing thermal symmetry to be assumed.
9. All thermal properties were assumed to vary linearly.

A cross-sectional view of the core area of a reactor and the fuel rods is shown in figure 1. Note the square channels formed by the bevelled corners of adjacent casings. Air is forced through these spaces for cooling purposes. A more detailed description of an individual fuel rod is shown in figure 2.

Even though the internal heat source and heat sinks are symmetric about the rod center, the expected hotspot on the rod would be off center due to the forced air convection. Since the thermal profile had to be known in detail at the hotspot, the thermal analysis became two-fold.

1. Determine the location of the hotspot on the fuel casing.
2. Determine the temperature profile of the casing at the hotspot location.

The transient thermal loading used was the power pulse shown in figure 3. Along the length of the rod, the pulse was scaled by a cosine function such that the intensity was 50% at the fuel/graphite boundary and 100% at the center of the rod.

FINITE ELEMENT MODEL

In order to perform the thermal analysis in a cost effective manner, two finite element models were built of the fuel rod. The first model utilized a coarse finite element grid and was used primarily for determining the location of the hotspot on on the fuel rod casing. The second model was exactly like the first except that the hotspot area had been refined so that an accurate temperature profile of the rod casing could be obtained.

The two models are shown in figures 4 and 5. Due to the symmetry of the fuel rod assembly, only one eight of the cross-section was modeled.

The table below indicates the elements used to model the various sections of the fuel rod.

Metal Casing	CQUAD4 (Shell element)
Fuel Material	CHEXA _CPENTA (Solid element)
Air Flow	CFTUBE (Line element with flow properties)
Convection	CHBDY (Element with convection properties)

TABLE 1

The heat sinks at the ends of the fuel rod were simulated by specifing a constant temperature at the boundaries of the model. Enforcing this constant temperature during the transient analysis was accomplished in the following manner;

1. Using a CELAS2 card, a large conductance was applied to the boundary grid points.
2. Using a DAREA card, a load scaling factor consisting of the desired ambient temperature times the conductance applied on the CELAS2 card, was set at the boundary points.
3. A TABLED1 card was used to define a unit forcing function which would act on the DAREA inputs for the duration of the of the thermal transient.
4. A TLOAD1 card was used to define this enforced temperature loading.

The interanl heat source created by the power pulse was simulated in the following manner;

1. Using QVOL cards the rate of internal heat generation within the elements was specified. This rate varied along the rod length as a cosine function.
2. Using a TABELD1 card the actual magnitude vs. time power pulse was defined.
3. A TLOAD1 card was used to define this internal heat source load.

The cooling due to forced air convection was simulated in the following manner; (see figure 6)

1. The air flow was modeled with the CFTUBE element and PFTUBE property cards. The property card allowed specification of the volume flow rate and the heat capacity of the air.
2. Using a CHBDY element with the element type specified as POINT, the convection heat transfer between the fuel rod casing and the air flow was simulated. Using the PHBDY property card, the effective convection area per grid point was specified.

The initial temperature of the fuel rod was specified with the following cards;

1. TEMP cards were used to specify the initial temperature at every grid point.
2. In the case control deck, an TEMP(ESTIMATE) card was used to select the set of TEMP cards in the bulk deck.

The integration and output time steps were specified by a bulk TSTEP card and selected by a TSTEP case control card.

RESULTS and CONCLUSIONS

The results of this thermal analysis were compared with results obtained from other analyses previously performed by an independent consultant. Good correlation was obtained between the two.

Since the results of this analysis are proprietary, it is felt by the authors that this paper will serve a useful purpose if it was used as an example of the utilization of the NASTRAN transient thermal analysis capabilities. With this purpose in mind, the following appendix has been written to demostrate how the NASTRAN data deck was set up to perform the above analysis.

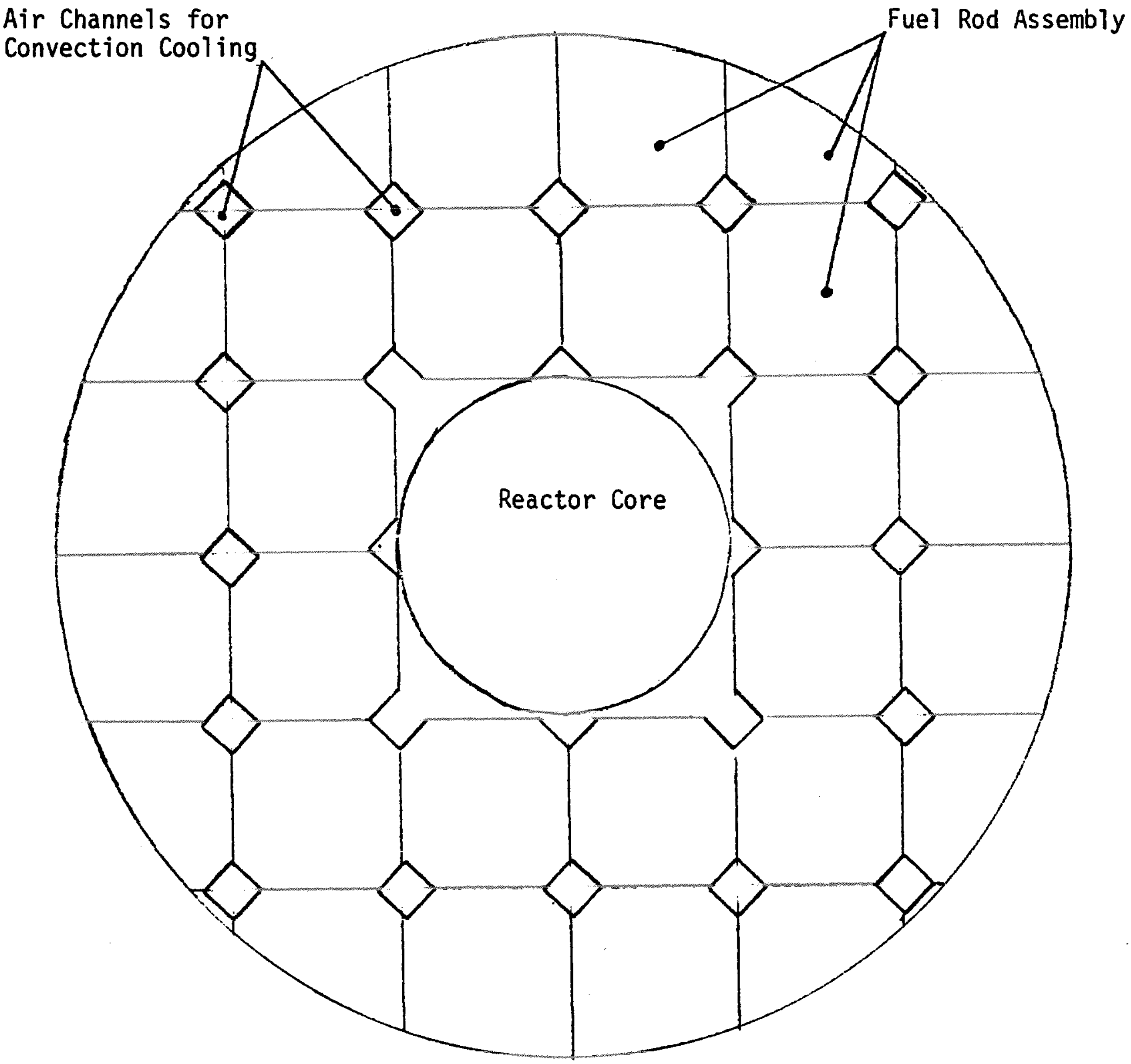
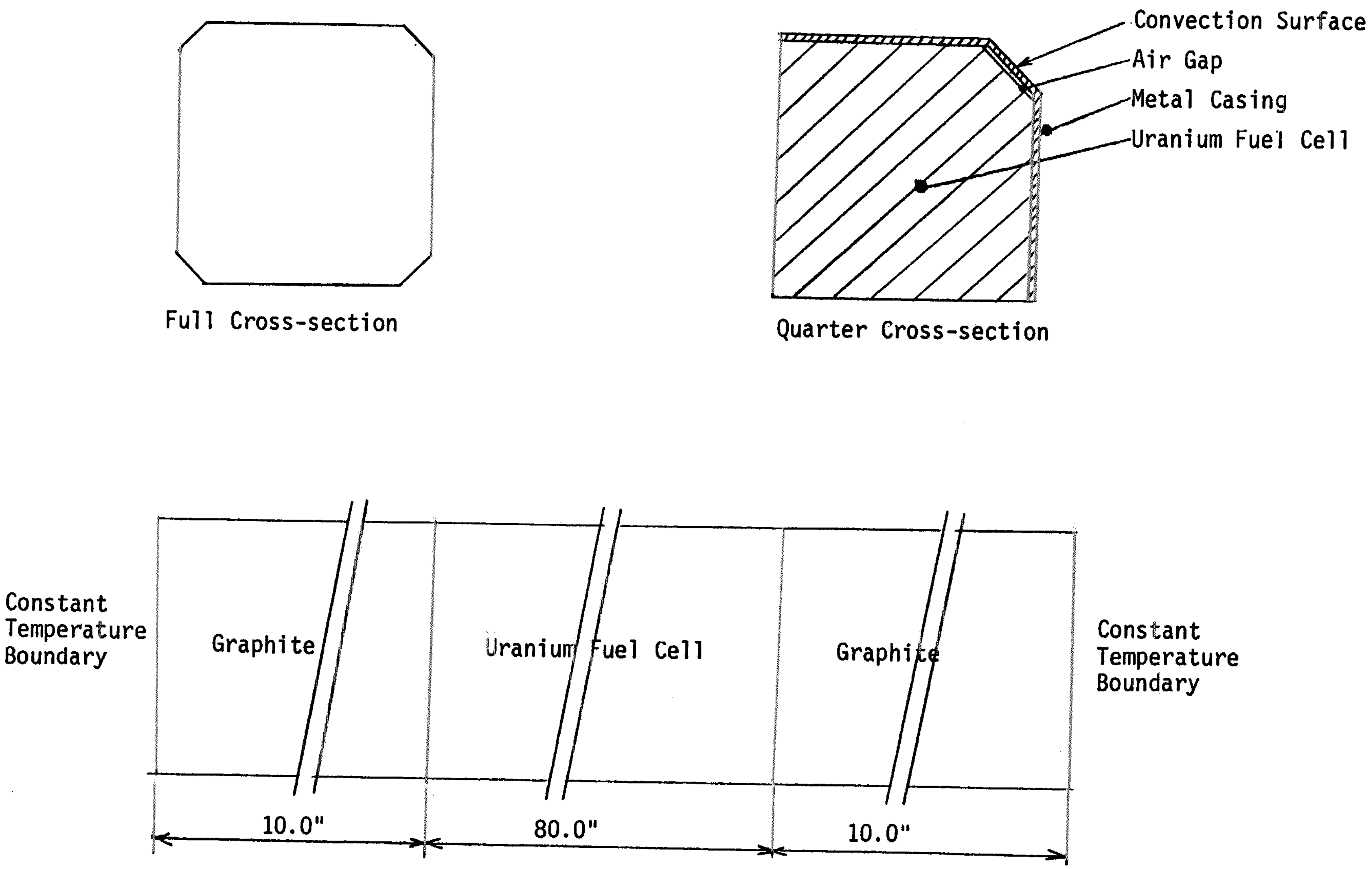


FIGURE 1, Cross-section view of the core area of a nuclear reactor.



Full Cross-section

Quarter Cross-section

FIGURE 2, Detailed drawing of individual fuel rod

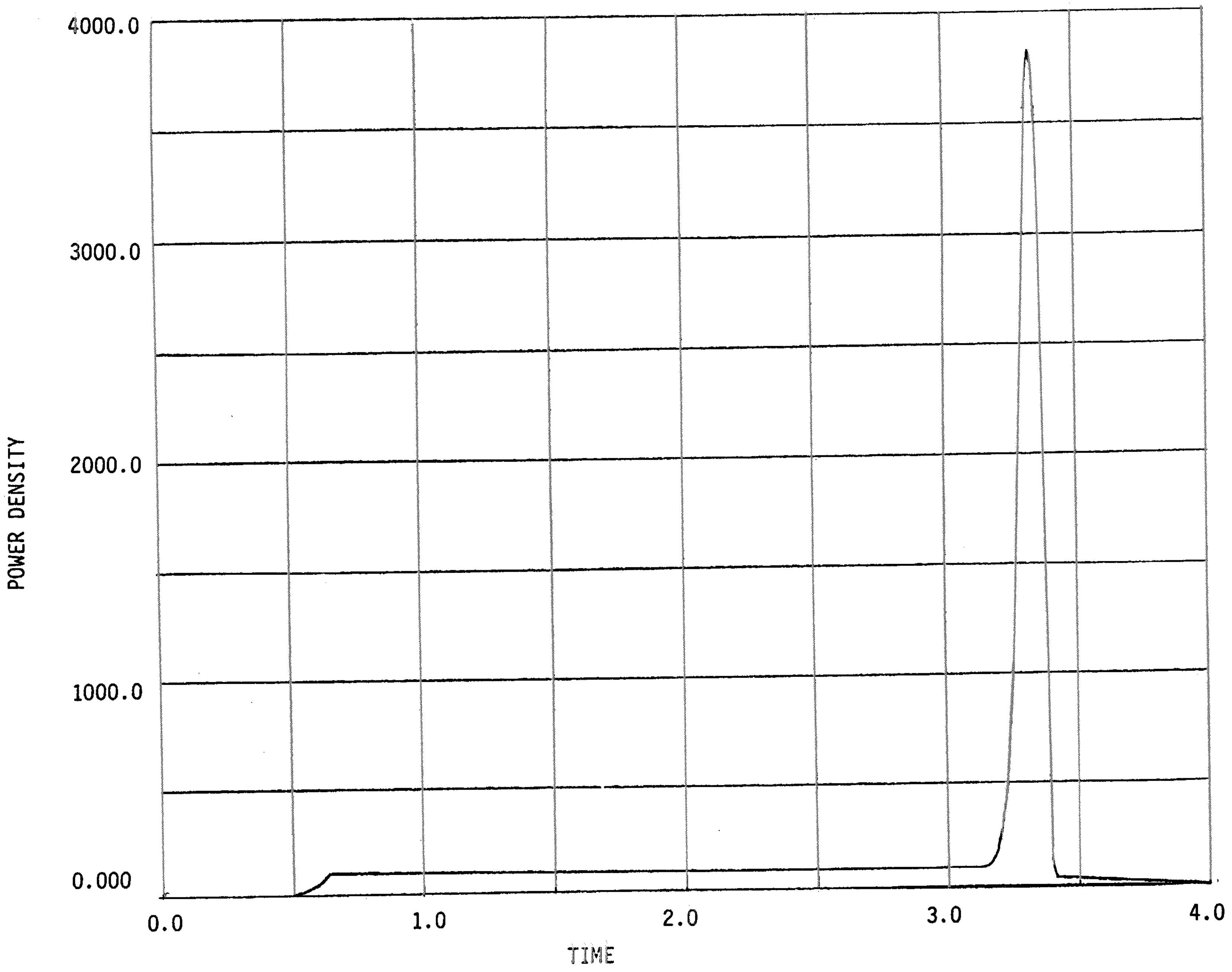


FIGURE 3, Power pulse input curve

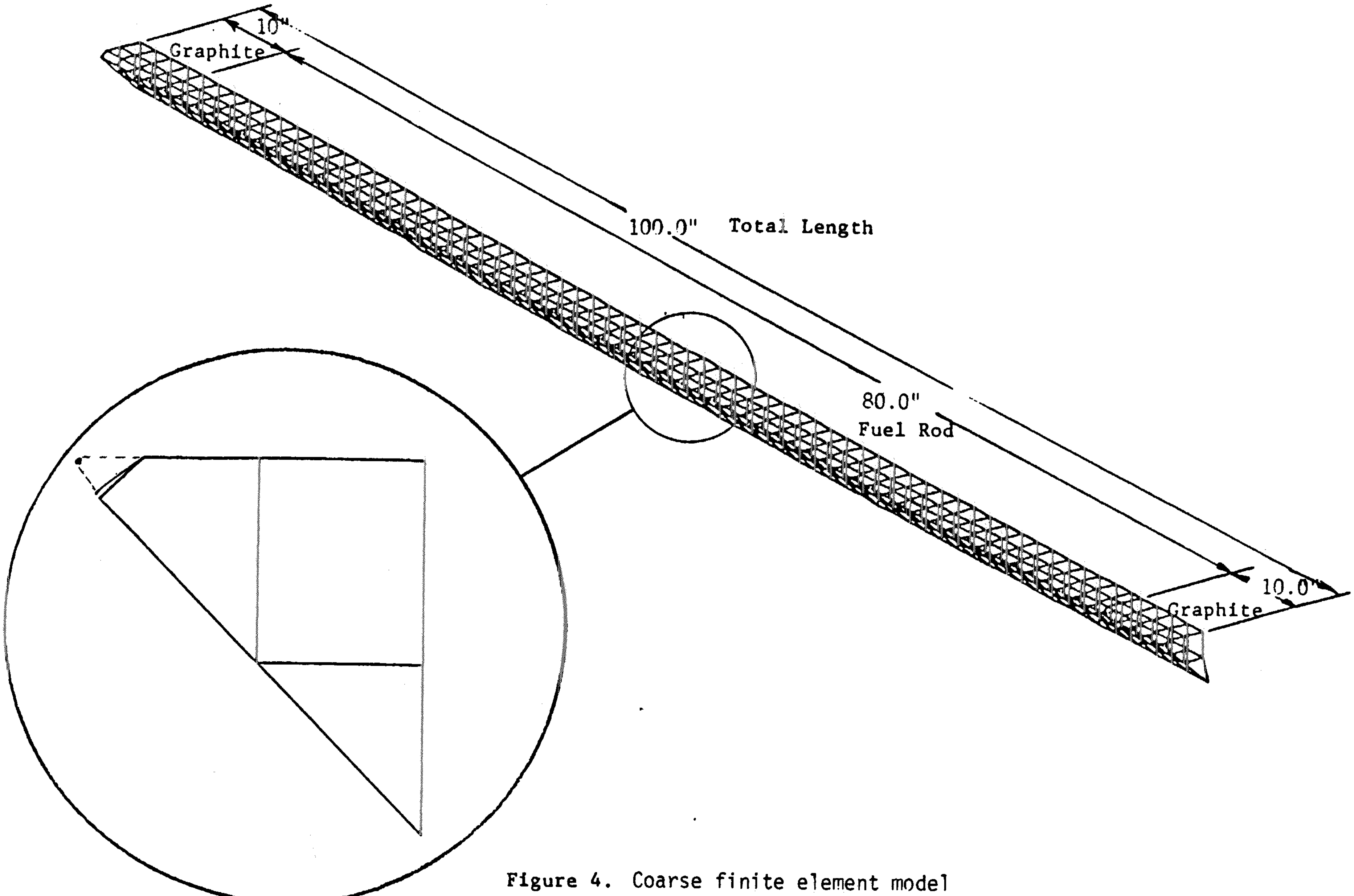


Figure 4. Coarse finite element model

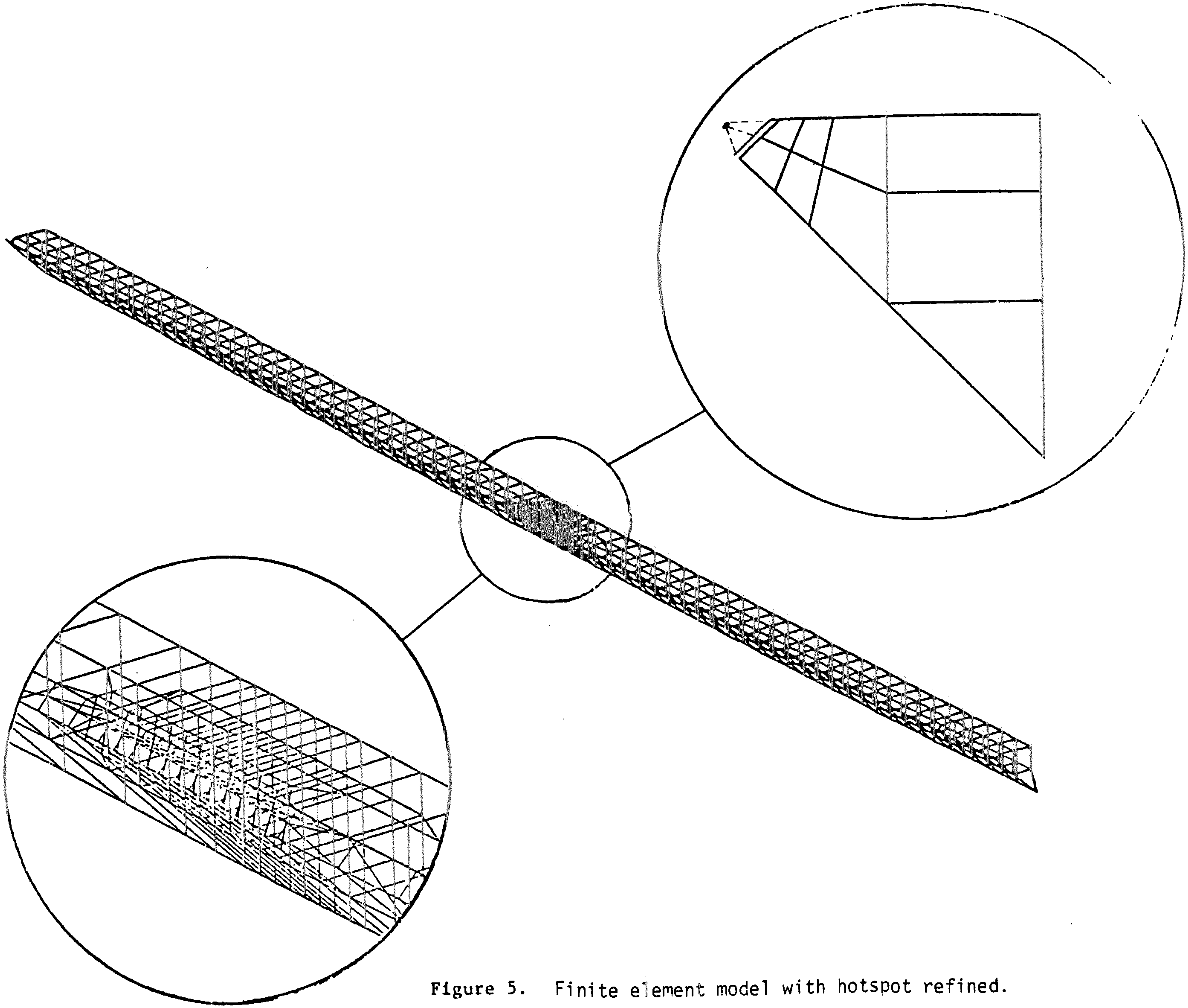


Figure 5. Finite element model with hotspot refined.

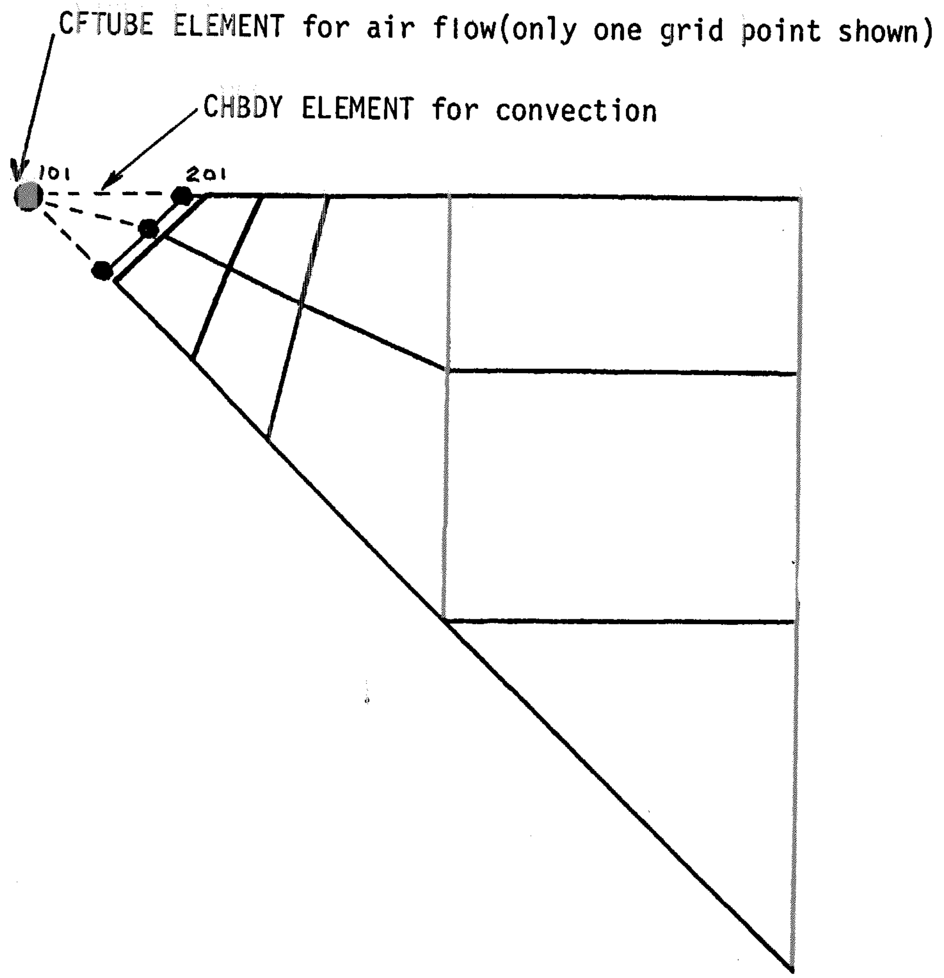


FIGURE 6, Drawing showing relationship between CFTUBE and CHBDY elements for the purpose of simulating forced air convection.

Appendix A Example Nastran Deck

BULK DATA DECK

```
BEGIN BULK
$ CARDS TO SPECIFY INITIAL TEMPERATURE

TEMP    1    1000      70.0                  (Specifies temp at pt. 1000 as 70. deg
                                              Set ID is equal to 1 for selection
                                              in the case control deck)

$ CARDS NECESSARY TO ENFORCE CONSTANT TEMPERATURE AT THE BOUNDARY

CELAS2  El No.    1.0+6    2000      1       (Conductance of 10E+6 set at Pt. 2000)
DAREA        1    2000      1   70.0+6       (Load scaling factor at Pt. 2000)
TABLED1      1    . . .    . . . +TBC1       (Table specifing force in conjunction
+TBC1   -10.0,1.0,  10000.,1.0    ENDT        with the DAREA card)
TLOAD1       1      1     0      0   1       (Identification of this loading)


$ CARDS NECESSARY TO SPECIFY TRANSIENT INTERNAL HEAT SOURCE

QVOL    2    0.5     3000                 (Defines rate of internal heat generation
QVOL    2    0.6     3001                  at the specified elements)
QVOL    2    0.5     3002
TABLED1   2                                                  +TAB1
+TAB1   -10.0,0.0  0.0,0.0  1.0,15.0   1.2,100.0             +TAB2
+TAB2                                                        +TAB3
+TAB3              (Definition of power pulse)               +TAB4
+TAB4   ENDT
TLOAD1       2      2      0      0    2


$ CARDS NECESSARY TO COMBINE THE TWO DYNAMIC LOADS

DLOAD     3    1.0    1.0    1    1.0     2


$ CARDS NECESSARY TO SIMULATE AIR FLOW

CFTUBE    El No.       1    101     102        (Air flow from Pt. 101 to 102)
PFTUBE       1    0.01    10.0     1.0         (Properties of air flow)


$ CARDS NECESSARY TO SIMULATE CONVECTION

CHBDY     EID      1   POINT    101    +HBDY1
+HBDY1    201       (Specifies point heat transfer between Pts 101 and 201)
PHBDY       1      2     1.0               (Specifies area factor)
MAT4        2    0.0004                    (Specifies convective film coefficient)
```

```
$ CARDS FOR SPECIFICATION OF ELEMENTS AND THERMAL PROPERTIES

GRID . . . . . . . . .
CHEXA . . . . . . . .
CPENTA. . . . . . . .
CQUAD4. . . . . . . .
CTRIA3. . . . . . . .
PSHELL. . . . . . . .
PSOLID. . . . . . . .
MAT4. . . . . . . . . (Specifies conductance and heat capacitance of material

$ CARD TO SPECIFY INTEGRATION AND OUTPUT TIME STEPS

TSTEP      1      1     1.0     1          +STEP1
+STEP1            5     0.1     1          +STEP2
+STEP2            6     0.5     2          +STEP3
+STEP3            3     0.1     1
```

CASE CONTROL DECK

```
$ CASE CONTROL DECK FOR THERMAL EXAMPLE
SET 1 = 1 THRU 10
SUBCASE 1
THERMAL = 1          (Output requests)
TEMP(ESTIMATE)=1     (Select initial temperature set)
TSTEP = 1            (Select integration and output time step)
DLOAD = 3            (Select dynamic loading set)
$
$ PLOTTING CARDS FOLLOW
 .
 .
 .
$ END OF CASE CNTL CARDS
```

EXECUTIVE CONTROL DECK

```
ID NASTRAN TRANSIENT HEAT TRANSFER EXAMPLE
SOL 59
TIME 5.
RF ALTER 59D33  (Generates conductance and capacitance matrices)
CEND
```