2

Idaho National Engineering Laboratory

Managed by the U.S. Department of Energy

EG&G Idaho

Work performed under DOE Contract No. DE-AC07-76ID01570

EGG-EAST-8680
August 1989

INFORMAL REPORT

VERIFICATION AND BENCHMARKING OF ABAQUS AND PATRAN FOR HEAT TRANSFER APPLICATIONS

G. L. Hawkes

EGG-EAST--8680

DE90 010928

VERIFICATION AND BENCHMARKING OF ABAQUS AND PATRAN FOR HEAT TRANSFER APPLICATIONS

G. L. Hawkes

August 1989

Idaho National Engineering Laboratory
EG&G Idaho, Inc.
P. O. Box 1625
Idaho Falls, Idaho 83415

Prepared for the U. S. Department of Energy
Idaho Operations Office
Under DOE Contract No. DE-AC07-76ID01570

SUMMARY

This report contains the verification and benchmarking performed at the Idaho National Engineering Laboratory (INEL) on ABAQUS and PATRAN for heat transfer calculations. ABAQUS and PATRAN were used to perform the thermal analysis of the Advanced Test Reactor (ATR) loop 2A piping components. These codes are commercially available and are used extensively within the industry to solve structural and heat transfer problems using finite element methods. They have been extensively verified and documented by the companies developing and providing them. This report documents work performed by EG&G Idaho Inc. to further verify that ABAQUS and PATRAN properly solve a variety of heat transfer problems and that performance on the INEL computers was the same as on the developer's computers.

Methods employed to verify ABAQUS and PATRAN are presented within this report. Heat transfer problems with known exact solutions are compared to ABAQUS results. Computer output from the ABAQUS introductory workbook was compared to calculations performed on INEL computers using the same input deck. The verification of ABAQUS and PATRAN at INEL has shown the codes applicable to solve heat transfer problems encountered at INEL.

CONTENTS

Figures

Tables

VERIFICATION AND BENCHMARKING OF ABAQUS AND PATRAN FOR HEAT TRANSFER APPLICATIONS

1. INTRODUCTION

This report was written as a supplement to the thermal analysis report[1] for the Advanced Test Reactor (ATR) loop upgrade analysis and lifetime extension. As part of the lifetime extention work it was necessary to evaluate the thermal stresses and their affect on piping and component life. A large number of ATR transients produce thermal gradients in the piping components which must be analyzed. The thermal gradients, the number of cycles, and component characteristics are used to calculate the expected lifetime of the component. A heat conduction code was used to calculate and analyze the thermal gradients. The finite element code ABAQUS[2] was used for the thermal analysis of ATR loop piping components for 75 transients. ABAQUS is a general purpose three-dimensional finite element code used in a variety of structural, heat transfer, and dynamic analyses. PATRAN[3] was used to generate the mesh and apply the boundary conditions for ABAQUS. A verification of ABAQUS and PATRAN was necessary to demonstrate that the codes were correctly performing heat transfer calculations for application on ATR loop piping components. This report contains the heat transfer verification and benchmarking performed at the Idaho National Engineering Laboratory (INEL) for ABAQUS and PATRAN.

ABAQUS is a production oriented, general purpose finite element code. It is simple to use and has capabilities for a wide range of nonlinear applications, one of which is the solution of three-dimensional, transient heat conduction, or thermal diffusion, problems. Steady-state solutions are obtained by direct integration of the spatial partial differential equation. Transient solutions are obtained by integrating the temporal/spatial equation with the backward difference operator (modified Crank-Nicholson method).

PATRAN is an open-ended, general purpose, three-dimensional computer aided engineering software system. It includes the capabilities for generating finite element meshes in cartesian, cylindrical, or spherical coordinate systems, using automated command sequences. Element material properties, volumetric heat generation rates for elements, surface heat flux values and convection heat transfer coeffficients for element faces can all be specified. PATRAN was used to generate the finite element models and apply the boundary conditions that were later used as input to ABAQUS. Translation of files from PATRAN to ABAQUS was accomplished with

PATABA to produce the input decks. ABAPAT was used to translate the ABAQUS results to PATRAN for post processing and generation of plots. PATABA and ABAPAT were supplied as part of the PATRAN software. This report documents the compliance of ABAQUS and PATRAN software packages, used for these calculations, with EG&G Quality Manual Section QP-21, Computer Software Configuration Mangement.

Four methods were selected to verify and benchmark ABAQUS and PATRAN, they were: (1) review the ABAQUS Example Problems Manual[4] for heat transfer analyses, (2) compare computer printouts in ABAQUS Introductoery Workbook [5] to INEL computer printouts for the same input deck, (3) compare ABAQUS transient heat transfer finite plate analysis to the exact solution, and (4) compare ABAQUS steady state heat transfer pin fin analysis to the exact solution. PATRAN was verified by checking the finite element mesh which PATRAN generates for the input deck of an ABAQUS run. PATRAN was also verified by visually inspecting the model generated to insure that it described the geometry expected.

Section 2 of this report discusses the method of verifying ABAQUS and PATRAN and the results obtained during this analysis. Conclusions are presented in Section 3.

2. METHOD OF VERIFICATION

This section discusses the method of verifying ABAQUS and PATRAN. Example problems reviewed are discussed in Section 2.1, with ABAQUS benchmarking discussed in Section 2.2. Problem setup, assumptions and results are presented in Section 2.3 for the finite plate heat transfer analysis. Section 2.4 presents the problem setup, assumptions and results for pin fin steady state heat transfer analyisis. Section 2.5 discusses the verification of PATRAN.

2.1 EXAMPLE PROBLEMS

Heat transfer problems in the ABAQUS Example Problems Manual were reviewed to understand existing code verification . There were two thermal analysis verification problems presented in this manual. These problems were: (1) Freezing of a Square Solid, The Two - Dimensional Stefan Problem (section 3.2.7 in Reference 4), and (2) Quenching of an Infinite Plate (section 3.2.11 in Reference 4). These two problems were presented with an exact solution, and show that the ABAQUS solution and the exact solution are the same. In addition Hibbitt, Karlsson & Sorensen, Inc. (developers of ABAQUS) have run numerous heat transfer analyses in their checking of ABAQUS, that are not formally documented.

2.2 BENCHMARKING OF ABAQUS ON INEL COMPUTERS

A duplicate input deck was created for Course Example 8 in the ABAQUS Introductory Workbook and run on the INEL computers. This verified that the INEL solution was the same as documented in the ABAQUS Introductory Workbook. A transient heat transfer problem with a user subroutine to calculate the heat transfer coefficient (HTC) and fluid temperature was used to verify the user subroutine capabilities. The results in the workbook and the results from the INEL computers were exactly the same and are documented in the design file stored in the E&ST Quality files.

2.3 TRANSIENT HEAT TRANSFER THROUGH FINITE PLATE

A transient finite plate heat transfer problem was analyzed using ABAQUS. Three-dimensional heat transfer elements were used to model the one-dimensional finite plate problem. The three-dimensional elements were used to verify the three-dimensional elements in ABAQUS since many applications may be reduced to a three-dimensional solution. Figure 1 shows a finite plate initially at a uniform temperature of 0.0 °C. A step change in temperature at 0.0 s initiates the transient. Heat is applied to the left side with the right side adiabatic. The nodalization and thermal properties are displayed in Figure 1. The plate is modeled as 20, three-dimensional finite elements, with each element 1 x 1 x 1 cm. An initial time step of one second was used during the analysis. The maximum temperature change of 5°C per time step was used as an upper limit to restrict the analysis from taking too large of time steps. Almost all of the transient analyses for ATR loop piping components by ABAQUS used this automatic time step incrementation technique. For this analysis ABAQUS increased the time step until the maximum temperature change in any node of the model did not exceed 5°C in the time step. At the end of the analysis ABAQUS time steps were 1000 s since the temperature was not changing very fast. Steady state was reached when the temperatures varied less than 0.001°C/s. ABAQUS stopped the calculations when this value was reached.

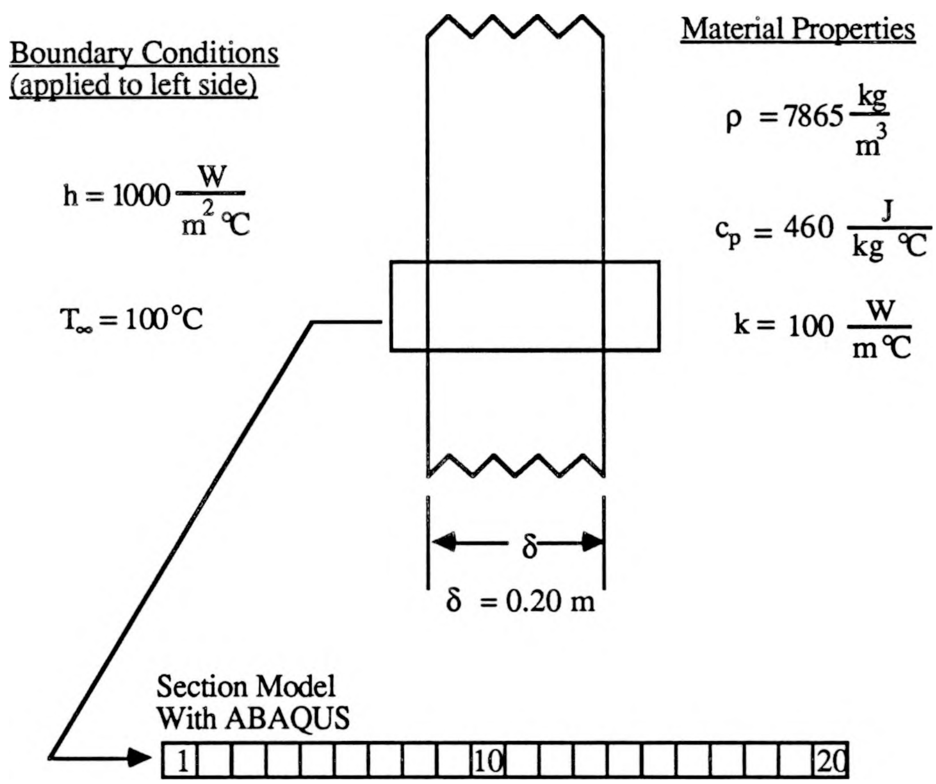


Figure 1. Transient heat transfer through a finite plate.

Table 1 compares the ABAQUS temperature profiles with the temperature response charts.[6] Fo is dimensionless time described as: Fo = $\alpha\theta / \delta^2$, where α is the thermal diffusivity, θ is real time, and δ is the plate thickness. X is the dimensionless position described as $X = x/\delta$. The Biot number described as $Bi = h\delta/k$ was constant at 2.0, where h is the heat transfer coefficient, and k is the thermal conductivity. The dimensionless temperature described as $T = (t - t_i) / (t_a - t_i)$ was calculated at various time steps for different depths of the plate. The variable t is the actual temperature, t_i is the initial temperature, and t_a is the ambient or infinite temperature. Table 1 shows the temperature gradient from the applied heat source to the right boundary. The ABAQUS predictions are in excellent agreement with the temperature response charts (TRC) with only minor variations at any time step and position.

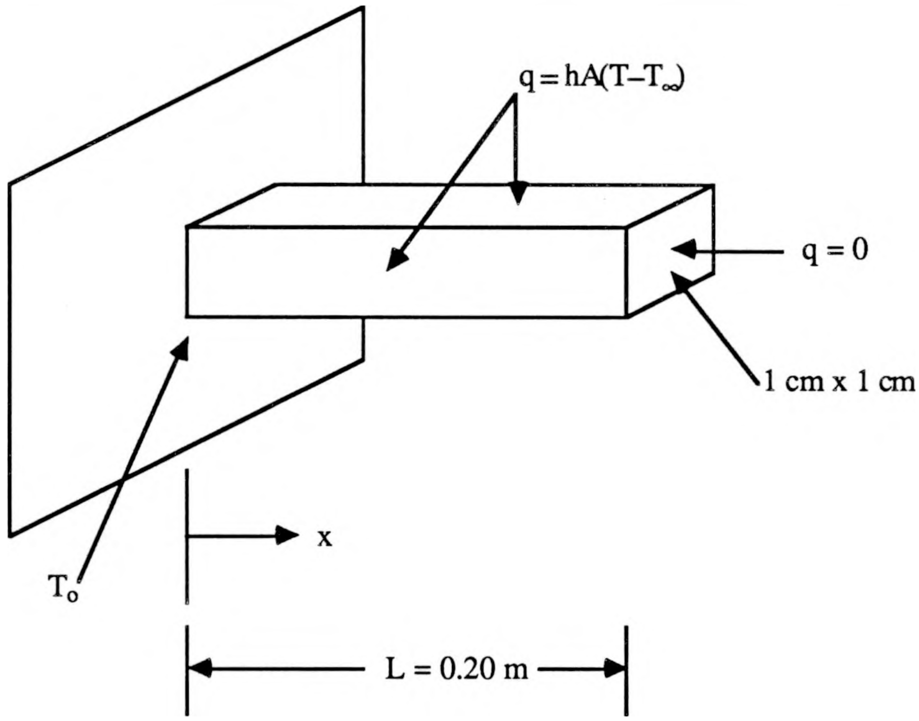
Table 1. Comparison of ABAQUS temperature profiles and Temperature Response Charts[a] (Bi = 2.0)

	$T = (t - t_i) / (t_a - t_i)$									
	X = 0.0		X = 0.1		X = 0.4		X = 0.7		X = 1.0	
Fo	ABAQUS	TRC	ABAQUS	TRC	ABAQUS	TRC	ABAQUS	TRC	ABAQUS	TRC
0.03	0.28	0.30	0.16	0.17	0.02	0.01	0.00	0.00	0.00	0.00
0.19	0.53	0.54	0.44	0.45	0.23	0.24	0.11	0.12	0.07	0.08
0.41	0.65	0.66	0.58	0.58	0.41	0.41	0.30	0.30	0.26	0.26
0.59	0.71	0.72	0.66	0.67	0.52	0.53	0.43	0.45	0.39	0.41
0.76	0.76	0.77	0.72	0.73	0.60	0.60	0.53	0.53	0.50	0.51
1.07	0.83	0.84	0.80	0.80	0.72	0.73	0.67	0.68	0.65	0.65
2.13	0.95	0.95	0.94	0.94	0.91	0.90	0.89	0.90	0.89	0.89

a Values were extrapolated from charts by hand.

2.4 Steady State Heat Transfer Through Pin Fin

A steady state heat transfer problem analyzing a pin fin was simulated using ABAQUS. The same model was used for the pin fin as was used for the finite plate. Boundary conditions were changed as is shown in Figure 2. Three-dimensional elements were used to model the one-dimensional problem. The slenderness ratio is high enough to model the pin fin as one-dimensional. This was done to verify that the three-dimensional heat transfer elements would correctly solve the problem. Figure 2 shows a pin fin with a fixed base temperature of 0.0 °C. The fin is subjected to an infinite fluid temperature of 200.0 °C with a heat transfer coefficient



<u>Boundary Conditions</u>

$h = 10.0 \frac{W}{m^2\,°C}$

$T_\infty = 200\ °C$

$T_o = 0.0\ °C$ = Base Temperature

<u>Material Properties</u>

$k = 19 \frac{W}{m\,°C}$

$c_p = 460 \frac{J}{kg\,°C}$

$\rho = 7865 \frac{kg}{m^3}$

Figure 2. Steady state heat transfer through pin fin with fixed base temperature.

applied to all sides except the end of 10.0 $W/m^2°C$. The analysis of the pin fin by ABAQUS was modeled as a steady state heat transfer calculation. A steady state solution was reached in one iteration by ABAQUS.

Table 2 compares the ABAQUS solution and the exact solution.[7] The ABAQUS results and the exact solution are in excellent agreement. The largest discrepency between the ABAQUS results and the exact solution is less than 0.1°C.

Table 2. Comparison of exact solution and ABAQUS solution for steady state heat transfer analysis of pin fin

x (m)	ABAQUS Temperature °C	Exact Temperature °C
0.00	0.00	0.00
0.02	50.06	50.02
0.04	87.39	87.32
0.06	115.10	115.07
0.08	135.70	135.62
0.10	150.80	150.70
0.12	161.70	161.61
0.14	169.30	169.26
0.16	174.40	174.30
0.18	177.20	177.17
0.20	178.20	178.10

2.5 PATRAN

PATRAN was used to generate the mesh and apply the boundary conditions for the model used in the finite plate and pin fin analysis in this report. PATRAN was also used to generate three models used in ATR thermal analysis of loop 2A, they are: 1 x 1 x 1-in. standard tee, 2 x 2 x 1-in. thermal mixing tee, and 3 x 2 reducer. These models were used 75 times during the thermal analysis of loop 2A with excellent performance. PATRAN was verified by inspecting the nodal coordinates, element connectivity, element sets, and boundary conditions which are translated to an ABAQUS input deck. PATRAN was also used for post-processing the ABAQUS results with color graphics. This was also a very valuable way of verifying the PATRAN mesh and boundary conditions.

3. CONCLUSIONS

ABAQUS and PATRAN have been adequately verified and benchmarked giving a high degree of confidence in solving typical INEL heat conduction problems. Exact solutions were in agreement with the ABAQUS results for the analyses found in the ABAQUS Example Problems Manual. ABAQUS was in excellent agreement with the exact solution for the transient finite plate and steady state pin fin heat transfer analyses documented in this report. The use of ABAQUS and PATRAN on the computers at the INEL produced the same results as the printed output in the ABAQUS Introductory Workbook. It was concluded that ABAQUS and PATRAN can be used to solve a wide variety of heat transfer problems encountered at the INEL.

6. REFERENCES

1. N. E. Pace and G. L. Hawkes, Thermal Analysis of ATR 2A Loop Piping Components, (to be published)

2. Hibbett, Karlsson & Sorensen Inc., ABAQUS User's Manual, Version 4.7, Hibbett, Karlsson & Sorensen Inc., 1988.

3. PDA Engineering, PATRAN Plus User Manual, version 2.3, PDA Engineering, July 1988.

4. Hibbett, Karlsson & Sorensen Inc., ABAQUS Example Problems Manual, version 4.7, Hibbett, Karlsson & Sorensen Inc., 1988.

5. Hibbett, Karlsson & Sorensen Inc., ABAQUS Introductory Workbook, version 4.7, Hibbett, Karlsson & Sorensen Inc., 1988.

6. P. J. Schneider, Temperature Response Charts, New York: John Wiley and Sons, Inc., 1963.

7. J. P. Holman, Heat Transfer, 5th Edition, New York: McGraw-Hill Book Company, Inc., 1981, p. 40.