PVP – Vol. 143

Advanced Topics in Finite Element Analysis

edited by
J. F. CORY, JR.
J. L. GORDON

PVP - Vol. 143

Advanced Topics in Finite Element Analysis

presented at

THE 1988 ASME PRESSURE VESSELS AND PIPING CONFERENCE
PITTSBURGH, PENNSYLVANIA
JUNE 19–23, 1988

sponsored by

THE COMPUTER TECHNOLOGY COMMITTEE OF
THE PRESSURE VESSELS AND PIPING DIVISION, ASME

edited by

JAMES F. CORY, JR.
STRUCTURAL DYNAMICS RESEARCH CORPORATION
MILFORD, OHIO

JERRY L. GORDON
WESTINGHOUSE ELECTRIC CORPORATION
BETTIS ATOMIC POWER LABORATORY
PITTSBURGH, PENNSYLVANIA

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
United Engineering Center 345 East 47th Street New York, N.Y. 10017

Library of Congress Catalog Number 88-71183

FOREWORD

Without question, the digital computer has had a significant impact on the engineering community. Larger and faster computers have made possible such developments as the finite element method as a practical and invaluable tool in the area of structural analysis. The pressure vessels and piping industry has always prided itself at utilizing state-of-the-art hardware and software as well as being involved in the development of this software for the performance evaluation and quality assurance of its equipment.

The goal of the Pressure Vessels and Piping Division's Computer Technology Committee is to present the high quality work representing the most recent developments in engineering computational methods, linear and nonlinear finite element analysis techniques and applications, graphics and the use of computers in manufacturing and design.

All of the papers contained within this volume were presented at the Pressure Vessels and Piping Division Conference of the American Society of Mechanical Engineers held in Pittsburgh, Pennsylvania, during the week of June 19-24, 1988. It is our hope that this volume will bring together, in one place, current developments and applications in Advanced Topics in Finite Element Analysis. The technical paper sessions covered by this volume include:

- MCAE for Design and Manufacturing
- Nonlinear Finite Element Analysis
- Graphics and Animation for the Finite Element Method

On behalf of the Computer Technology Committee and the Conference staff, we wish to thank all of the authors and speakers for their invaluable contributions. The contributions of David E. Dietrich, Joop C. Nagtegaal and Richard S. Gallagher, who developed the sessions for which the Computer Technology Committee had responsibility, are gratefully acknowledged. Without all of their conscientious voluntary efforts, this volume and this conference would not have been possible.

James F. Cory, Jr.
Jerry L. Gordon

CONTENTS

TEMPERATURE PREDICTION CALCULATION FOR WELDING USING A PERSONAL COMPUTER

L. A. Van Gulick and J. E. Richardson
Department of Mechanical Engineering
Lafayette College
Easton, Pennsylvania

ABSTRACT

16-bit personal computers suitable for moderately computationally intensive analyses have been available for several years. Associated nonlinear finite element method (FEM) computer codes appear to offer the features necessary for predicting welding temperatures. An investigation of the use of a 16-bit personal computer and a nonlinear FEM computer code for this application is presented. Predicted and measured temperatures for a simple welding experiment are compared and the feasibility of using personal computers for welding temperature prediction discussed.

INTRODUCTION

Welding processes use intense local heating to form a region of molten material, the weld pool, in parts being joined. The bond between the parts is formed by solidification of the weld pool.

Mechanical properties of the solidified weld pool material are strongly influenced by temperatures experienced during welding. Material in the adjacent heat affected zone is also heated enough to significantly affect its mechanical properties. Properties in the solidified weld pool and heat affected zone may be quite different after welding from pre-welding properties.

The intense local heating used to form the weld pool produces transient heat affected zone temperatures approaching the material melting point and severe thermal gradients through much of the weld vicinity. Nonuniform expansion and contraction due to the thermal gradients result in large local stresses which frequently lead to plastic material behavior. Residual stresses are left in the welded material following cooling and residual displacements which alter weld geometry are produced.

The design of a satisfactory weld joint can be a challenging task. Enough heat must be applied to form an adequate bond, but unacceptable degradation of material mechanical properties, residual stresses, and changes in weld geometry avoided. Although an empirical design approach based on experience and trial and error is usually followed, recent progress in finite element method analysis of welding processes promises to make analysis-based weld joint design possible.

Accurate prediction of welding temperatures is essential if analysis-based weld joint design is to be accomplished. Temperature-time histories are needed to evaluate post-weld material mechanical properties and to serve as input for stress-displacement computations carried out as part of a sequential analysis procedure.

The physical complexity of welding processes presents a number of computational difficulties to welding temperature prediction. The processes are nonsteady state and require transient analyses beginning at the initiation of heating and continuing until cooling is essentially complete. Weld geometries are almost always three-dimensional, and two-dimensional analyses can only describe them approximately. The large temperature variations which occur during welding make welding analyses inherently nonlinear. Filler metal addition introduces an additional computational difficulty for many welding processes.

Temperatures experienced during welding range from ambient to well above material melting points. They necessitate the treatment of material thermal properties as temperature dependent. Heats of fusion associated with melting and solidification and heats of transformation associated with solid phase changes must be included in analyses. Conduction heat transfer occurs throughout parts being welded. Other important heat transfer mechanisms may include convection through the molten weld pool and radiation, convection, and weld pool vaporization heat losses to surroundings.

Nonlinear finite element method (FEM) computer codes offer features capable of overcoming the analytical difficulties associated with welding temperature prediction (1), (2), (3). Available features include the inclusion in analyses of temperature-dependent material properties, heat of fusion effects, and radiation and convection heat losses. Efficient numerical solution schemes facilitate the transient analyses required.

The computationally intensive nature of finite element method welding temperature prediction had in the past made the use of personal computers for this application impractical. The introduction several years ago, however, of moderately computationally powerful 16-bit personal computers and the subsequent development of associated nonlinear finite element method software have made personal computer welding temperature prediction worth investigating.

This paper describes an investigation of the use of a 16-bit personal computer and a nonlinear FEM code to predict welding temperatures. A simple Gas Tungsten Arc (GTA) plate welding experiment for which well-documented experimental parameters and measured temperatures have been published (4) is considered. Comparisons of measured and predicted temperatures and weld pool decay following flux termination serve to evaluate prediction accuracy. Analytical procedures used are presented. Conclusions as to the feasibility of personal computer welding temperature prediction are discussed.

COMPUTER HARDWARE AND FEM SOFTWARE

Available 16-bit personal computers are capable of carrying out moderately computationally intensive tasks. Their 16-bit microprocessors with associated math coprocessors, single or multiple hard disks, and memory sizes many times larger than those of earlier personal computers, offer substantial computing power. Auxiliary removable cartridge storage devices can expand available disk space and make archival file storage possible. Color graphics displays of analysis input data and computed results are readily interpreted and facilitate effective interactive computing.

An IBM AT Personal Computer was used to predict welding temperatures. Its capability is typical of first generation 80286 processor-based MS-DOS AND PC-DOS personal computers. It was equipped with an 80287 math coprocessor, 640 Kb. of directly addressable memory, and an internal 20 Mb. hard disk. An Iomega Bernoulli Box with two 10 Mb. drives gave removable cartridge auxiliary file storage. Color graphics capability with 640 x 480 pixel resolution, 16 color display, and light source shading was provided by an IBM Professional Graphics Controller and Display.

ANSYS-PC/THERMAL (5), (6) was used in this study. It is a subset of the ANSYS program which provides nonlinear steady state and transient heat transfer analysis capability. Nonlinearities which can be considered in analyses include temperature-dependent material properties, phase change effects associated with melting and/or solidification, and radiation heat flow. Prescribed nodal heat flows and nodal temperatures, internal heat generation, and radiation, conduction, and convection boundary conditions can be modeled.

Transient temperatures are solved for using an implicit direct integration scheme based on a modified Houbolt method. Time step optimization can be used to reduce computation and minimize computer run times.

Color graphics-based preprocessors and postprocessors permit effective interactive use. Model definition is supported by preprocessor node and element plots of model geometry. Viewing distance, view angle, and view window center can be user specified to review local model details. Hidden line plots are possible for three-dimensional models. Postprocessor procedures for sorting and selecting results, plotting temperature-time histories, and plotting temperature contours assist the user in results interpretation and evaluation.

The PC/THERMAL preprocessor uses simple direct node and element generation procedures to define model geometry. Solid modeling-based procedures can offer a complementary capability particularly useful in describing complex geometries. They are not incorporated in PC/THERMAL, but can be accessed, if needed, with another member of the ANSYS-PC family of programs, PC/SOLID, which provides the solid modeling capability of the full ANSYS program.

TEMPERATURE PREDICTION

Temperature predictions were made for a simple gas tungsten arc (GTA) welding experiment carried out by Duncan specifically to furnish data for evaluating FEM code suitability for welding analyses (4). He made a 200 amp, 2.1 second duration, GTA spot weld at the center of 76.2 mm. (3 inch) diameter, 4.76 mm. (3/16 inch) thick, circular nickel plate, Figure 1. Thermocouple temperature measurements were made at four locations on the plate. Weld pool decay following flux termination was observed and recorded.

Duncan's welding experiment is well suited for its intended purpose. The simple axisymmetric plate geometry is two-dimensional and avoids errors introduced when two-dimensional analyses are used to describe three-dimensional welds. The simple axisymmetric finite element meshes which can be used to describe the experiment are easily generated and input heat flux and radiation and convection boundary conditions easily specified. The simple meshes contain relatively few nodes and elements, minimizing FEM analysis computer run times and facilitating parametric series of computations investigating the effects of changes in computational procedures or modeling details. Welding a pure nickel plate eliminated the effects of solid phase change associated heats of transformation from experimental results and avoided related computational difficulties. No filler metal was introduced during the weld, avoiding another computational difficulty.

The accurate prediction of welding temperatures requires that limitations of available high temperature material thermal property data, input heat flux data, and basic understanding of welding related heat transfer phenomena be overcome (4). Material property data required for welding heat transfer analyses include coefficient of conduction and specific heat values for temperatures ranging from ambient up to the maximum temperature likely to be experienced in the molten weld pool. High temperature data is often not available and must be estimated. Input welding heat fluxes are difficult to characterize. Their time dependence and spatial distribution must be approximated using simplified physical models and experimentally-based empirical corrections. Although progress is being made in developing a fundamental understanding of weld pool convection heat transfer and weld pool vaporization heat losses (7) (8), these phenomena are still not amenable to analytical description as a part of welding temperature computations. Their effects are empirically included in computations through the use of artificially enhanced coefficients of conduction for molten material and adjustments to input heat flux descriptions.

Nickel conductivity and specific heat values used in the temperature prediction are shown in Figures 2 and 3. The large increase in conductivity for temperatures above 1750 deg. K (2700 deg. F) is an empirical correction recommended by Duncan based on experience with similar welds to approximate the effects of convection heat transfer through the molten weld pool. The increase in specific heat between temperatures of 1655 deg. K (2520 deg. F) and 1783 deg. K (2750 deg. F) incorporates the nickel heat of fusion in the analysis.

The spatial dependence of the GTA generated heat flux applied to the upper surface of the plate is described by Figure 4. Flux time dependence is included in the analysis by multiplying flux values from Figure 4 by the time dependent scale factor from Figure 5. Linear flux decay to zero over a .1 second duration time interval following arc termination at 2.1 seconds was assumed.

Radiation and convection heat losses from the plate to its surrounding occur during the welding experiment. An emissivity of .5 and surroundings at 294 deg. K (70 deg. F) were assumed in describing radiation losses. Convection losses were considered to be unimportant during the time period of interest and were neglected.

The axisymmetric finite element model shown in Figure 6 was generated with direct node and element generation procedures. Solid modeling procedures were felt to be unnecessary because of the plate's simple two-dimensional geometry. The plate is modeled with 656 nodes and 600 axisymmetric four-node heat transfer (STIF55) elements. A fine mesh is used near the center of the plate where severe thermal gradients are expected and a coarser mesh outside this region.

Two constant 294 deg. K (70 deg. F) temperature nodes above and below the plate represent its surroundings for radiation heat losses. The constant temperature nodes are connected to nodes on the upper and lower plate surfaces by radiation link (STIF31) elements. Significant radiation heat loss occurs only near the center of the plate where relatively high temperatures are experienced. Radiation link elements were accordingly inserted only in this region. Effective areas were manually calculated for each radiation link element and input directly. This direct manual procedure is satisfactory for simple geometries, but would be unacceptably time-consuming and tedious for complex ones.

Generation of the finite element model was facilitated by interactive color graphics display capability. Node and element plots served to confirm that data was being input correctly and that model geometry was correctly described. User specification of viewing distance and view window center permitted review of local model details but was found to be significantly less convenient than use of the "zoom" feature available with the full ANSYS program.

Prescribed nodal heat flows describe the GTA weld heat flux applied to the plate. Locally uniform heat fluxes were assumed over each of the one radian in arc length areas existing between nodes. Each area heat flow was manually calculated and allocated equally to the two nodes adjacent to the area. Total heat flow into each node was then input directly. As with radiation link area description, this direct manual procedure is satisfactory for simple geometries, but would be unacceptable for complex ones.

Heat flux time dependence was modeled using multiple load steps. Time ramped load scaling was applied to nodal heat flows over each load step. Time increments during load steps were chosen directly rather than through time step optimization.

The FEM code writes a complete set of computed temperatures to the output data file at the end of each time increment. Computations are terminated by the computer operating system if the output file becomes so large that it fills all the available disk storage space. The total number of increments that can be included in a temperature prediction is accordingly limited. Increment sizes were chosen to obtain acceptably accurate predictions within this constraint.

Two welding computations were carried out. One computation, with 7 load steps and a total of 62 time increments, described plate temperatures during the 5 seconds following arc initiation. Temperatures predicted at thermocouple locations are compared to measured temperatures in Figure 7. The second computation, with 5 load steps and total of 40 time increments, described the 2.44 seconds following arc initiation. Smaller time increments were used after arc termination to better define weld pool decay. Predicted and measured weld pool decay are compared in Figure 8.

Excellent agreement is seen to exist between measured and predicted quantities. Differences observed between them are quite small given uncertainties in material properties and weld flux time and spatial dependence, experimental errors in temperature and weld pool radius measurements, and discretization and numerical errors in FEM computations.

Adequate storage space was available on the 20 Mb. internal hard disk to store data files and simultaneously required program files during each of the computations. Additional data file storage space could be provided by storing program files in a logical hard disk subdirectory actually physically located on one of the auxiliary cartridge drives. Even with this measure taken to maximize available data storage space, however, care in time increment selection would still be necessary to avoid unacceptably large output data files and premature run termination.

Elapsed run times were approximately 9 hours and 5.5 hours for the two welding computations. They were essentially equal to indicated CP times. They compare quite favorably to times required to carry out the same computations in batch mode on a heavily loaded DEC VAX 11/750 minicomputer with the general ANSYS program. Minicomputer elapsed times were up to twice as long as personal computer elapsed times, even though minicomputer CP times were less than one third of personal computer CP times.

Effective interactive postprocessing of computer results was supported by personal computer color graphics capability. Computed temperatures were readily evaluated and interpreted using temperature-time history and temperature contour plot displays. Selecting and sorting procedures assisted in extracting a description of weld pool decay from the computed results.

CONCLUSIONS AND DISCUSSION

Use of a 16-bit personal computer and associated nonlinear finite element software to predict temperatures for a simple GTA plate welding experiment has led to several conclusions.

> 16-bit personal computer welding temperature prediction is feasible, at least for relatively simple geometries. Predictions as accurate as can be expected given uncertainties in material properties and weld flux heat input definitions can be obtained with reasonable computer run times.
>
> The complexity of welding geometries and the duration of welding processes for which temperature predictions can be made can be seriously limited by available personal computer disk data storage space. This is particularly true for finite element software which saves computed quantities for each time increment.
>
> Finite element software preprocessor and postprocessor use of personal computer color graphics can lead to effective interactive welding temperature prediction. Color displays of node and element plots support weld geometry definition. Displays of temperature-time histories and temperature contour plots support evaluation and interpretation of computed results.
>
> Simple direct node and element mesh generation techniques and manual calculation and input of radiation boundary element parameters and welding arc nodal heat fluxes are acceptable for simple weld geometries. Complex geometries will require solid modeling based mesh generation and simplified radiation boundary and heat flux definition procedures.

The introduction within the last year of personal computers with 32-bit processors several times faster than available 16-bit processors, as well as memory sizes as large as 16 Mb. and maximum fixed disk storage of over 600 Mb., should facilitate personal computer welding computations. This will be particularly true when personal computer operating systems with the features, including adequate direct memory and single volume disk addressability and virtual memory capability, necessary to effectively carry out computationally intensive tasks become available (9), (10) and associated finite element software is developed. Personal computer prediction of welding stresses and displacements, as well as temperatures, may then become feasible.

REFERENCES

1. Goldak, J., McDill, M., Oddy, A., Bibby, M. J., House, R., and Chi, X., "Computational Heat Transfer For Weld Mechanics," International Symposium On Computer Modeling of Fabrication Processes and Constitutive Behavior of Metals, Ottawa, Canada, May, 1986.

2. Goldak, J., Patel, B., Bibby, M., and Moore, J., "Computational Weld Mechanics," International Symposium on Computer Modeling of Fabrication Processes and Constitutive Behavior of Metals, Ottawa, Canada, May, 1986.

3. Van Gulick, Leonard A., "Finite Element Welding Computations Using General Purpose Nonlinear Analysis Codes," to be presented at the ASME-PVP Conference, Pittsburgh, PA, June 1988.

4. Duncan, D. A., "Thermal-Mechanical Modeling of the Gas-Tungsten-Arc (GTA) Welding Process," Lawrence Livermore Laboratory, Report UCRL-52891, Livermore, CA, 1980.

5. ANSYS NEWS, First Issue, 1986, Swanson Analysis Systems, Houston, PA, pp. 1-3.

6. ANSYS-PC/THERMAL User's Manual, K. S. Krishna, R. W. Gorman, Swanson Analysis Systems, Houston, PA, 1986.

7. Shapiro, A. B. and Mahin, K. W., "Thermal-Mechanical Modeling of Gas Tungsten Arc Welding," Lawrence Livermore National Laboratory, Report UCRL-94627, Livermore, CA, 1986.

8. Oreper, G. M., Eagar, T. W., and Szekely, J., "Convection in Arc Weld Pools," The Welding Journal, Vol. 62, No. 11, November, 1983, pp. 307s-312s.

9. Robinson, P., "Worlds Collide in Low-End Market," Computer Graphics World, Vol. 11, No. 2, February, 1988, pp. 60 - 72.

10. Burdick, D. B., "Personal Computers and Workstations Collide," Computers in Mechanical Engineering, Vol. 6, No. 4, January/February, 1988, pp. 12-16.

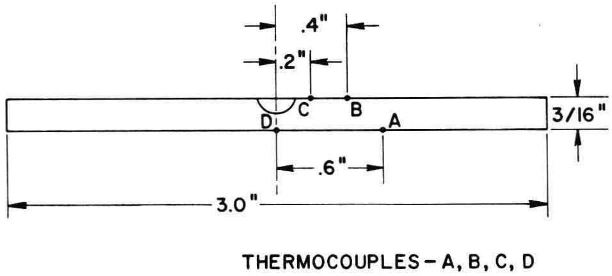

FIGURE 1 WELDING GEOMETRY

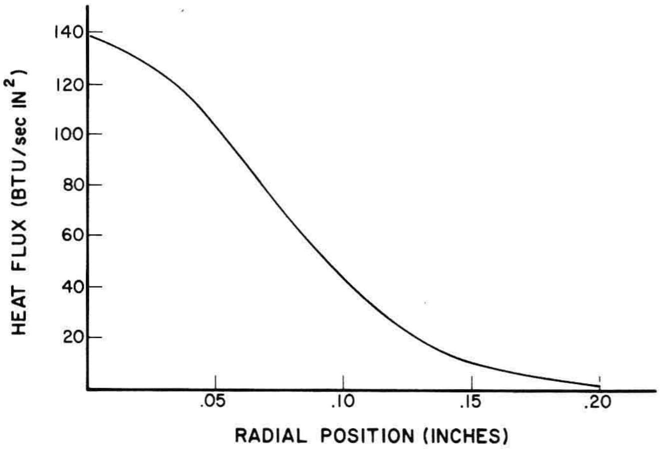

FIGURE 4 GTA HEAT FLUX

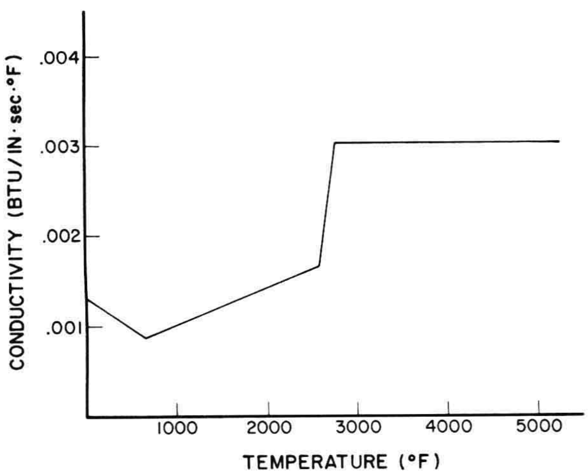

FIGURE 2 NICKEL CONDUCTIVITY

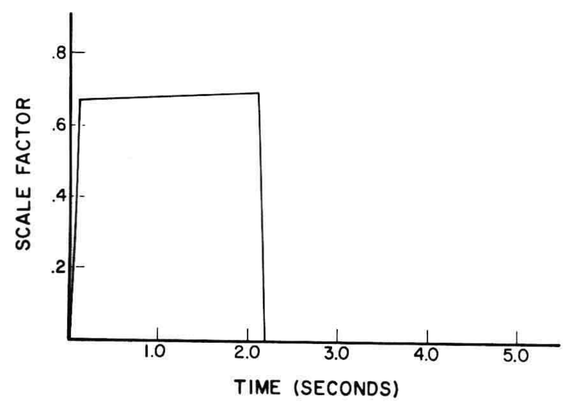

FIGURE 5 FLUX SCALE FACTOR

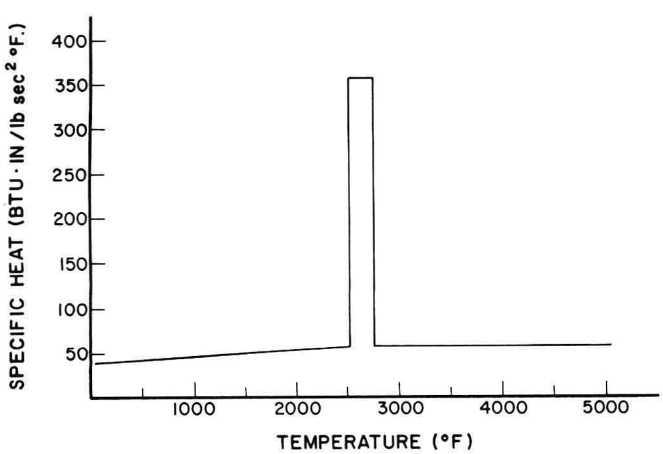

FIGURE 3 NICKEL SPECIFIC HEAT

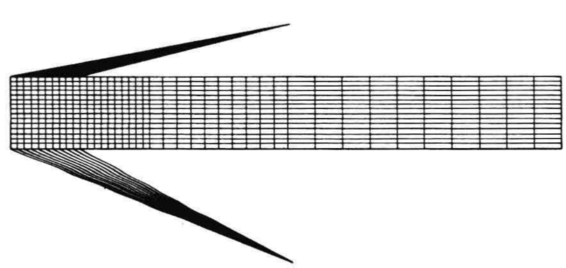

FIGURE 6 FINITE ELEMENT MODEL

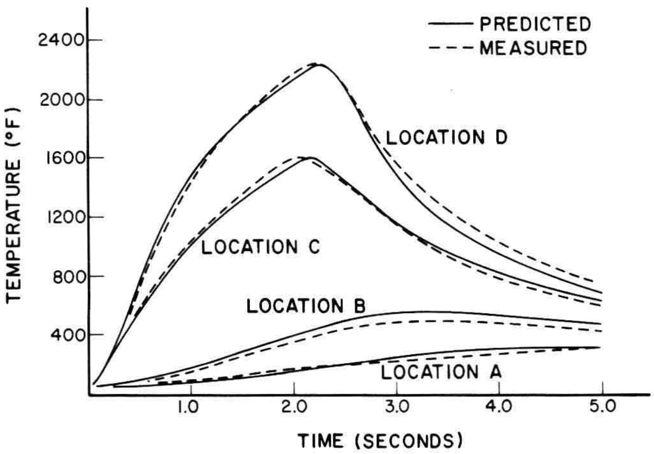

FIGURE 7 WELDING TEMPERATURES

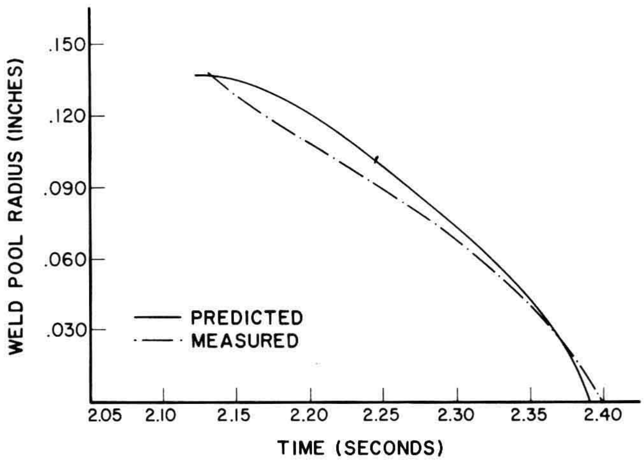

FIGURE 8 WELD POOL DECAY

APPROPRIATE LEVELS OF CAD TRANSLATORS FOR FINITE ELEMENT ANALYSIS

J. I. Finkel
Swanson Analysis Systems, Inc.
Houston, Pennsylvania

ABSTRACT

Large databases of geometric information created by CAD systems exist in most manufacturing enterprises. A majority of these firms also employ some type of computerized analysis program, typically finite element analysis. Yet, the sharing of CAD data between drafting and analysis has been limited because of the difficulties associated with the translation process. This paper will focus on real-world examples of translating CAD models into a format suitable for finite element analysis. Examples will be included, with varying levels of detail. Topics discussed include acceptable levels of modeling detail, data paths through CAD and FEA codes, and desirable features in the translation and modeling processes. Detailed discussions will be presented on model geometry created on one system and analyzed on another.

INTRODUCTION

The availability of mainframe and PC-based drafting packages makes computerized drawing accessible to manufacturing firms of all sizes. The end goal of these programs is to produce a set of drawings for the manufacture of a product. To ensure high quality and prevent field failures, the same geometric data can be passed to analysis programs.

The function of the analysis programs is to verify the suitability of a design and its ability to stand up to expected loads and stresses. With this type of computer-based analysis, products can be subjected to a full range of testing without extensive prototyping. Since the geometric data already exists, the most time-consuming task of finite element analysis (FEA), modeling, is eased.

There are many ways to get geometric data from a CAD program into an FEA code. The problem is to find the best way, where *best* is defined as the path that takes the least amount of engineering effort and minimizes the time of geometry modification and modeling time.

This paper will deal with the varying options for the *best* methods and the differing requirements of CAD and analysis. The focus of the paper will be on the most time-effective and accurate methods of moving CAD data into an analysis program. Specific examples will show the paths of data movement from the CAD programs Dimensions and Prime-Medusa into the ANSYS FEA code.

GEOMETRIC REQUIREMENTS IN FEA

A CAD drawing must be sufficiently complete to allow manufacture of the part. Requirements for a finished drawing include detailed geometry, large numbers of witness and dimension lines, complete dimensional and tolerance data, and all associated notes (Ref. 1). However, developing a CAD model for FEA relaxes many of the restrictions imposed on a working drawing.

With FEA any text data is eliminated in the translation process. All dimension and reference lines are also unnecessary. Fillets, chamfers, and holes, while they may serve as stress concentrators, are often unnecessary in developing an FEA model (Ref. 2). In fact, in the CAD-FEA translation process, close to 90% of the data on a drawing can be eliminated (Table 1). The FE model made from this data, though, exceeds the size of the initial CAD neutral data file by a factor of one hundred or more.

In view of the extensive number of deletions, an incomplete CAD drawing may be a better starting point for an FEA model. When using a CAD system to create working drawings or an FE model, it may be best to put all notes, dimensions, and other non-part related geometry on separate layers (Ref. 3). This step reduces the amount of data to be passed through a translator, and results in a geometric model that is easier to mesh. Meshing requirements will be covered in detail in later sections of this paper.

Setting up boundary conditions and ensuring accuracy of edges and surfaces for FEA impose restrictions on CAD models. For example, it is a good idea to use a effects of element skew and other factors on modeling accuracy are covered extensively in a series of articles in "Finite Element News" which began in February, 1986, and continues today.

The best level of translator to use with the FE programs that have translators and meshing algorithms is generally the last, where minimal geometry information is passed. With this technique, where creation of additional geometry required to perform analysis is done within the FEA code, users have additional assurance that lines, areas, and volumes can be defined to ease the task of creating a finite element mesh. It is important to note that within the FEA code, though the element mesh is created on the model geometry, the mesh can be easily regenerated without rebuilding the model. The following examples will support this conclusion.

PISTON EXAMPLE

Initial geometry for this simplified quarter symmetry model of a piston came from a Prime-MEDUSA CAD system (Fig. 1). Both lines and areas were created and the geometry then passed into the ANSYS program as keypoint, line and area information.

Two attempts were made to mesh this model. The first used a subset of the initial geometry from the CAD system and the intent was to use various sweeps to create the full model. The second used a superset of the initial geometry, with lines added to parse the model into areas better suited to the ANSYS mesh generator. Reasons for abandonment of the first and the steps taken in the second case will be presented.

In the first case, the model could be thought of as the joining of a quarter section of a piston with a rectangular prism. Initially, the quarter section was created as a solid of revolution from one face (Fig. 2). The difficulty caused by this approach was the joining of the two solids. This joint required a good deal of geometry creation to ensure proper continuity between the two parts.

With this approach, the prism would have intersected two different volumes. To find the intersection of a single volume with two adjacent volumes and creating acceptable areas for meshing would have taken hours of engineering time. In view of this effort, it was decided to use more of the initial geometry.

The second attempt was based on the initial geometry with the regeneration and addition of some lines. The intent was to divide the model into four volumes instead of three as before. The piston head contained one interior and two exterior volumes. The fourth was the rectangular rod, which was to be attached by forming the intersection of this prism with a single volume.

To improve the accuracy of the model, some of the initial 90° circular arcs were recreated as two 45° segments. This step was taken because at angles >45° the cubic parametric spline representation of a line segment used by the ANSYS code may show distortion of the order of 0.1% from the exact quadratic definition of a circular arc (Ref. 7).

Knowing the geometry and the requirements of the mesh generator helped in the parsing of the model into appropriate areas. As previously mentioned, the 90° segment was divided into 45° segments by sweeping the edge area in Fig. 2 through the model. Once this step was performed, the next step was to define the appropriate areas and then volumes.

It must be noted that the initial areas passed through the translator were deleted prior to the defining of the regions with the ANSYS code. This step was taken because areas defined outside the ANSYS program were not chosen with FEA meshing in mind. Therefore, they may not have meshed well with the automatic algorithms of the ANSYS code. Were the original areas used, the mesh divisions could not be easily changed by the operator.

Using ANSYS commands, the volumes were defined by on-screen picks of the areas surrounding the desired volume. Because of the close proximity of adjacent areas, some effort was expended in ensuring that adjacent areas were not included in the volume. The ANSYS solid modeler will generate an error message if a volume is not correctly defined by only contiguous areas. Similarly, if an unclosed area is defined, the ANSYS code will generate an error message.

Once the volumes were generated, the user selected an appropriate element type, constrained the model, and ran the analysis. For this model, tetrahedral elements were selected because of the irregular part geometry (Fig. 3). As in the first example, elements were sized to ensure adequate aspect ratios and numbers or elements through the part thickness.

CONNECTING ROD EXAMPLE

This connecting rod model was originally created on an Apple Macintosh II computer using the Dimensions CAD program. Geometry from this 2-D model was used to drive the translation and subsequent analysis. The data path in this example illustrates the degree of engineering and computer expertise that may be required to translate even a simplistic model using the best level of translation if several computers and programs are involved.

The initial geometry for this analysis came from a CAD system as a DXF file (Fig. 4). The associated text and dimensions border, legend, as well as the witness and dimension lines should not be passed through a translator, so these drawing layers were removed. The resultant geometry only file was then passed into AutoCAD on an IBM-PC because there is not a translator between Dimensions and the ANSYS code(Fig. 5).

From AutoCAD, the geometry file was run through the ANSYS-AutoCAD translator written by Swanson Analysis Systems, Inc. The output of this translator contains line and keypoint information. This data was then used as an input file for the ANSYS-PC/SOLID program. It is important to note that this file can serve as input for either a PC-based or any other platform running the ANSYS code. In this case, all the modeling was done on a Prime 9950, and the initial geometry came from an Apple Macintosh-based system.

Once the line and point data was in ANSYS, the next step was creation of the FEA model. This procedure took several steps, and began with a cleanup of the initial geometry. Cleanup consisted of ensuring that all areas were well connected and some slight geometric modifications. For example, the cutouts in a non-critical region were eliminated and replaced with a sharp corner as opposed to an inset.

TABLE 1. MODEL SIZES

CONNECTING ROD MODEL

	FILE SIZE (Words)	PERCENTAGE OF INITIAL SIZE
Original Geometry	45899	100.0
Translator Output	5228	11.4
Meshed Model	64765	141.1

FE DATA

	Initial Geometry	Meshed Geometry
Keypoints	87	84
Lines	67	47
Areas	0	8
Nodes		719
Elements (Triangles)		313

PISTON MODEL

	FILE SIZE (Words)	INCREASE FROM INITIAL SIZE
Original Geometry	N/A	
Translator Output	1387	1.0
Meshed Model	153,120	110.4

FE DATA

	Initial Geometry	Meshed Geometry
Keypoints	33	34
Lines	39	72
Areas	12	43
Nodes		1488
Elements (Tetrahedra)		802

TABLE 2. NUMERIC INACCURACIES

DATA AS TRANSLATED INTO INCHES

X	Y	Z
4.882021E-01	3.081674E-05	1.736210E+00
4.882015E-01	3.099581E-05	-5.208007E-07
4.882015E-01	-6.574790E-01	-5.208007E-07
4.882021E-01	-6.574793E-01	1.736210E+00

INITIAL METRIC DIMENSIONS (CM)

X	Y	Z
1.24	0.00	4.41
1.24	0.00	0.00
1.24	1.67	0.00
1.24	1.67	4.41

FIGURE 1.

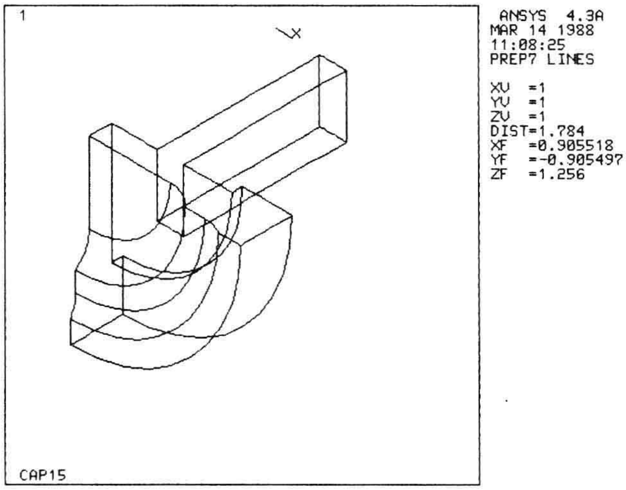

FIGURE 2.

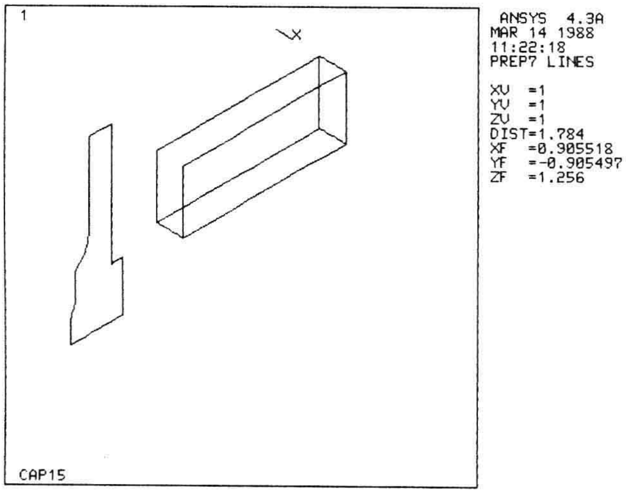

FIGURE 3.