**SAE TECHNICAL PAPER SERIES**

**980750**

# Inserted Tool Design Using Finite Element Analysis

**Mehul A. Dharia**
Amcast Industrial Corporation

Reprinted From: Product and Process Integration: Advanced Technology (SP-1337)

The Engineering Society
For Advancing Mobility
Land Sea Air and Space®
INTERNATIONAL

International Congress and Exposition
Detroit, Michigan
February 23-26, 1998

400 Commonwealth Drive, Warrendale, PA 15096-0001 U.S.A. Tel: (724) 776-4841 Fax: (724) 776-5760

The appearance of the ISSN code at the bottom of this page indicates SAE's consent that copies of the paper may be made for personal or internal use of specific clients. This consent is given on the condition however, that the copier pay a $7.00 per article copy fee through the Copyright Clearance Center, Inc. Operations Center, 222 Rosewood Drive, Danvers, MA 01923 for copying beyond that permitted by Sections 107 or 108 of the U.S. Copyright Law. This consent does not extend to other kinds of copying such as copying for general distribution, for advertising or promotional purposes, for creating new collective works, or for resale.

SAE routinely stocks printed papers for a period of three years following date of publication. Direct your orders to SAE Customer Sales and Satisfaction Department.

Quantity reprint rates can be obtained from the Customer Sales and Satisfaction Department.

To request permission to reprint a technical paper or permission to use copyrighted SAE publications in other works, contact the SAE Publications Group.

**GLOBAL MOBILITY** DATABASE

*All SAE papers, standards, and selected books are abstracted and indexed in the SAE Global Mobility Database.*

No part of this publication may by reproduced in any form, in an electronic retrieval system or otherwise, without the prior written permission of the publisher.

**ISSN 0148-7191**
**Copyright 1998 Society of Automotive Engineers, Inc.**

Positions and opinions advanced in this paper are those of the author(s) and not necessarily those of SAE. The author is solely responsible for the content of the paper. A process is available by which discussions will be printed with the paper if it is published in SAE Transactions. For permission to publish this paper in full or in part, contact the SAE Publications Group.

Persons wishing to submit papers to be considered for presentation or publication through SAE should send the manuscript or a 300 word abstract of a proposed manuscript to: Secretary, Engineering Meetings Board, SAE.

**Printed in USA**

980750

# Inserted Tool Design Using Finite Element Analysis

**Mehul A. Dharia**
Amcast Industrial Corporation

## ABSTRACT

Elkhart Industrial Division of Amcast Industrial Corporation has developed the FORMCAST™ process to manufacture near-net shape components. Tooling used for this process consists of complex shaped cavities subjected to very high forming pressure. It is a common practice to use inserts that reduce stresses in such tooling.

This paper will describe how a general purpose Finite Element Analysis (FEA) software can be used to develop inserted tooling designs. FEA simulations will be discussed for two different inserted tooling designs, one using interference fit, and other using slide fit. Data presented in this paper will explain the computer models and analysis iterations used to develop optimal inserted tooling designs.

## INTRODUCTION

Finite Element Analysis (FEA) has been in use for many years. With the recent advancement of computer hardware and numerical software, many engineering offices have included FEA simulations in their product development cycle. However, a few are using it to predict structural behavior of the complex multi-part tooling with inserts.

The purpose of this paper is to discuss modeling and analysis techniques for two such tooling assemblies. It describes the FEA performed on two different tooling designs that were developed to manufacture automotive parts namely, Insert Tube and Miniblock, using the FORMCAST™ process.

There are several other traditional manufacturing processes, (forming, extrusion, etc.), that use very high pressure inside the die cavity and require some form of inserted tooling. The material presented in this paper can also be applied to tool designs for these processes.

## INSERTED TOOL DESIGN

Elkhart Industrial has developed several inserted tooling assemblies used in the high pressure FORMCAST™ process. The tools for this process generally include a precise machined cavity of complex shapes for higher dimensional repeatability. In order to achieve near-net shape of the cavity, the metal being formed is enclosed within the tooling cavity surfaces during the complete production cycle. This in turn generates very high forming pressure on the surfaces of die cavity.

The high pressure during the forming operation results in high stresses at the inside surfaces of the die cavity. The pressure, and therefore the stresses in the die, are progressively reduced in the areas away from the cavity. Any sudden changes or sharp corners in the geometry of the die cavity surface will give rise to very high stresses. This often results in a crack initiation in the die.

To avoid the crack initiation, it is a common practice to use one or more inserts in the critical forming area. The inserts can be placed either in the longitudinal and/or transverse loading direction depending upon the complexity of the shape being formed.

In case of fatigue failure, it is more economical to replace small inserts rather than replacing the entire tool. In addition to this, inserts can be made of the same material as the die or of different material. This can be decided based on the deflection pattern and stress behavior in the forming area. For example, if the overall compressive stresses are dominating, a material behaving well under compression may be chosen as an insert.

## FINITE ELEMENT ANALYSIS SIMULATION

There are several design standards, data tables and mathematical formulas available to develop inserted tooling. Most of them, however, are developed for typical axisymmetric shapes undergoing specified uniform pressure loading. It is extremely difficult to apply this data to a three dimensional unsymmetrical shaped tooling subjected to nonuniform loading, without making gross assumptions.

This often results in a development process involving trial and error approach. Several prototypes need to be developed and tested before releasing the tooling for production usage.

The current trend of cost reduction in the automotive industry demands production of better parts made faster and made cheaper. Any tool that will help in achieving this will result in shorter design cycle time to market for automotive manufactures and suppliers. FEA is one such tool which simulates the behavior of inserted tooling assemblies in two or three dimension, subjected to actual forming conditions. It not only reduces the need to develop many prototypes but also ensures development of optimized tooling designs.

FEA simulation helps to predict structural behavior for several inserted tooling configurations. Several contact conditions, (slide fit, interference fit for prestressing, etc.), among die pieces can be analyzed to optimize the tooling for the stresses. The deflection pattern predicted from the analysis provides the valuable insight on how one piece of the tooling will affect the other pieces, and the extent of flash generated.

Elkhart Industrial has been using general purpose FEA software as an integral part of their tooling design process. In the following sections, two such FEA simulations, used for developing FORMCAST™ inserted tooling, are presented.

## EVAPORATOR CORE INLET TUBE

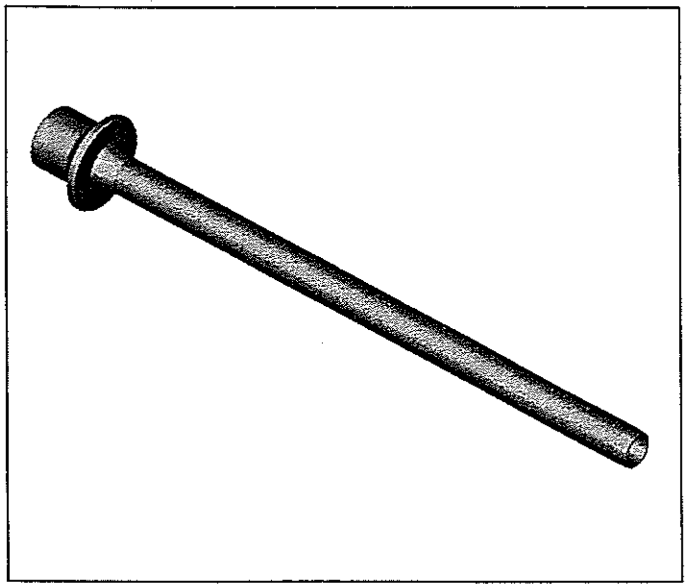

Figure 1 Evaporator Core Inlet Tube

The Evaporator Core Inlet Tube, shown in Figure 1, is an aluminum component of a car air-conditioning unit.

The process used to form the component from a hollow aluminum billet is very similar to the combined, forward and backward, extrusion process. The schematic of the process is shown in Figures 2a and 2b. During the forming operation, a blank is placed in the large upper cavity and a punch extrudes a portion of the metal into the smaller cavities to form a flanged tube.

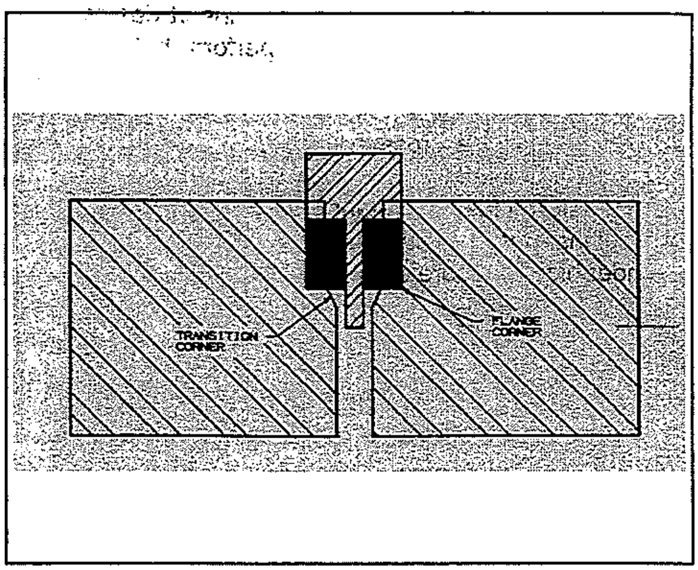

Figure 2a Beginning of Forming Cycle

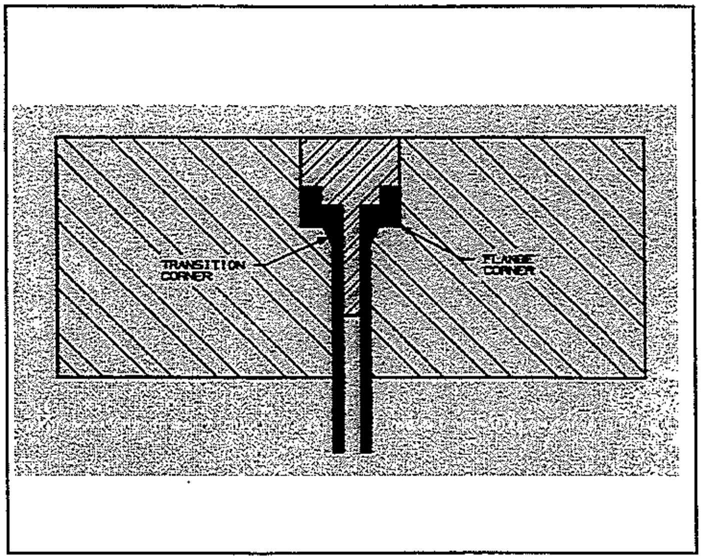

Figure 2b End of Forming Cycle

## ONE PIECE TOOLING

A finite element analysis on a one piece tooling, made of H13 steel material, was performed. Taking advantage of the tooling's symmetry about the center axis, a half finite element model was prepared using axisymmetric elements. Figures A1 and A2 displays von mises stress contours on the half die model and forming area of die cavity respectively. Very high stresses at the transition corner and at the flange corner are evident in these plots. The stresses predicted at the flange corner are very near to the yield stress of the material used for the tooling. This predicts die failure at a very low number of production cycles, and calls for an inserted tooling design to prevent premature die failures.

## MULTI-PART INSERTED DESIGN

It is clear from Figures A3 and A4 that compressive stresses are dominating the high stresses in the forming area of die cavity. A tungsten carbide insert design was developed because of its ability to perform better under compression.

Figure 3 provides a cross section of tooling developed using carbide insert. This design was developed using traditional formulas and standard charts. The tapered shape of insert is to ensure that it will not come out of the die under high pressure of forming.

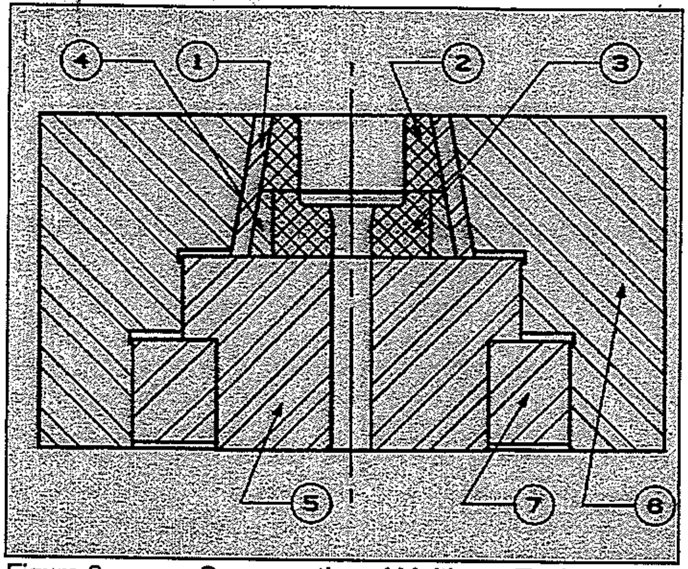

Figure 3 Cross-section of Multi-part Tooling

A finite element analysis was performed on this multi-part design. Appropriate material properties for each part was defined. Boundary conditions among the parts of the tooling were defined so that each part will be just in contact with the other at the beginning of the forming simulation. This nature of contact is referred as a "slide fit". Zero length gap elements were defined at the contact surfaces of various parts. The various stress results are shown in Figures A5 through A8.

It can be seen that the von mises stress values in the critical forming area, namely flange corner and transition corner, has not decreased much from the one piece tooling configuration. However, the carbide material used for insert has higher load bearing capacity than the H13 steel used earlier. This may increase the fatigue life of tooling to some extent. But the tensile stresses in the flange corner did not show any improvement. Since carbide is more brittle, it is extremely important to reduce these tensile stresses to prevent failure.

### Pre-Stressing Of Insert

To avoid this type of failure, it is necessary to provide initial compression to the insert. The forming load application will then have to overcome that initial compression before putting the insert into tension. The tensile stresses on the insert in the fully loaded die can be decreased using this concept. This initial compression can be provided by introducing interference between the outside surface of insert and inside surface of the die casing. As a side effect, the interference will increase tensile forces on the die casing. The idea is to optimize the interference value so that enough initial compression is provided to the insert without jeopardizing the die casing. The following two illustrations provide an idea of how FEA simulations can be utilized to optimize the design.

### FEA With 0.25 mm Interference

The multiple die insert sleeves, made of M2 steel material, ensures that proper interference can be introduced without having an excessive die size. The following table lists the different interference values used among the multiple parts of the die. Please refer to Figure 3 for the part names, (numbers), used in this table.

| Contact Between | Interference in mm | Type of Fit |
|---|---|---|
| Upper Sleeve (1) and Front Insert (2) | 0.0254 | Shrink |
| Lower Sleeve (4) and Rear Insert (3) | 0.0254 | Shrink |
| Upper Sleeve (1) and Lower Sleeve (4) | 0.0699 | Shrink |
| Upper Sleeve (1) and Die Casing (6) | 0.254 | Shrink |
| Front Insert (2) and Rear Insert (3) | 0 | Slide |
| Front Insert (2) and Lower Sleeve (4) | 0 | Slide |
| Rear Insert (3) and Plug Filler (5) | 0 | Slide |
| Lower Sleeve (4) and Plug Filler (5) | 0 | Slide |
| Upper Sleeve (1) and Plug Filler (5) | 0 | Slide |

Table 1 Interference values among die parts

For illustration purpose, only the interference between upper sleeve (1) and die casing (6) is used as a variable for the discussions in this and the following section. All the contact conditions were modeled using unidirectional gap elements on coincident node points. The interference were defined using negative values as initial gap openings. Table 2 summarizes the findings of this analysis.

| Case | Flange corner | Transition corner | Die casing |
|---|---|---|---|
| No Interference | 1484 | 1385 | 100 |
| Press fit only | 1909 | 3180 | 1698 |
| Fully loaded | 1400 | 2185 | 1821 |

Table 2 Maximum von mises stresses in Mpa

Figures A9 through A11 display stresses for the press-fit condition only. The interference of 0.254 mm between upper sleeve and die casing results in very high stresses on cavity surfaces of the carbide insert. The principal stress plots suggest that the high tensile stresses at the flange corner of one piece design has been eliminated. The high stresses at that corner are now compressive. Although the high stresses are compressive, they are high enough to cause concern, because the assembled die is not going to be in the operation at all times.

It is also evident that excessive tensile forces are applied, as a result of interference, to the die casing. This results in high stresses near to yield point of the die casing. In addition, the die case will experience further stresses from the forming load during the process.

Stresses on the fully loaded tooling during forming are presented in Figures A12 through A14. As the forming load is applied, the stresses on the insert cavity decreases in the range of 25 to 30 percent. However, compared to its yield stress, it is still on the higher side. As expected, the stresses on die case increased further.

It was apparent that the interference of 0.254 mm helped in decreasing tensile stresses at the flange corner, but made matters worse for other parts of tooling. It was decided to analyze the configuration by reducing the interference.

### FEA With 0.12 mm Interference

All the interference values defined earlier in Table 1 were kept the same, except the one between upper sleeve (1) and die casing (6). This was reduced by 50 percent to 0.12 mm. Figures A15 through A20 presents various stress results on the die components. Table 3 summarizes the findings of this analysis.

| Case | Flange corner | Transition corner | Die casing |
|---|---|---|---|
| No Interference | 1484 | 1385 | 100 |
| Press fit only | 1070 | 1683 | 951 |
| Fully loaded | 997 | 997 | 898 |

Table 3 Maximum von mises stresses in Mpa

The stresses on the carbide flange corner are not only compressive but are also within safe allowable limits. Stresses at the transition corner have also been reduced by more than 50 percent. The stresses on die casing have also been reduced substantially, and are now within its capacity.

## OPTIMAL TOOL DESIGN

This approach can be utilized to arrive at an optimum interference values between all the parts listed in Table 1. Since the modeling does not have to be redone, many different configurations can be analyzed very quickly. It is also possible to study variable interference along the same contact surface. This can be very useful to provide compression in the localized forming area of complex die cavities.

On the same note, this technique can be further extended to 3D models of irregular shapes that cannot be optimized using the available data and charts for 2D and standard axisymmetric shapes.

## MINIBLOCK

Miniblock, also known as Tube and Block, is a one piece aluminum component used in the evaporator system of a car air-conditioner. It replaces a three piece brazed assembly. Figure 4 shows two different miniblocks, one with an oval shaped block, and the other with a rectangular block.

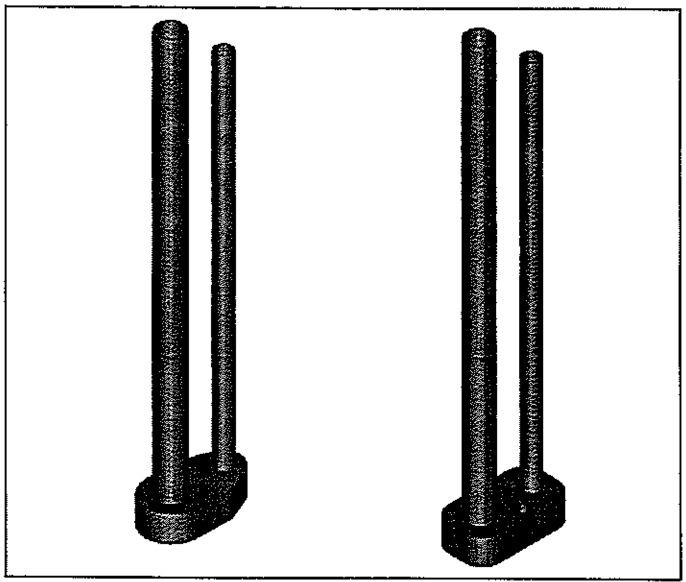

Figure 4 Oval and Rectangular Shaped Miniblocks

The FORMCAST™ tooling consists of a die block and an insert, both made of H13 steel material. The insert is placed inside the die block using slide fit.

## FEA OF OVAL MINIBLOCK

Figures B1 and Figure B2 shows half model of two piece tooling used for oval shaped miniblock. This configuration was first analyzed without modeling the contact between them. The radial displacement results confirmed that some portion of insert deflects more than its adjacent part of die block. In reality, this would not be possible. The insert, trying to deflect more than the die, will have to push much bigger die block and will experience compressive stresses. It is difficult to measure this behavior of nonsymmetrical insert quantitatively. The accurate way to predict this behavior is to model contact between the outer face of the insert and inner face of the die block. This slide-fit contact was modeled using zero length unidirectional gap elements between the surfaces. The forming load was gradually applied in the die cavity to simulate the process.

Figure B3 shows the radial displacements in the cavity area and Figure B4 presents von mises stresses on the tooling. It was observed that during the simulation, the insert gets supported by the die block at left and right radial positions. This results in higher stresses on the insert at mid point of its outer periphery. This will cause yielding of the insert at that point.

### FEA OF RECTANGULAR MINIBLOCK

Half model of two piece tooling for rectangular miniblock can be seen in Figures B5 and B6. When analyzed without the contact, it was clear that none of the insert points were deflecting more than the die block. Figures B7 and B8 present radial displacement and von mises stresses respectively. In this case, modeling the slide-fit using gap elements was not required. However, stresses in the two radial areas of die are closer to the yield value than the oval shaped die. Some insert mechanism in the die block, similar to the one used in insert tube tooling, may have to be considered to increase its fatigue life.

## SUMMARY

It is a common practice to use inserted tool designs for manufacturing processes using high pressure loading The data and charts available to design such tooling work best for axisymmetric shaped tooling undergoing uniform hydrostatic pressure.

Manufacturing processes involving complex three dimensional tooling shapes and varying forming load within the die cavity can not use the available data to develop inserted tooling without making gross assumptions.

Finite element analysis is a powerful tool that can be applied to develop complex inserted tool designs. Structural behavior of multi-part tooling can be predicted and corrected using FEA simulations without having to make many number of prototypes.

The application of this technique in developing shrink fitted and slide fitted multi-part tooling was demonstrated. FEA simulations are especially useful in developing unconventional complex inserted tooling with either nonuniform pressure and/or varying interference among its parts.

## APPENDIX A

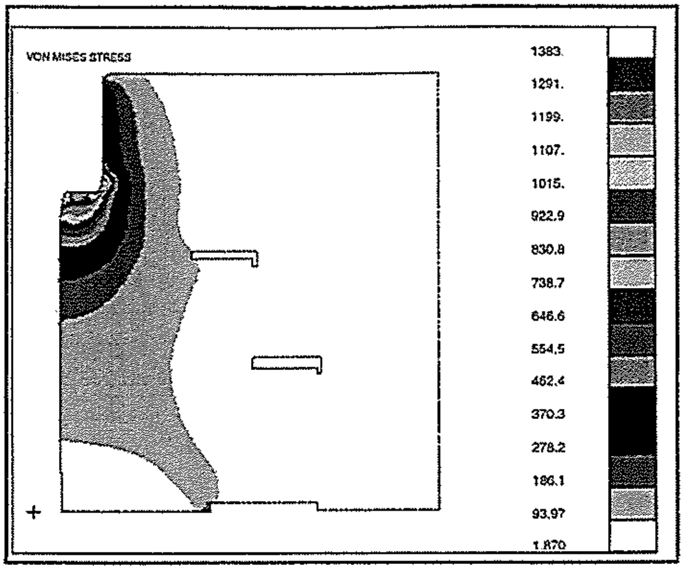

Figure A1 Inlet Tube - One Piece Die
Von Mises Stress

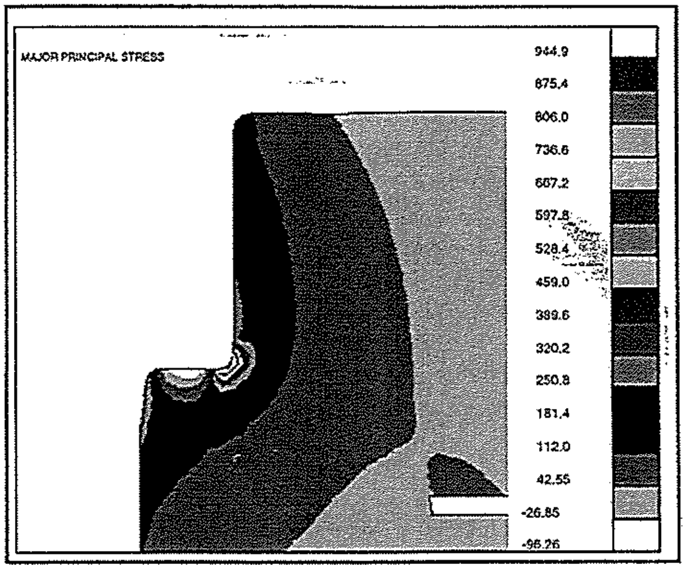

Figure A3 Inlet Tube - One Piece Die
Major Principal Stress

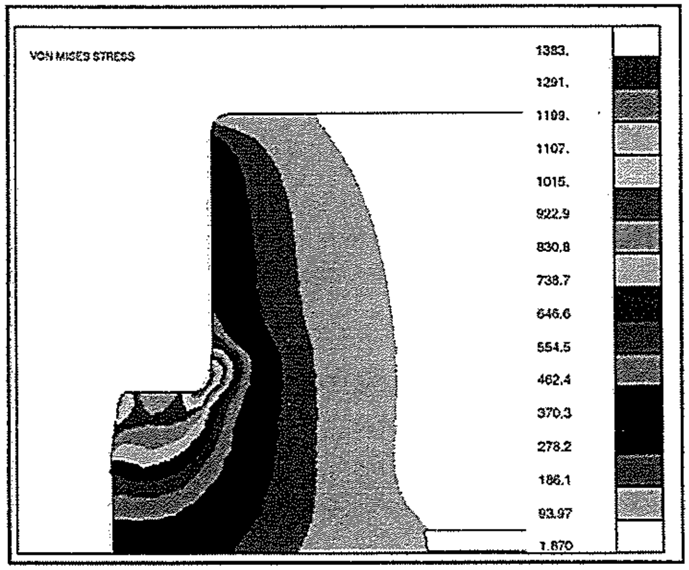

Figure A2 Inlet Tube - One Piece Die
Von Mises Stress - Magnified Plot

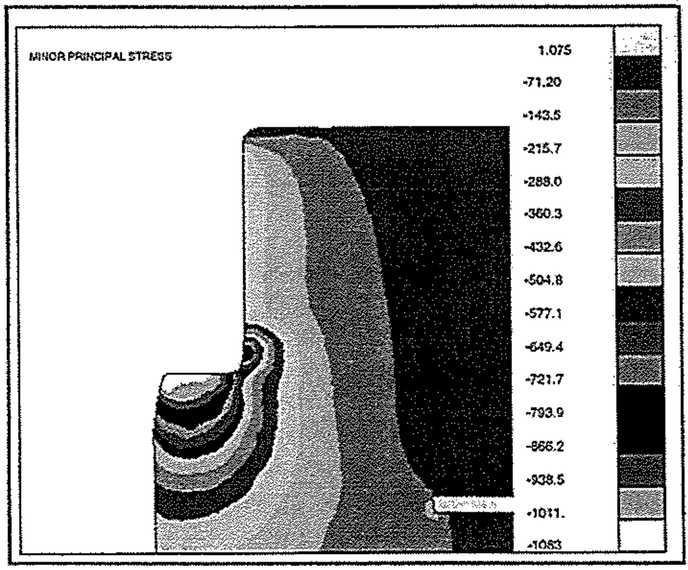

Figure A4 Inlet Tube - One Piece Die
Minor Principal Stress

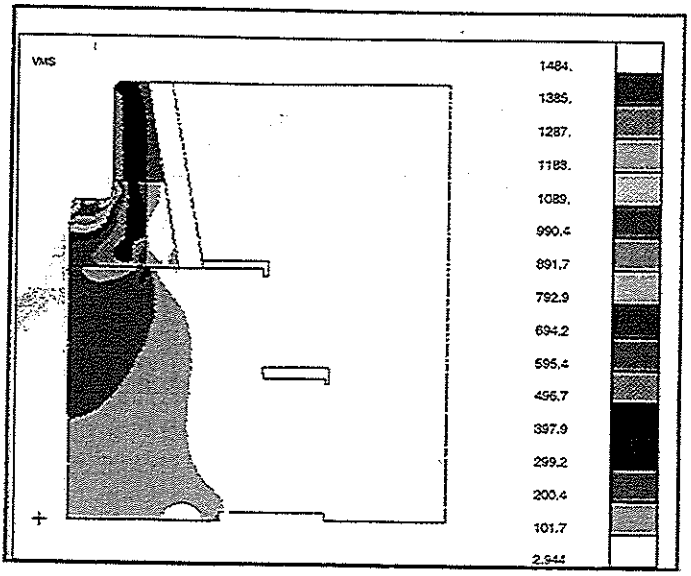

Figure A5 Inlet Tube - Slide-fit Inserted Design
Von Mises Stress

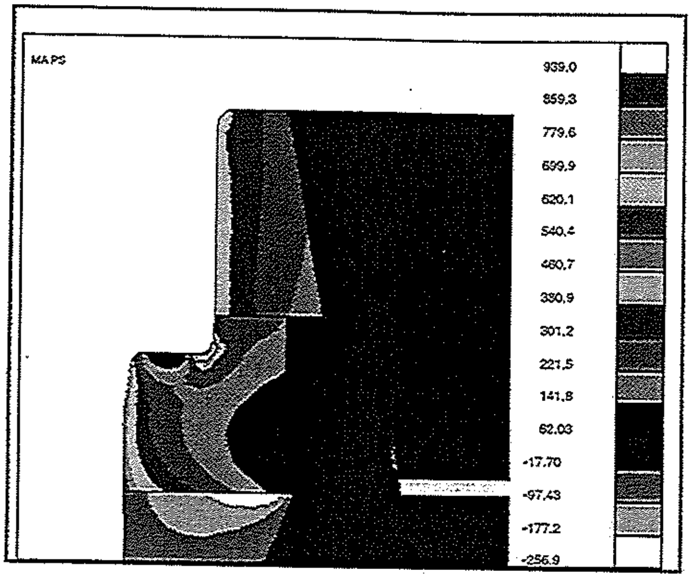

Figure A7 Inlet Tube - Slide-fit Inserted Design
Major Principal Stress

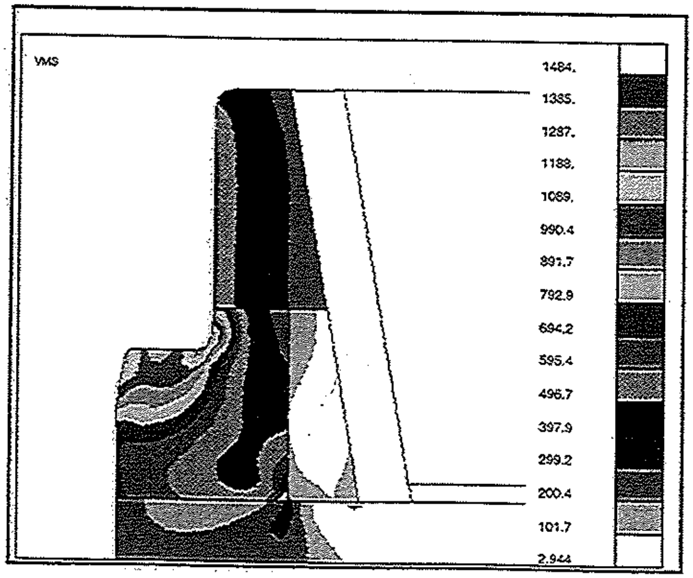

Figure A6 Inlet Tube - Slide-fit Inserted Design
Von Mises Stress - Magnified Plot

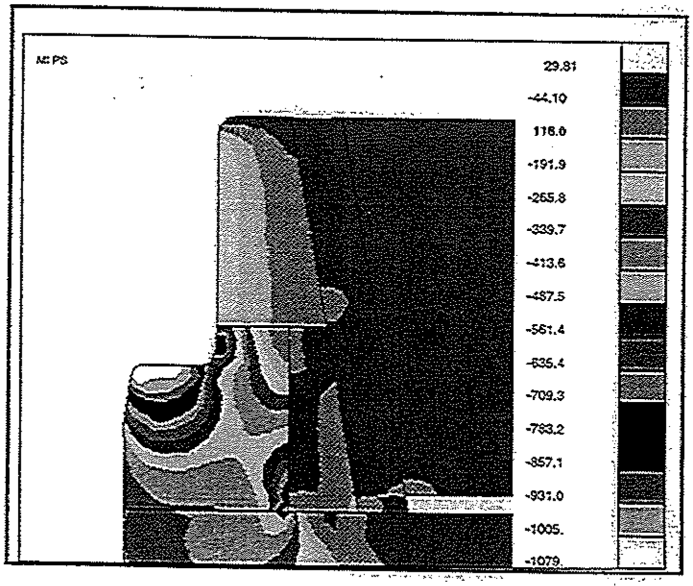

Figure A8 Inlet Tube - Slide-fit Inserted Design
Minor Principal Stress

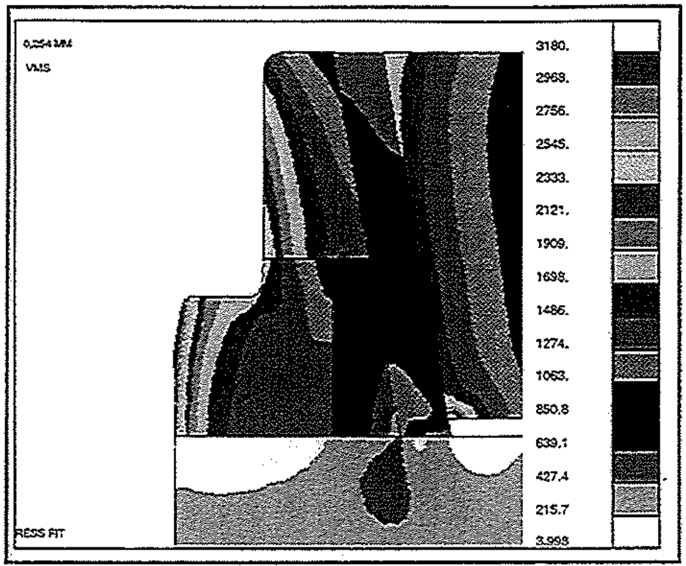

Figure A9 Inlet Tube - Interference 0.254 mm
PressFit, Von Mises Stress

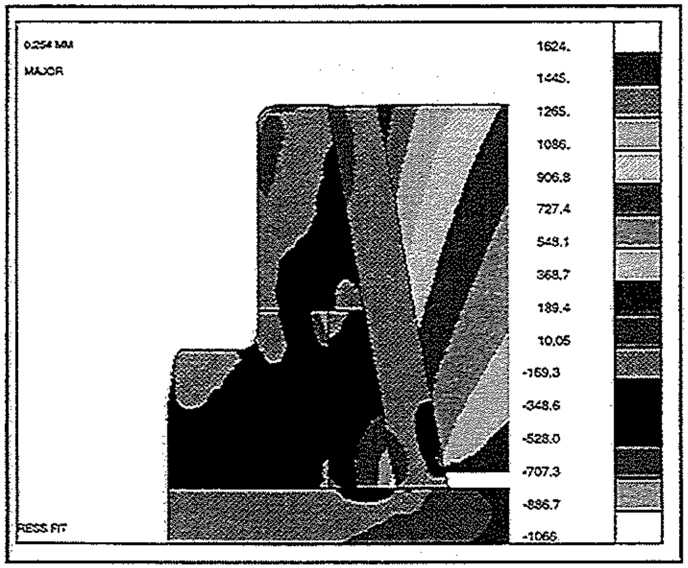

Figure A10 Inlet Tube - Interference 0.254 mm
PressFit, Major Principal Stress

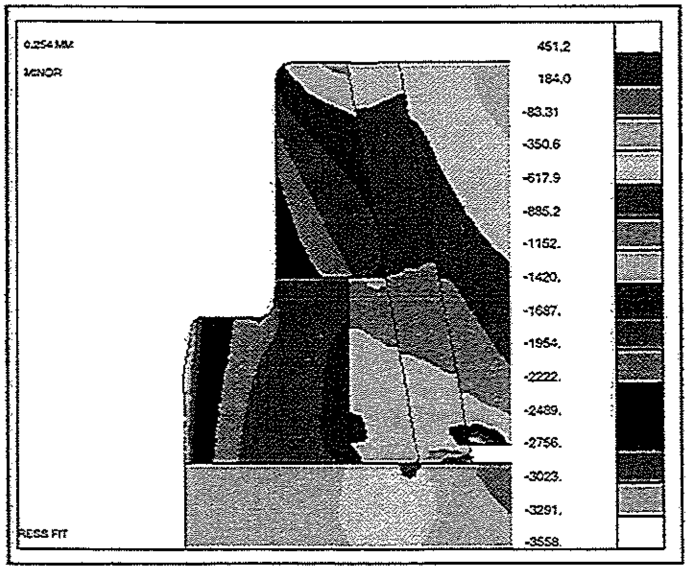

Figure A11 Inlet Tube - Interference 0.254 mm
PressFit, Minor Principal Stress

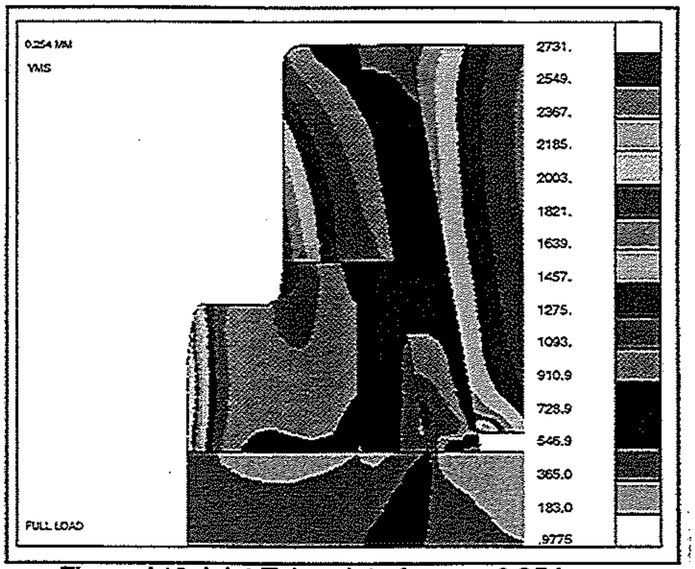

Figure A12 Inlet Tube - Interference 0.254 mm
Full Load, Von Mises Stress

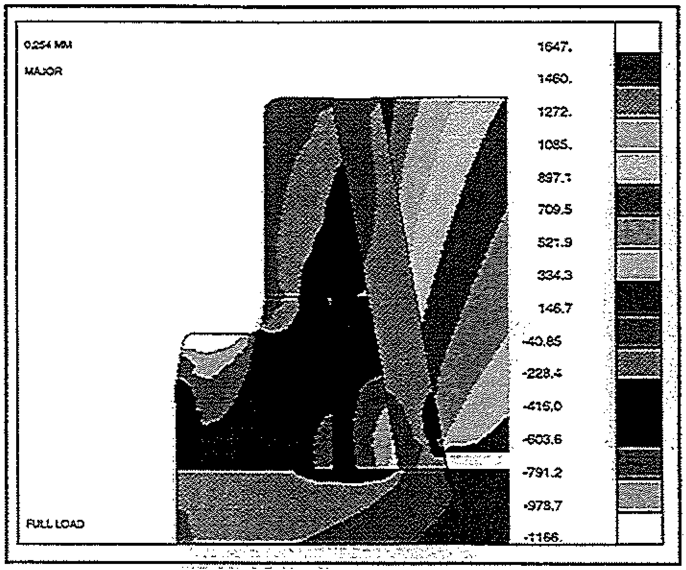

Figure A13 Inlet Tube - Interference 0.254 mm
Full Load, Major Principal Stress

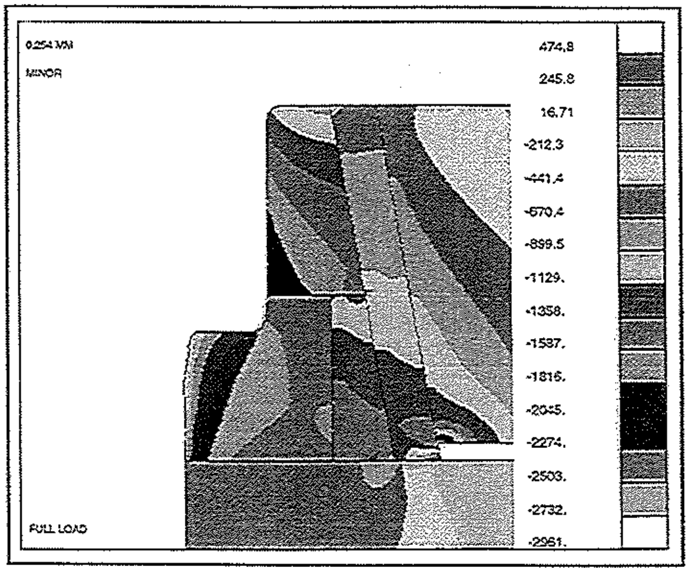

Figure A14 Inlet Tube - Interference 0.254 mm
Full Load, Minor Principal Stress

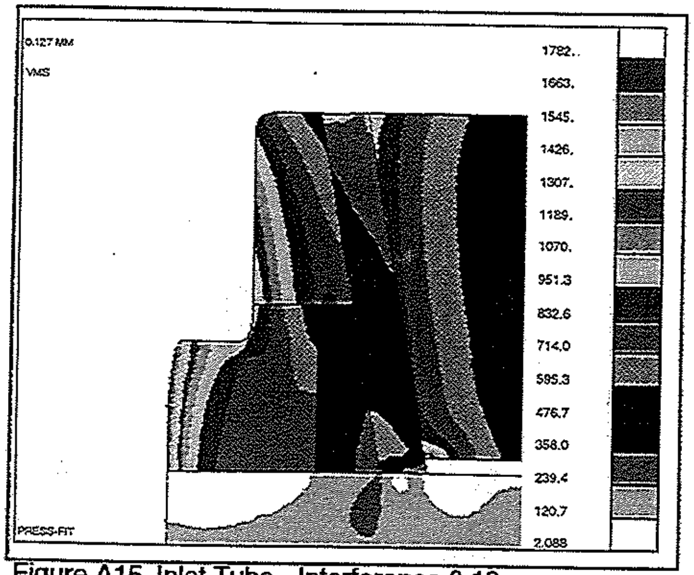

Figure A15 Inlet Tube - Interference 0.12 mm
PressFit, Von Mises Stress

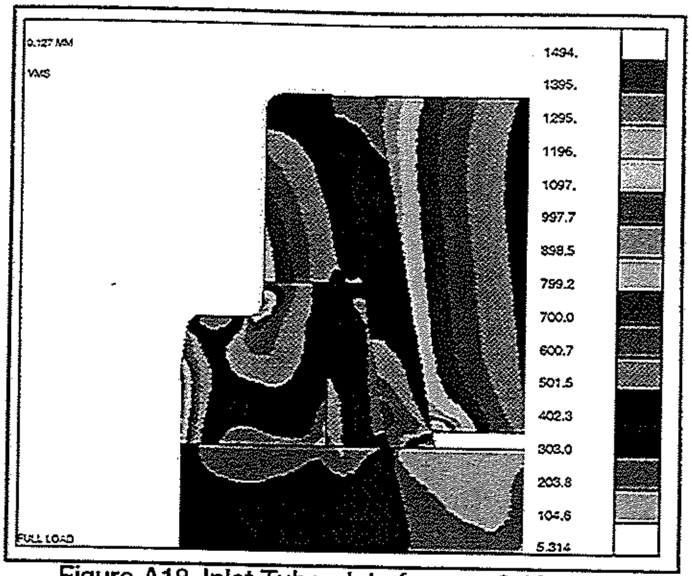

Figure A18 Inlet Tube - Interference 0.12 mm
Full Load, Von Mises Stress

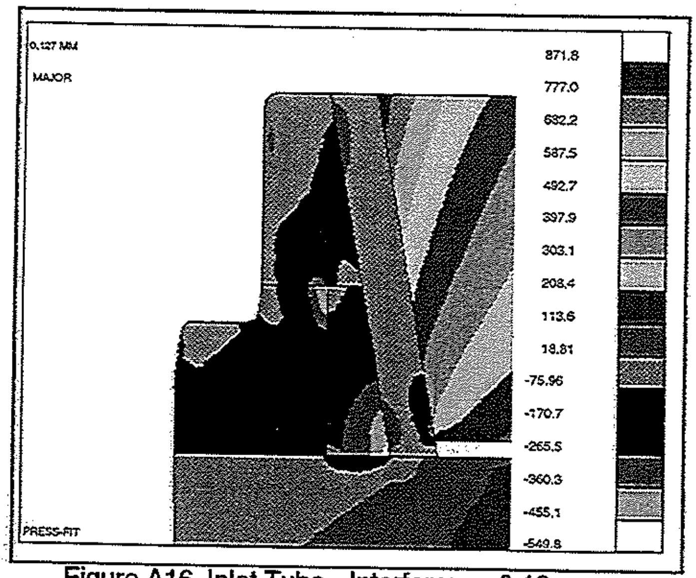

Figure A16 Inlet Tube - Interference 0.12 mm
PressFit, Major Principal Stress

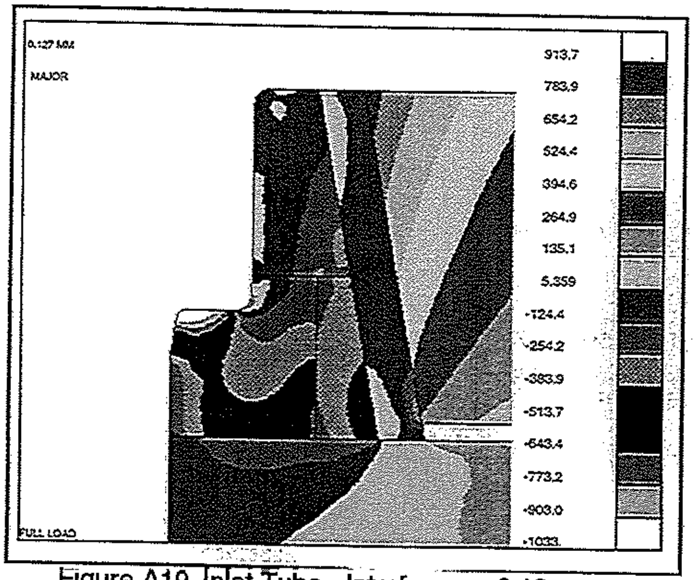

Figure A19 Inlet Tube - Interference 0.12 mm
Full Load, Major Principal Stress

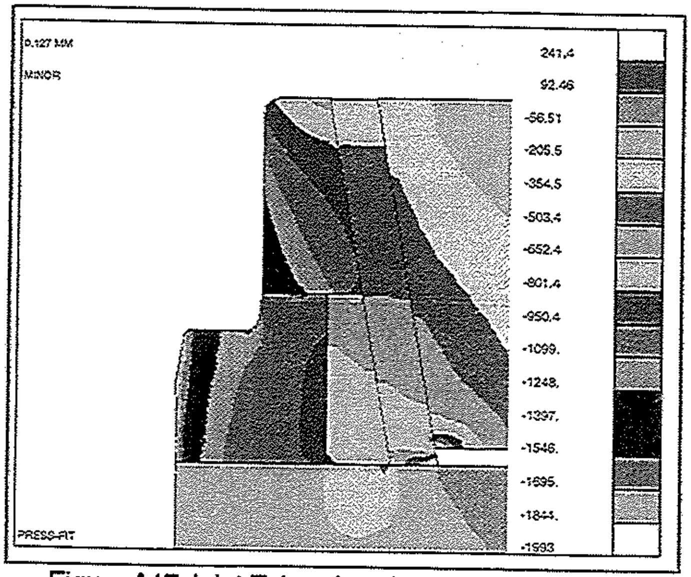

Figure A17 Inlet Tube - Interference 0.12 mm
PressFit, Minor Principal Stress

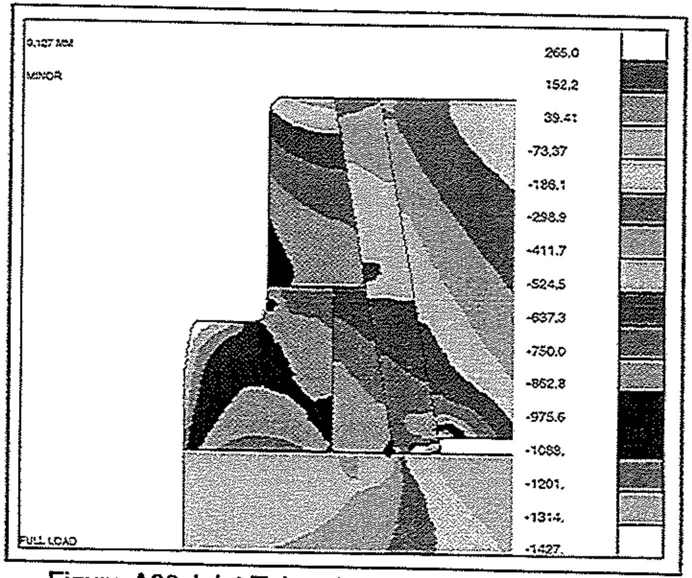

Figure A20 Inlet Tube - Interference 0.12 mm
Full Load, Minor Principal Stress

## APPENDIX B

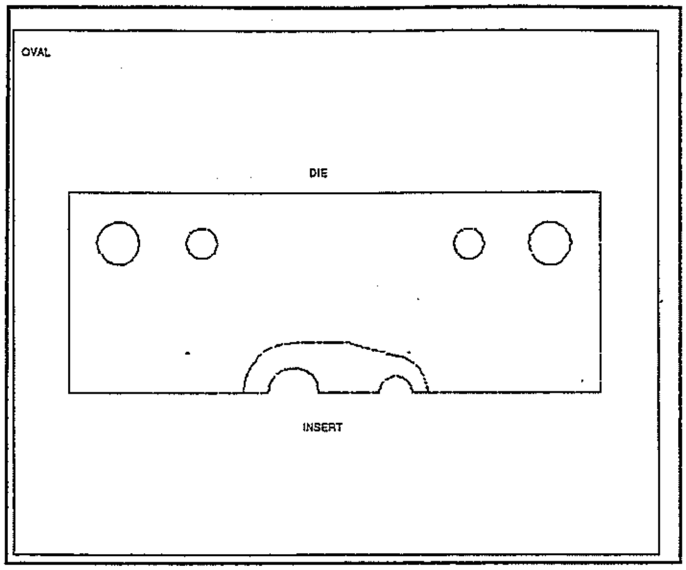

Figure B1 Miniblock Tooling - Oval Shape
Top View

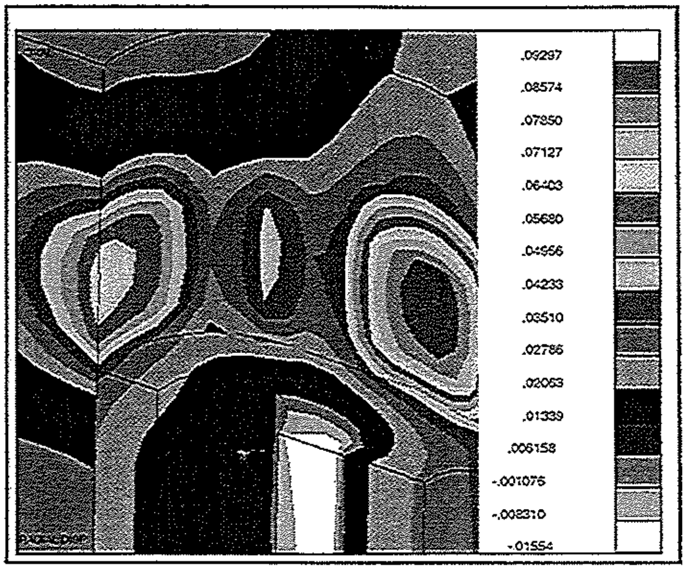

Figure B3 Miniblock Tooling - Oval Shape
Radial Displacement Plot

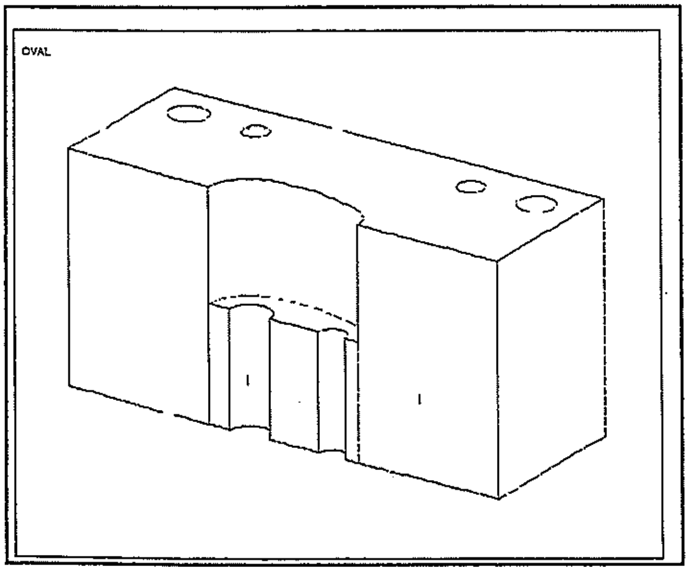

Figure B2 Miniblock Tooling - Oval Shape
Isometric View

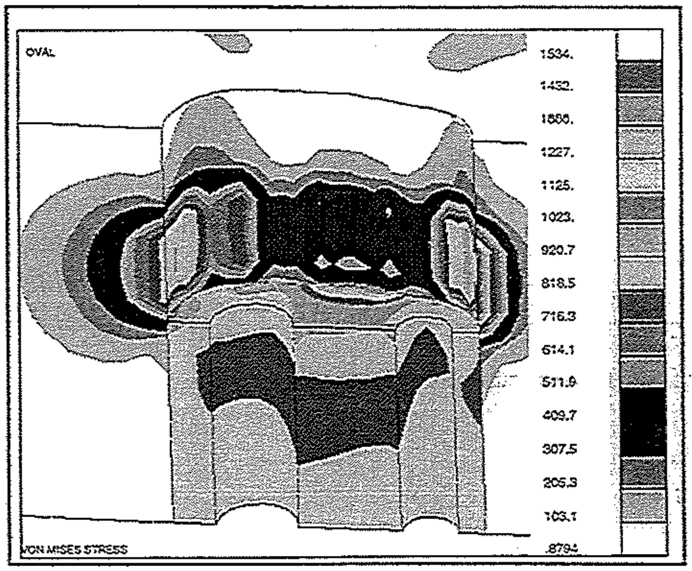

Figure B4 Miniblock Tooling - Oval Shape
Von Mises Stress Plot

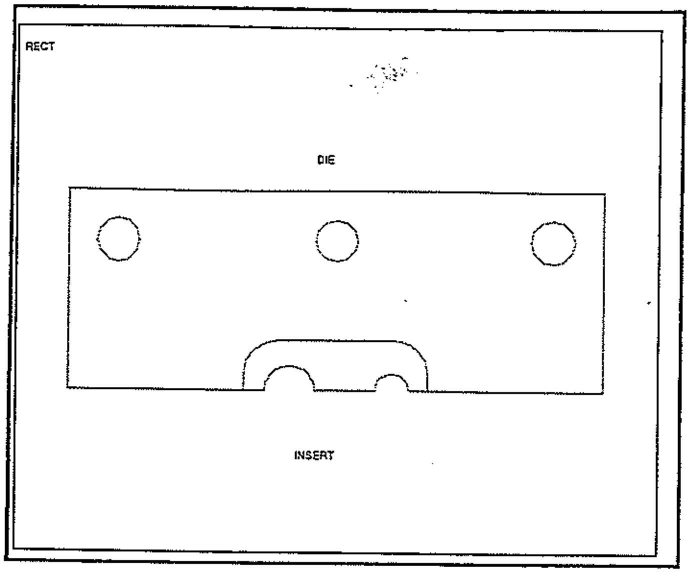

Figure B5 Miniblock Tooling - Rectangular Shape Top View

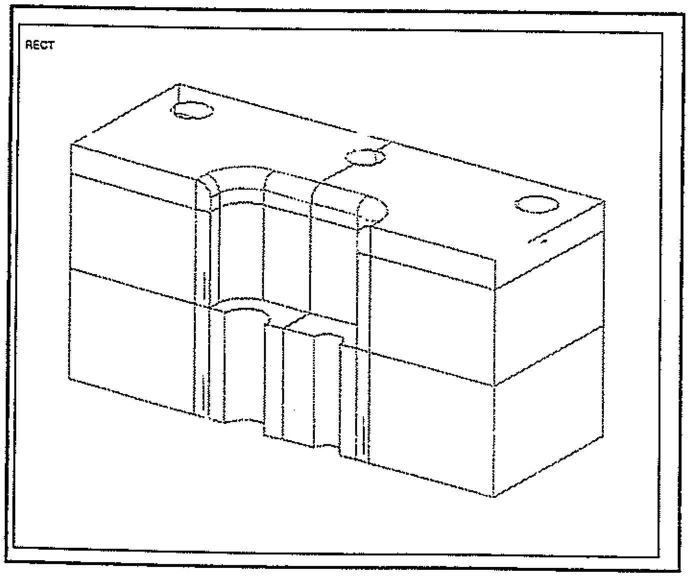

Figure B6 Miniblock Tooling - Rectangular Shape Isometric View

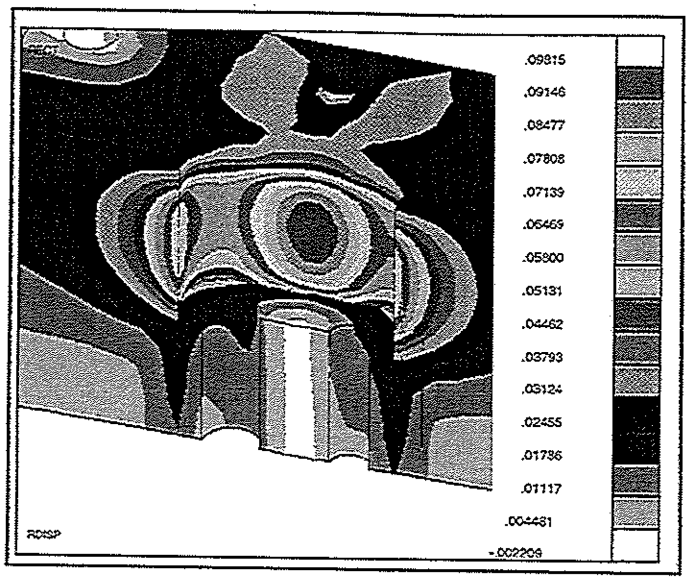

Figure B7 Miniblock Tooling - Rectangular Shape Radial Displacement Plot

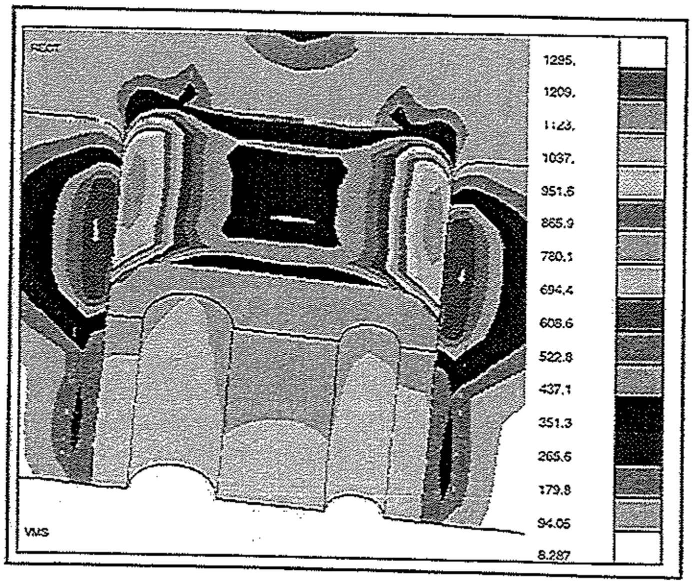

Figure B8 Miniblock Tooling - Rectangular Shape Von Mises Stress Plot