AD/RHIC/RD-93

RHIC PROJECT

Brookhaven National Laboratory

Calculation of Stresses Induced by RHIC Beam on a Titanium Foil

N. Tsoupas, A. J. Stevens, E. S. Rodger

September 1995

CALCULATION OF STRESSES INDUCED BY RHIC BEAM ON A TITANIUM FOIL

N. Tsoupas, A.J. Stevens, and E.S. Rodger

INTRODUCTION

The RHIC machine will operate with up to 114 full beam buckets which will circulate in each of its two counter-circulating rings. The buckets will be filled with beam bunches originating in the AGS machine, and transferred for injection into RHIC via the AGS-RHIC transfer line (Ref. 1). For Au ions each beam bunch will contain 10^9 ions with each ion having minimum energy of 10.8 GeV/u and maximum energy after acceleration in RHIC of 100 GeV/u. For protons the corresponding numbers are 10^{11} protons with minimum energy of 24 GeV and maximum energy of 250 GeV. (Ref. 2).

During RHIC operation the circulating beam bunches may need to be extracted from the RHIC rings and subsequently dumped onto the beam-dump (Ref. 3). In the initial design of the beam-dump system, extraction of the beam was to be accomplished with a set of magnets which deflect the beam through a Ti foil which separates the main beam-dump from the vacuum of the RHIC rings. A set of "kicker" magnets provide a very fast risetime in the horizontal direction, while slower "sweepers" act in the vertical direction to spread the bunches on the face of the foil. As the beam bunches pass through the foil, they deposit an amount of energy which depends on the beam energy the number of particles in each bunch and the transverse spatial separation of the bunches. This sudden deposition of energy into the foil results in an increase of the foil temperature which in turn causes thermal stresses. A qualitative discussion of dynamic thermal stresses is given elsewhere (Ref. 4). In this report results of the stresses developed on the Ti foil during the time (and few μsec after) the beam passes through the foil are presented.

OUTLINE OF THE CALCULATION METHOD

The calculation of the stresses on the Ti foil were performed using the computer code ANSYS which is installed on a DEC ALPHA 3000-400 workstation. Before the calculations on the Ti foil were performed, the computer code was tested using models of mechanical systems which cannot only be solved numerically using the code, but also have solutions in a closed algebraic form. (Ref. 4).

Static as well as time varying mechanical systems were tested. In static mechanical systems, the agreement between the numerical results (ANSYS code), and the results obtained from the closed form solutions was better than 1%. To test the code in the time varying domain, a mechanical system was selected which was represented analytically by a zero thickness disk (Ref. 4) which was heated instantaneously in the center. This system is solved in 2-dimensions and provides a closed form algebraic solution. The numerical results of this solution was compared to the numerical calculations obtained using the ANSYS code. The comparison showed that the time varying stresses, (at the early times $t <$ 100 nsec after the central region of the disk was heated), as computed by the ANSYS code were 1.6 times larger than the stresses as computed by the closed form solution. This difference is attributed to

the fact that the model used in the code to represent the disk had finite thickness. As a result of the above exercise confidence was developed in the computer code which was used in order to model and solve the mechanical system which will be described in more detail in the following section.

DESCRIPTION OF THE MECHANICAL SYSTEM UNDER STUDY AND OUTLINE OF THE STEPS TAKEN TO CALCULATE THE STRESSES ON THE SYSTEM USING THE ANSYS CODE.

The foil was assumed to be a rectangular box (Fig. 1a) of 2"x2" and thickness 0.01" (z-direction). To generate the computer model of the foil and subsequently calculate the stresses on the foil, the procedure described below was followed.

The procedure followed appears also as an ANSYS input file in Appendix:

a) Generate the foil's front face which is positioned along the z=-0.005" plane (Fig. 1b) . This face consists of 5 areas (numbered 1 through 5 in Fig.1b) with the central area being a rectangle centered at the origin of the coordinate system,and with dimensions of 0.224" in the x-direction and 0.28" in the y-direction. The remaining areas (2 to 5) are equal among each other and surround the central area.

b) Mesh the five areas.[1] In this step subdivision of the five areas of the front foil surface, mentioned above is explained. During beam extraction, when a beam bunch passes through the extraction magnets, the bunch is deflected by an angle proportional to the integrated B-field $\int B(t).dl$. Since the B-field varies with time, each beam bunch will be deflected at a different angle than the previous bunch and the beam bunches will be spread on the Ti foil. In simulating the spread both horizontal "kicking" magnets and vertical "sweepers" were assumed to be present. With the magnets that were assumed to be present the total spread of bunches on the foil would have been 1.7" in length. An attempt to simulate the effect of the entire set of bunches on the foil, would have been an almost prohibitive task in terms of both the required computer memory and the time to obtain the results. For this only the last eight beam bunchs which are the most closely spaced on the Ti foil were chosen to be included in the calculations. These are all the bunches within 2.5 standard deviations of the point of the highest temperature. As a result the largest stresses will be developed in the area where the last eight bunches will pass through the foil and will fit well within the dimensions of the central region of the foil, (Fig. 1b) mentioned above. Based on the eight beam bunch scenario, the central region of the foil was subdivided into 56x56 elements each being a rectangle with corners (nodes) 0.004" apart in the x-direction and 0.005" in the y-direction. These closely spaced nodes of the central region can be justified by looking at the distribution of the beam bunches on the foil. This distribution is of a Gaussian shape with standard

---

[1]The code ANSYS is a finite element computer code which studies and calculates various physical quantities (stresses etc) of mechanical systems by subdividing each system into finite elements over which it calculates the physical quantities. The larger the number of elements the more accurate the results. However, computer memory requirements as well as time constraints call for a compromise between mesh size and accuracy.

deviations σ_x=0.025" and σ_y=0.034" and the temperature generated on the foil will also have the same distribution superimposed on the 70^{0}F background temperature of foil. Thus these closely spaced nodes allow one to assign on the foil a node temperature distribution which will closely resemble the actual Gaussian temperature distribution on the foil. Each of the remaining four areas was subdivided into 28x10 elements, with the 10 subdivisions unequally spaced and with the higher mesh density towards the central region. The more widely-spaced mesh points of the outer four regions can be justified for two reasons. First there are no bunches deposited on the foil at these regions and therefore the initial foil temperature of 70^{0}F will remain uniform in these outer regions. Such a uniform temperature can be well described with a less dense mesh than the mesh used to describe the temperature of the central region. Second, the time interval of 2 μsec, which is needed for the study of the problem, is not long enough for any heat conduction to occur[2]. In this time interval the shock wave created by the sudden increase of the foil's temperature will travel at a distance of ~0.4" from the center of the bunch. At these distances from the center of the bunch, the node density is adequate to describe the stresses with good accuracy.

c) Extrusion of the front face in the z-direction, of the plane at z=0. This extrusion generates five volumes each corresponding to one of the five areas of the plane. The volumes were subdivided into three elements in the z-direction; otherwise their subdivision in the x and y directions is similar to the one corresponding to each of the five surfaces. Thus the 2"x2"x0.01" foil was partitioned into five volumes, with the central volume having the highest node density. Out of symmetry considerations only the z≥0 half of the foil was modeled by the code.

d) Assignment of material properties. The Ti foil can be considered as a single material which has linear properties. The various physical properties which were used for the Ti foil are tabulated below.

Young modulus (Ex)	=	1.65 10^7 lb/in^2
Coef. of linear exp (alpha)	=	5.4 10^{-6} /^{0}F
Material Density (DENS)	=	0.16 lb/in^3
Poisson ratio (ν_{xy})	=	0.3
Tensile Strength	=	1.65 10^3lb/in^2
Yield Stress	=	1.65 10^3lb/in^2
Specific Heat ©	=	5.3 BTU/(lb.^{0}F)
Conductivity (KXX)	=	1.23 BTU/(sec.in.^{0}F)

The Temperature Loads (assign a temperature to each node) were supplied in an input data file. Since it was assumed that there is no heat conduction during the time span the mechanical system was studied,

---

[2]An approximate formula which gives the time for heat conduction in a distance=d in a material with thermal conductivity=k, material density=ρ and specific heat=c is given by $t=d^2/(4k/\rho c)$. For the material under study and for d=0.004" which is the closest distance between nodes, this time is ~36 msec. This time is much longer then the time of 2 μsec required to study the problem. Thus the assumption that there is no heat conduction in the foil during the time all the bunches are being extracted from RHIC and long after, is valid.

the Specific Heat and Conductivity of the material were not used by the code. The rest of the properties were given in a consistent set of units, namely, lb , in , sec and ^{0}F for mass, length, time and temperature respectively.

e) Assignment of boundary conditions: The surfaces at $x=\pm 2$ $y=\pm 2$ of the foil were constrained not to move (zero displacement). Also the surface at $z=0$ was assigned a symmetry boundary condition; the assignment of this boundary condition can be justified under the assumption that the gradient of the temperature along the z-direction is zero. This symmetry boundary condition allows the study only one half of the mechanical system ($z \geq 0$) since the other half ($z < 0$) exhibits identical physical properties.

f) Temperature assignment of the nodes: For a beam bunch to traverse the foil, a time of ~10 nsec is required. During this time the temperature of the foil increases locally at the region where the beam bunch and the foil intersect. This temperature increase is proportional to the energy deposited from the beam bunch on the foil. A time interval of ~12 μsec elapses until all the beam bunches of RHIC pass through the foil. During this time the heat flow from one node to a neighboring node which is located a distance of 0.004" away, is practically zero.[2] Therefore it is assumed that the initial temperature distribution on the foil remains the same during the time the system is being studied. The intensity of each beam bunch projected on a plane normal to the beam direction yields a Gaussian distribution with standard deviations σ_x=0.025" and σ_y=0.034". As a consequence the temperature distribution on the foil is also Gaussian with the same standard deviations as the beam distribution. It is assumed that there is no lateral movement of the beam on the foil during the time the beam bunch passes through the foil. However the transverse displacement of the next bunch which meets the foil 220 nsec later[3] is taken into account. Since there is a considerable overlap among the eight bunches on the foil, a computer program was written to calculate the temperature as a function of location on the foil. The output of this program was used as an input in the ANSYS code in order to assign temperatures at any given node of the foil. The nodes which were not irradiated by the beam were assigned a temperature of 70 ^{0}F. The table below shows the maximum temperature attained just after each bunch passed through the foil.

Bunch #	Temp (^{0}F)
1	220
2	353
3	441
4	508
5	575
6	632
7	665
8	675

g) Calculation Procedure: The actual ANSYS calculations for the stresses as a function of time begin by assigning a uniform temperature of 70 ^{0}F on the foil. This is a step required (at t=0) for dynamic

---

[3]In this simulation it was assumed that the number of bunches circulating in each RHIC ring is 57. Thus each bunch is 220 nsec away from the following one.

(time varying) type of calculations. When the calculations for this step are completed, the first bunch is delivered; in the nomenclature of the ANSYS code this is termed "Applying Temperature Loads" to the nodes of the system under study. The Temperature Loads are applied through an input data file as described in the previous section. The next bunch follows after 220 nsec. During the 220 nsec time interval the computer calculates and stores into output data files the results of the various stresses, at time intervals of 20 nsec. The time interval of 20 nsec was chosen so as not to skip any significant variations of the stresses as a function of time. Although the results are stored at 20 nsec time intervals, the computer calculates (but does not store into output data files) the stress results at smaller time intervals (substeps) by using the "automatic time stepping"procedure of the code which is based on convergence criteria.

RESULTS

The stress results are stored in time steps of 20 nsec for the time span of 1900 nsec. Using the results files (FILE.RST) of any of the time steps the computer calculates the nodal stresses S_x, S_y, S_z, principal stresses S_1, S_2, S_3, and the Equivalent Stress[4] at each node of the model. The code has the feature to display each kind of stress on the terminal; from the display terminal the maximum and minimum of each kind of stress is recorded and plotted as a function of time (see Fig.2)[5]. The sudden change in the S_x, S_y, S_z stresses in Fig. 2, indicates the arrival of a new beam bunch. The equivalent stress is a measure of the shearing action that takes place in the material, and many textbooks (ref. 5) recommend that this stress can be used to determine whether the mechanical system can withstand the particular stresses. This test can be accomplished by comparing the equivalent stress to a limiting stress (Yield Stress) of the material. The effect of the mesh density on the results was studied, by increasing the mesh density of the model in the x-y plane by 20% and the computer calculations were repeated; the results of the stresses changed by less than 2%. Subsequently, the number of mesh points in the z-direction was doubled and the calculations were repeated for a single bunch and up to 160 nsec. From the above test, it was found that the equivalent stress (plotted as a function of time in Fig. 3) attains the same maximum for either three or six subdivisions along the z-direction, with the equivalent stress of the high density mesh varying at a higher frequency. The study of the effect of the thickness on the stresses produced equivalent results as in the comparison of the low and high density meshes. Fig. 4 shows the equivalent stress for a single bunch v. time for thin (0.01") and thick (0.02") foil and either foil subdivided into three elements along the z-direction. The results show the thicker foil developing stresses later times but this is likely, simply a consequence of the mesh dependence mentioned above.

CONCLUSIONS

The maximum equivalent stress in Fig. 2 of 124 kpsi is to be compared to the tensile strength

---

[4]The Equivalent Stress is defined as:

$$S_{eq}=[(S_1-S_2)^2+(S_2-S_3)^2+(S_3-S_1)^2]^{1/2}$$

S_1, S_2, S_3 are the principal stresses of the node.

[5]Positive stress corresponds to tension; Negative to compression.

of Ti-6Al-4V (aged) of 165 kpsi. Although this may well be satisfactory, there is no margin to allow for failure of any of the extraction magnet modules. For this reason, a beam-dump system in which the first element encountered by the extracted beam is a carbon composite, rather than a metallic window, is currently under consideration.

REFERENCES

Ref. 1 J. Claus and H. Foelsche, Beam Transfer from AGS to RHIC, AD/RHIC-47

Ref. 2 RHIC Design Manual, May 1994

Ref. 3 A. J. Stevens, "Maximum Energy Deposition Densities in the Internal Dump", AD/RHIC/RD-41, June,1992

Ref. 4 P. Sievers, "Elastic Stress Waves in Matter due to Rapid Heating by Intense High-Energy Particle Beam", CERN LAB.II/BT/74-2,1974

Ref. 5 I. Shames, Introduction to Solid Mechanics, Apprentices Hall

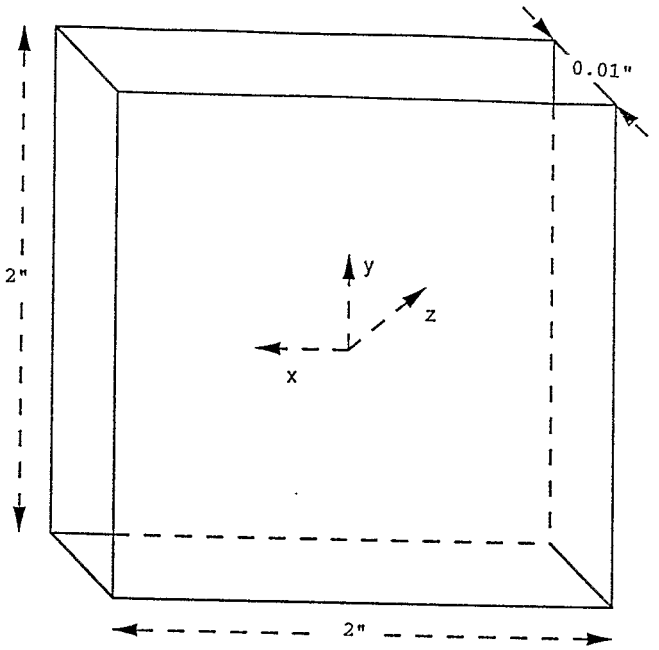

Fig. 1a

Fig. 1a Schematic diagram of the Ti Foil geometry with its dimensions.

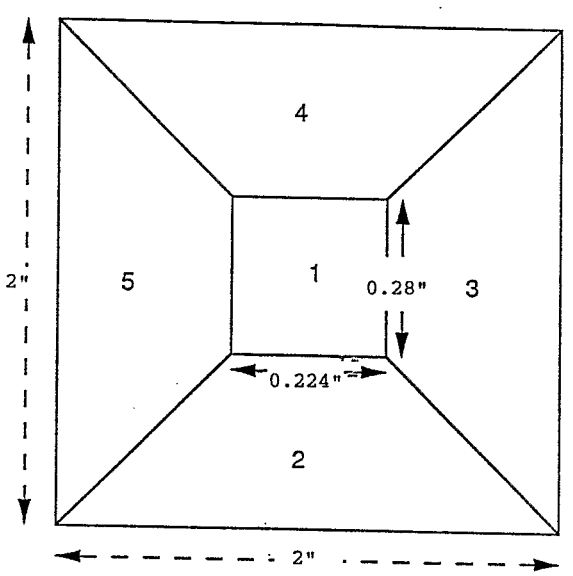

Fig. 1b

Fig. 1b Partition of the front surface of the foil into five regions.

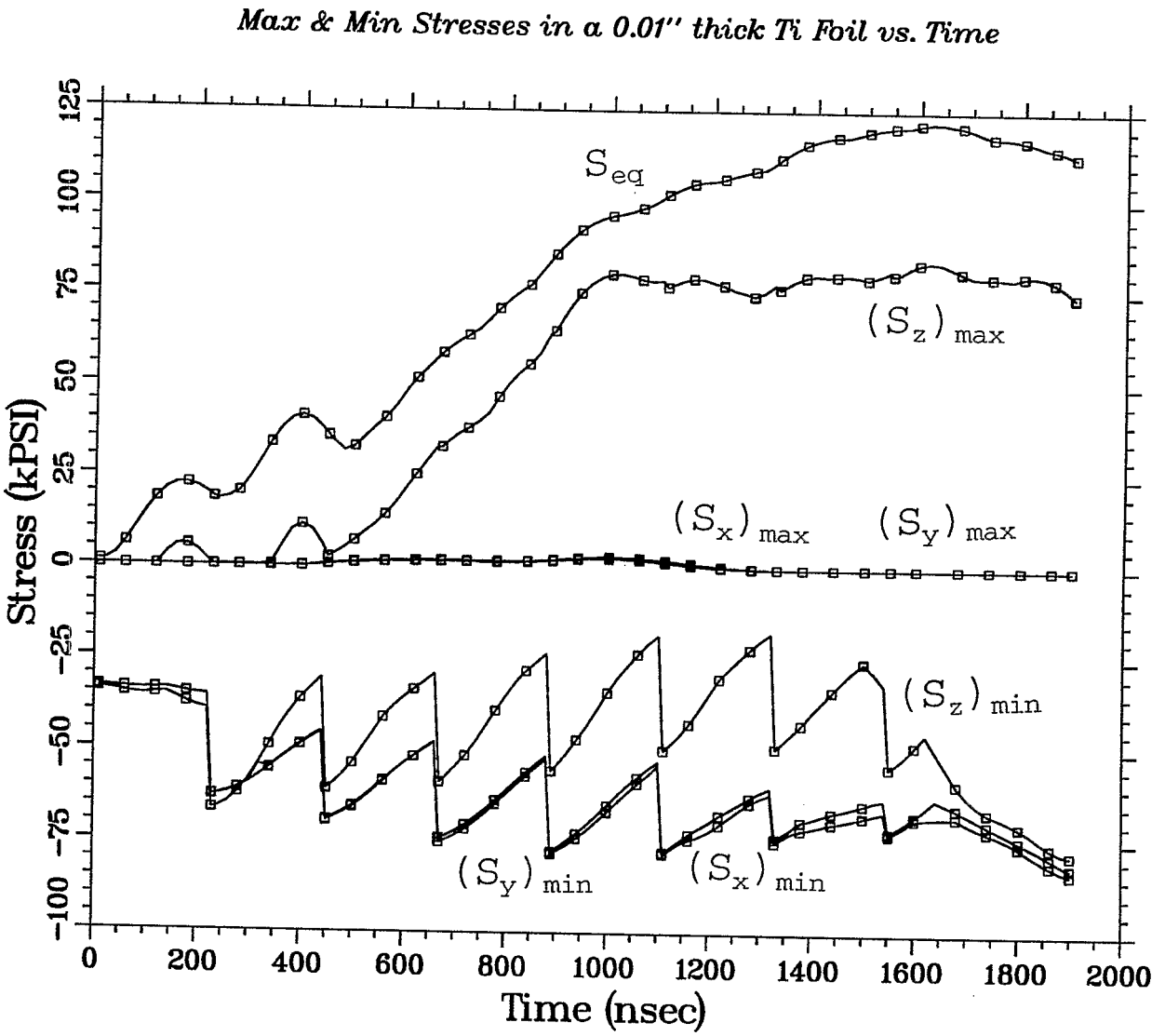

Fig. 2 The maximum and minimum stressses on a 0.01" thick foil as a function of time. The sudden change of the stresses indicates the arrival of a beam bunch.

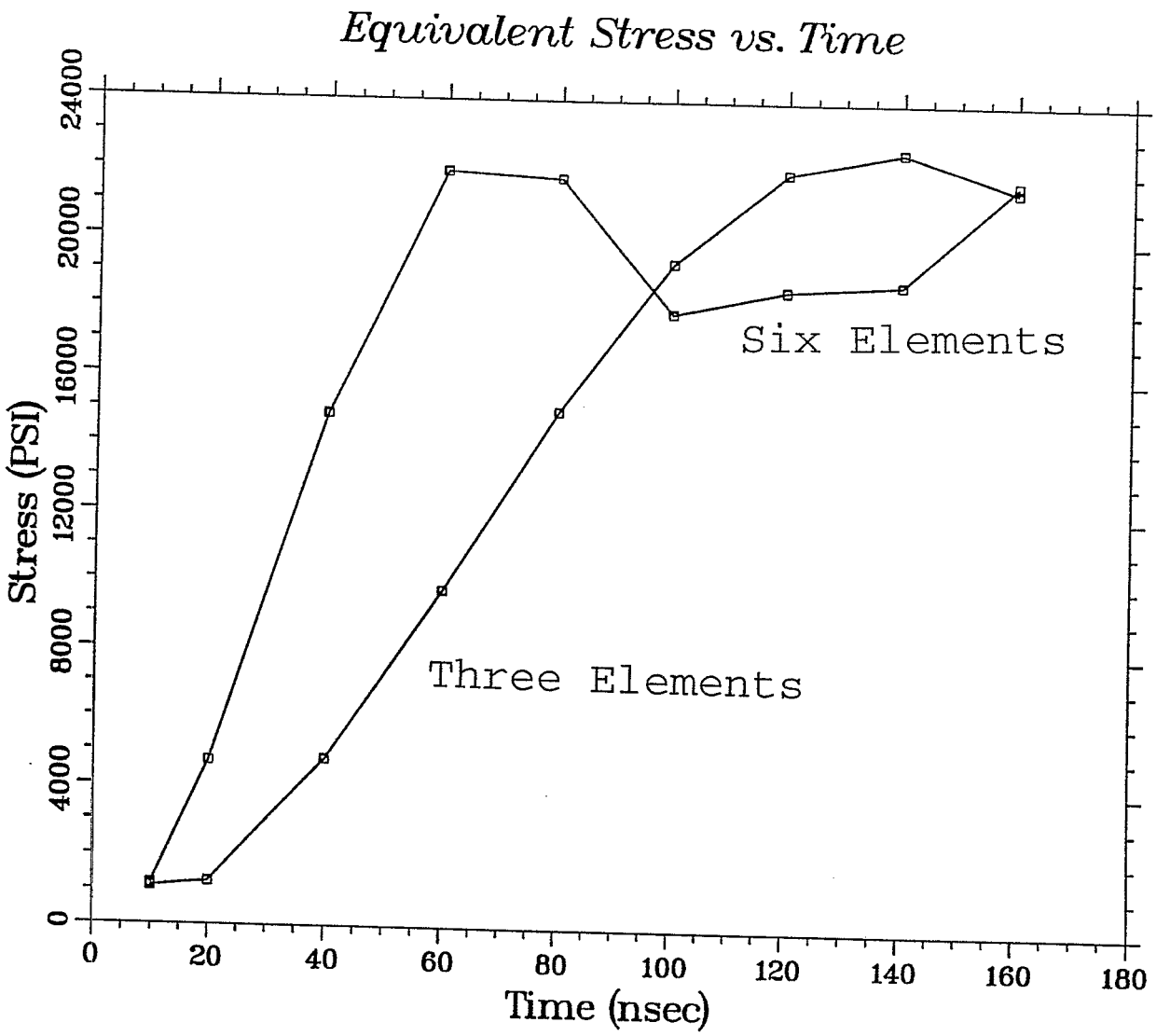

Fig. 3 Maximum equivalent stress as a function of time for a 0.01" thick Ti foil subdivided into three and seven elements.

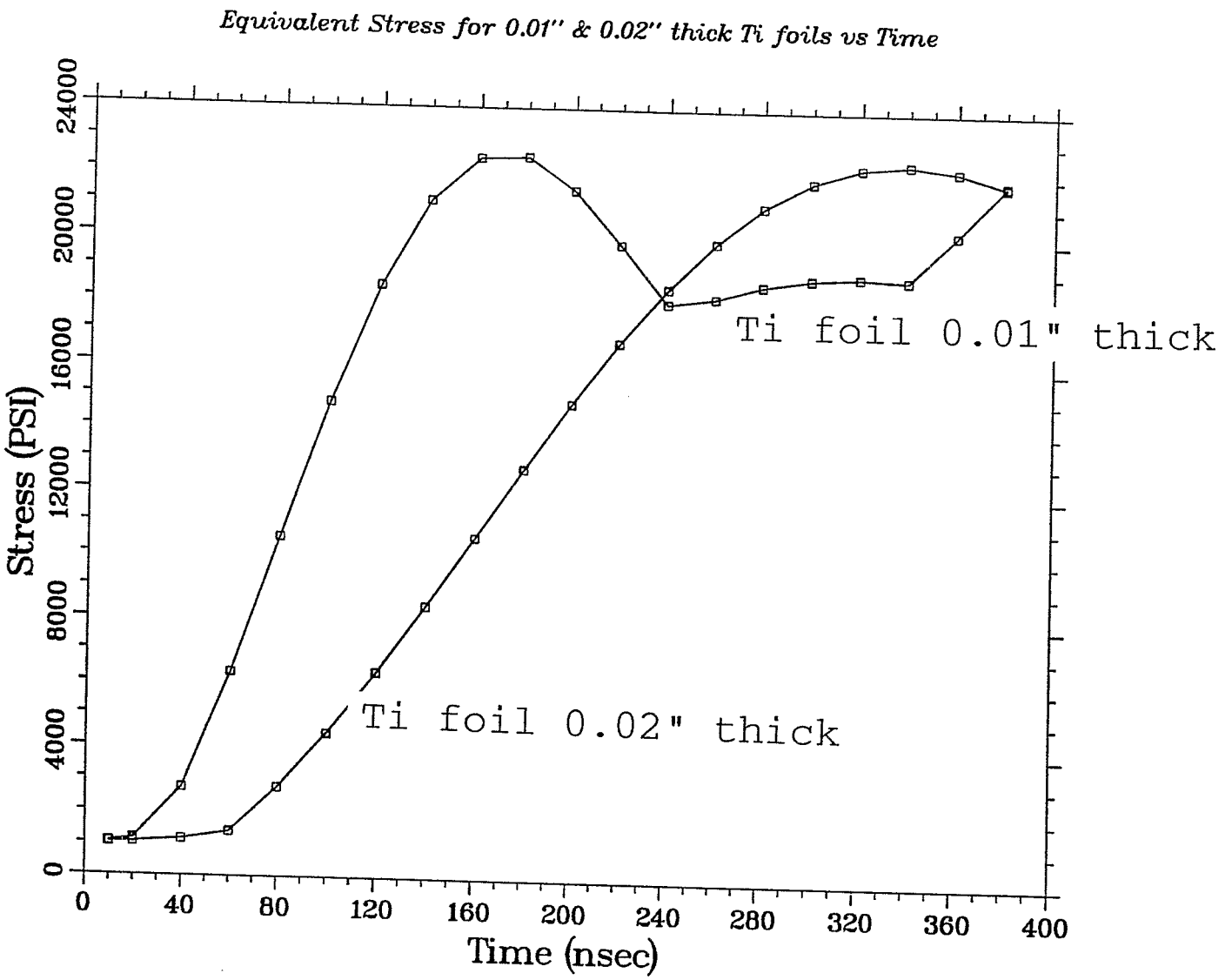

Fig. 4 Maximum equivalent stress as a function of time for a 0.01" and a 0.02" thick Ti foil. In both cases the number of subdivisions of the foils thickness was three.

APPENDIX

```
/FILE,XYS
/SHOW,X11
/MENU,ON
/UNITS,BIN
/TITLE, FOIL X=1" Y=1" NON SYMM GAUSSIAN
/PREP7
MP,EX,1,1.65E7
MP,ALPX,1,5.4E-6
MP,NUXY,1,0.3
MP,DENS,1,0.16
MP,C,1,0.135
MP,KXX,1,1.23E-5
ET,1,45
K,1,-0.112,-0.140,0.0
K,2, 0.112,-0.140,0.0
K,3, 0.112, 0.140,0.0
K,4,-0.112, 0.140,0.0
K,5,-1.000,-1.000,0.0
K,6, 1.000,-1.000,0.0
K,7, 1.000, 1.000,0.0
K,8,-1.000, 1.000,0.0
K,101,-1.0,-1.0,0.005
A,1,2,3,4
A,5,6,2,1
A,2,6,7,3
A,4,3,7,8
A,5,1,4,8
LESIZE,1,,,28
LESIZE,2,,,28
LESIZE,3,,,28
LESIZE,4,,,28
LESIZE,5,,,28
LESIZE,8,,,28
LESIZE,10,,,28
LESIZE,12,,,28
LESIZE,7,,,10,20
LESIZE,6,,,10,0.05
LESIZE,9,,,10,0.05
LESIZE,11,,,10,0.05
L,5,101          ! LINE CONNECTING POINTS 1 & 101
LESIZE,13,,,3      ! 3 DIVISIONS ALONG LINE(1 & 101) Z-DIR
```

```
VDRAG,1,2,3,4,5,,13
VMESH,ALL
FINISH
SAVE
/SOLU
ANTYPE,TRANS
TIMINT,OFF
AUTOTS,ON
NSUB,1
TUNIF,70
TREF,70
ALPHAD=0.0
BETAD=3.18E-10
NSEL,S,LOC,X,0.999,1.001
D,ALL,UX,0
D,ALL,UY,0
D,ALL,UZ,0
NSEL,S,LOC,X,-0.999,-1.001
D,ALL,UX,0
D,ALL,UY,0
D,ALL,UZ,0
NSEL,S,LOC,Y,0.999,1.001
D,ALL,UX,0
D,ALL,UY,0
D,ALL,UZ,0
NSEL,S,LOC,Y,-0.999,-1.001
D,ALL,UX,0
D,ALL,UY,0
D,ALL,UZ,0
DA,1,SYMM
DA,2,SYMM
DA,3,SYMM
DA,4,SYMM
DA,5,SYMM
NSEL,ALL
TIME,0.5E-9
LSWRITE,1
! INPUT TEMPERATURE DISTR.  OF FIRST Bunch
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.1400,-0.1292
BF,ALL,TEMP,   0.705E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.1292,-0.1185
BF,ALL,TEMP,   0.709E+02
```

```
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.1185,-0.1077
BF,ALL,TEMP,   0.714E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.1077,-0.0969
BF,ALL,TEMP,   0.719E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.0969,-0.0862
BF,ALL,TEMP,   0.725E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.0862,-0.0754
BF,ALL,TEMP,   0.729E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.0754,-0.0646
BF,ALL,TEMP,   0.731E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.0646,-0.0538
BF,ALL,TEMP,   0.730E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.0538,-0.0431
BF,ALL,TEMP,   0.726E+02
NSEL,S,LOC,X,-0.0431,-0.0345
NSEL,R,LOC,Y,-0.0431,-0.0323
.............................
.............................
```

At this point the "Temperature load" input list
is interupted due to its length.

```
NSEL,ALL
/PBF,TEMP,,1
NSUB,2000
TIME,10E-9
TIMINT,ON
KBC,1
! The various times to Calculate the Stresses
! are specified bellow.
LSWRITE,2
TIME,20E-9
LSWRITE,3
TIME,40E-9
LSWRITE,4
TIME,60E-9
LSWRITE,5
```

```
TIME,80E-9
LSWRITE,6
TIME,100E-9
LSWRITE,7
TIME,120E-9
LSWRITE,8
TIME,140E-9
LSWRITE,9
TIME,160E-9
LSWRITE,10
TIME,180E-9
LSWRITE,11
TIME,200E-9
LSWRITE,12
TIME,220E-9
LSWRITE,13

! At this point of time another list of "Temperature Loads"
! is read from the computer.  This list corresponds to the
! temperatures of the Ti foil due to the first and second
! beam bunches.

.................................
.................................

! The various times to Calculate the Stresses
! are specified bellow.

.................................
```