DESIGN ANALYSIS OF A FUEL INJECTOR PROTOTYPE
WITH THE AID OF FINITE ELEMENT MODELS

by

ADRIAN JOHN POST

B.A.Sc. (Mechanical Engineering), The University of British Columbia, 1996

A THESIS SUBMITTED IN PARTIAL FULFILMENT OF
THE REQUIREMENTS FOR THE DEGREE OF
MASTER OF APPLIED SCIENCE

in

THE FACULTY OF GRADUATE STUDIES

(Department of Mechanical Engineering)

We accept this thesis as conforming
to the required standard

THE UNIVERSITY OF BRITISH COLUMBIA

August 1999

©Adrian John Post, 1999
In presenting this thesis in partial fulfilment of the requirements for an advanced
degree at the University of British Columbia, I agree that the Library shall make it
freely available for reference and study. I further agree that permission for extensive
copying of this thesis for scholarly purposes may be granted by the head of my
department or by his or her representatives. It is understood that copying or
publication of this thesis for financial gain shall not be allowed without my written
permission.

Department of Mechanical Engineering

The University of British Columbia
Vancouver, Canada

Date August 22, 1999
Abstract

A fuel injector utilising High Pressure Direct injection technology is being developed by Westport Innovations Ltd. It is designed to allow heavy-duty diesel engines to run primarily on natural gas and yet still retain the inherent advantages of diesel engines. The operation of the injector is fairly complex and the cost of prototyping is high. Fatigue cracks were detected in injector components subjected to engine cycle testing.

To better understand the stresses in the injector and decrease the stress levels, several finite element models of the different components are created for analysis. Using submodelling techniques, the fine details of the design are examined at important sections of the injector. Alternative design choices are examined by creating alternative submodels with different hole intersection configurations and comparing the results to the original submodel. The results of this analysis indicate that more significant changes than altering the hole configuration are necessary if the stress levels are to be decreased.

The diesel and gas needle components are studied in a separate analysis in which contact elements are utilised to model the interaction with each other and the injector tip. The sealing surface at the contact between the components is predicted to be smaller than was previously assumed. Decreasing the relative angle between the contact surfaces results in an increase in the sealing surface.

The significance of thermal stresses as a cause of the fatigue failures is shown to be minor. A transient heat transfer analysis of the injector tip is performed. This predicts very small thermal variations once steady state had been reached.

The end of the tip is a critical area as the injection holes act as a stress concentration and the repercussions of failure of this section are great as this section of the injector is exposed to the engine cylinder. Different design alternatives for this particular section are modelled and compared to assess the relative merits of each configuration.
Finally, an investigation is conducted into the effect of pressurising the spring bore of the injector. Under the current conditions of the model, the deformation caused by this pressurisation will be unacceptable as there will no longer be sufficient clearance for the plungers in the cage and check block components of the injector.
Table of Contents

Abstract ........................................................................................................... ii

Table of Contents .......................................................................................... iv

List of Tables .................................................................................................. vii

List of Figures ................................................................................................. viii

Acknowledgement ......................................................................................... xi

1 Introduction ................................................................................................. 1

1.1 Problem Statement .................................................................................. 1

1.2 Objectives ............................................................................................... 2

1.3 Westport HPD Fuel Injector ..................................................................... 2

1.3.1 Background on Injector Technology ................................................... 2

1.3.2 The Westport Injector ......................................................................... 4

1.3.3 Discussion of Failures ......................................................................... 7

1.4 Research Plan ........................................................................................... 9

1.4.1 Brief Description of Analyses .............................................................. 9

1.4.2 Other Aspects of the Study ................................................................. 10

1.5 Submodelling Techniques ....................................................................... 11

1.5.1 Introduction ......................................................................................... 11

1.5.2 Submodelling Details ......................................................................... 12

1.5.3 Independent Verification of NFBC Technique .................................... 15

2 Analysis of Global Structure ..................................................................... 20

2.1 Introduction ............................................................................................. 20

2.2 Global Modelling Procedure .................................................................. 20

2.2.1 General Concept ................................................................................ 20
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.2.2</td>
<td>Boolean Operators</td>
<td>22</td>
</tr>
<tr>
<td>2.2.3</td>
<td>ANSYS© To NISA© Translation</td>
<td>24</td>
</tr>
<tr>
<td>2.3</td>
<td>Global Model Description</td>
<td>25</td>
</tr>
<tr>
<td>2.3.1</td>
<td>Geometry</td>
<td>25</td>
</tr>
<tr>
<td>2.3.2</td>
<td>Boundary Conditions</td>
<td>26</td>
</tr>
<tr>
<td>2.3.3</td>
<td>Temperature Distribution</td>
<td>29</td>
</tr>
<tr>
<td>2.4</td>
<td>Submodelling of Critical Area</td>
<td>30</td>
</tr>
<tr>
<td>2.4.1</td>
<td>The Vent Block Submodel</td>
<td>30</td>
</tr>
<tr>
<td>2.4.2</td>
<td>Fine Submodel</td>
<td>32</td>
</tr>
<tr>
<td>2.5</td>
<td>Design Modifications for Hole Configurations</td>
<td>34</td>
</tr>
<tr>
<td>2.6</td>
<td>Global Analysis Results</td>
<td>36</td>
</tr>
<tr>
<td>2.6.1</td>
<td>Overview</td>
<td>36</td>
</tr>
<tr>
<td>2.6.2</td>
<td>Critical area</td>
<td>38</td>
</tr>
<tr>
<td>2.7</td>
<td>Discussion of Error</td>
<td>40</td>
</tr>
<tr>
<td>2.8</td>
<td>Verification of Global Model</td>
<td>45</td>
</tr>
<tr>
<td>2.9</td>
<td>Conclusions and Recommendations</td>
<td>46</td>
</tr>
<tr>
<td>3</td>
<td>Component Analyses</td>
<td>47</td>
</tr>
<tr>
<td>3.1</td>
<td>Needle and Tip Interaction</td>
<td>47</td>
</tr>
<tr>
<td>3.1.1</td>
<td>Introduction</td>
<td>47</td>
</tr>
<tr>
<td>3.1.2</td>
<td>Static Analysis Assumption</td>
<td>48</td>
</tr>
<tr>
<td>3.1.3</td>
<td>Boundary Conditions</td>
<td>49</td>
</tr>
<tr>
<td>3.1.4</td>
<td>Comparison of 2-D and 3-D Models of Tip and Needles</td>
<td>51</td>
</tr>
<tr>
<td>3.1.5</td>
<td>Contact Surface</td>
<td>52</td>
</tr>
<tr>
<td>3.1.6</td>
<td>Contact Angle Variation</td>
<td>54</td>
</tr>
<tr>
<td>3.1.7</td>
<td>Conclusions</td>
<td>55</td>
</tr>
<tr>
<td>3.2</td>
<td>Transient Heat Analysis</td>
<td>55</td>
</tr>
<tr>
<td>3.2.1</td>
<td>Introduction</td>
<td>55</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Boundary Conditions</td>
<td>56</td>
</tr>
</tbody>
</table>
3.2.3 Results ........................................................................................................... 57
3.2.4 Conclusions ................................................................................................... 59

3.3 Stress Concentration at Holes in Tip.................................................................. 61
  3.3.1 Introduction ................................................................................................. 61
  3.3.2 Fatigue Crack and Failure Analysis ............................................................ 64
  3.3.3 Model Description and Boundary Conditions ............................................. 65
  3.3.4 Discussion of Design Options ................................................................... 66
  3.3.5 Results ....................................................................................................... 69
  3.3.6 Conclusions ............................................................................................... 72

3.4 Effect of Pressurised Spring Bore on New Design ............................................ 73
  3.4.1 Introduction ................................................................................................. 73
  3.4.2 Model Description and Boundary Conditions ............................................. 73
  3.4.3 Results ....................................................................................................... 75
  3.4.4 Analytical Check ....................................................................................... 78
  3.4.5 Conclusions ............................................................................................... 79

4 Summary and Future Work ................................................................................... 80

References .............................................................................................................. 84

Appendix A: The Finite Element Method .................................................................. 86

Appendix B: Other Mesh Refinement Techniques ................................................... 90

Appendix C: Procedure to Implement NFBC Submodelling Technique ................. 93
List of Tables

Table 1 Component Materials ........................................................................................................ 8.

Table 2 Summary of Results of NFBC Verification Analyses ...................................................... 18

Table 3 Load Case Pressures ......................................................................................................... 27

Table 4 Design Modifications and Features and the Effect on the Maximum Stress ..... 72
List of Figures

Figure 1 Westport HPD Fuel Injector and Manifold ........................................... 1

Figure 2 Injection Schematic ................................................................................. 3

Figure 3 Injector Component Cross-Sections ......................................................... 4

Figure 4 Design Pressures Over Engine Cycle ....................................................... 5

Figure 5 Pressure Locations ................................................................................... 6

Figure 6 NFBC Verification Model and Submodels .............................................. 16

Figure 7 Illustration of Tip Simplification ............................................................... 21

Figure 8 Examples of Complex Hole Intersections in the Injector ....................... 22

Figure 9 Example of Solid Model of a Component and Corresponding Meshed Model in ANSYS® ................................................................. 23

Figure 10 Interface for ANSYS® To NISA® Translation Code ................................ 25

Figure 11 Design Pressure Locations ................................................................. 27

Figure 12 Injector Nut ........................................................................................... 28

Figure 13 Rough Surface Vs. Smooth with Tetrahedral Elements ....................... 32

Figure 14 The Fine Submodelling Procedure ....................................................... 33

Figure 15 Original Hole Configuration ................................................................. 35

Figure 16 Alternative Hole Configuration #1 ....................................................... 35

Figure 17 Alternative Hole Configuration #2 ....................................................... 36

Figure 18 Crack Initiation Sites on a Failed Injector ........................................... 36

Figure 19 Von Mises Stresses In Global Model .................................................... 37
Figure 41 Exaggerated Deformation in Spring Bore ..................................................... 75
Figure 42 Displacement Contours in Spring Bore ....................................................... 76
Figure 43 Von Mises Stress in Spring Bore ............................................................... 76
Figure 44 Displacement Along Plunger Bore ......................................................... 77
Figure 45 Comparison of Analytical and F.E. Methods ....................................... 79
Acknowledgement

I would like to acknowledge the Natural Sciences and Engineering Research Council of Canada and Westport Innovations Ltd for their financial support and also for all of the guidance and input provided by the engineers at Westport Innovations Ltd.

I very much appreciate the invaluable assistance and supervision provided by both Dr. Mohamed Gadala and Dr. Philip Hill.

Also, I would be remiss not to mention my mother and father for all their quiet support and encouragement through my years of education.

Thank you to my wonderful wife, Colleen, for all her love and understanding when I returned to university and for any sacrifices she may have shared in my pursuit of this degree.
1 Introduction

1.1 Problem Statement

Increasingly strict environmental laws are forcing significant changes to be made in the field of diesel engine design. The emissions performance of current diesel engines will not be adequate under regulations coming into effect in the U.S. as soon as 2004. Conversion to alternative fuels such as natural gas is a method of retaining the use of these engines. Current methods of conversion involve, however, a considerable loss in engine performance. Westport Innovations Ltd. of Vancouver, British Columbia, is developing a fuel conversion system that would allow diesel engines to be run primarily on natural gas and still retain the inherent benefits of the diesel engine. The fuel injector is the heart of the new fuel system. It will be required to provide optimum performance and reliability and durability comparable to standard diesel injectors.

The purpose of this project is to achieve a better understanding and facilitate an improved design for the Westport fuel injector prototype though the use of finite element analysis. Stress analysis will be used to predict important areas of stress concentrations. Alternative designs will be modelled in order to quantitatively gauge the effect of various design modifications. The models will predict deflections and deformations and therefore possible areas of contact and wear.

![Figure 1 Westport HPD Fuel Injector and Manifold](image)

Although it is a powerful tool, the consideration of finite element modelling in the design of these fuel injector prototypes is primarily due to the lack of alternatives in terms of stress analysis of the design. It is very difficult to predict the stress state in the injector by analytical means due to the complexity of the geometry involved. Experimental methods are hampered by the small size of the components, the inaccessibility of the injectors when they are in operation, and the harsh
Chapter 1: Introduction

engine environment in which they are located. Injectors are currently tested in engine test cells but these tests reveal little about the actual stresses in the injector. These tests are designed primarily to assess the performance of the injectors with respect to efficiency and combustion properties.

1.2 Objectives
Although it is sometimes obvious that certain features will result in stress concentrations, it is less clear which stress concentrations are critical and what changes can be made within the design constraints to minimise such effects. The finite element models in this project are used to predict the important areas in the injector design and to quantitatively compare the effect of design modifications. Every effort is made to obtain realistic results although due to the uncertainty in the loading conditions and the lack of experimental or analytical verification, most analyses in this project can not be directly compared to experimental stress values in the injector. The analyses should provide a better understanding of the weakness in the design and indicate steps for improvement.

1.3 Westport HPD Fuel Injector
1.3.1 Background on Injector Technology
The fuel injector prototypes are unique in that they utilise High Pressure Direct (HPD) injection technology. This technology enables the user to employ gaseous fuels in engines designed for diesel fuels. These retrofit engines would then primarily consume natural gas as fuel along with a much smaller amount of diesel fuel. The small amount of diesel fuel injected into the engine cylinder with each cycle acts as a pilot fuel to aid in the ignition of the natural gas. The natural gas is then injected under high pressure (3000 psi) into the diesel combustion chamber near the end of the compression stroke. The fuels are injected from the same injector that is designed to inject the two fuels sequentially as shown in Figure 2.

As with ordinary diesel engines, this system does not require an additional source of ignition such as a spark or glow plug. This approach of direct injection is not a new concept but it has proven very difficult to develop from the conceptual stage. Despite research and testing by diesel
manufacturers, this technology has yet to be successfully implemented and commercialised. Westport Innovations is now leading the market in this technology and holds several proprietary patents [1].

There are other existing methods with which diesel engines may be modified to consume gaseous fuels. These methods fall into two classifications: spark ignition and dual fuel.

Spark ignition requires the use of a spark plug to initiate combustion. One of the main advantages to spark ignition is that the process is well understood due to much experience with spark ignition in gasoline engines. Also, emission performance of spark ignition systems is comparable to that of HPD systems. However, the major disadvantage of spark ignition is the loss of overall efficiency. Fuel economy drops by more than 30% with spark ignition over the equivalent diesel engine. The torque available at low speed is also much lower.

Dual fuel approaches consist of mixing air with the natural gas and then injecting diesel fuel. This system works well only at peak power generation and emission performance can be poor at idle or low speed. HPD Injection technology allows for a system with the cost and emission performance of natural gas with the performance and efficiency of diesel engines.

Environmental regulations are becoming increasingly more stringent. By the year 2004, regulations of the Environmental Protection Agency (EPA) in the United States will effectively eliminate all traditional 2-stroke diesel engines. This is a concern as diesel engines dominate
certain market niches such as heavy-duty commercial trucks and buses. Significant capital has been invested into these diesel engines and they have certain inherent advantages over gasoline engines. Diesel engines are typically 30% more fuel-efficient than spark ignition engines. They are also very robust and they operate for a relatively longer time with little maintenance. They also have the important ability to deliver high torque at low engine speed. The HPD fuel injectors are being designed so that existing diesel engines can be retrofit to run primarily on natural gas. Emission performance of these engines will be much better and yet they will still retain the advantages of the diesel engine.

1.3.2 The Westport Injector

The development of the Westport HPD Fuel System began over 10 years ago as a research project of Dr. Philip Hill at the University of British Columbia. Since then, Westport Innovations has taken over the development and marketing of this system with the co-operation of the
university. In September 1997, at the beginning of this specific project, the version of the injector prototype at that time was experiencing failure after only approximately 5 to 6 hours of engine testing. Three generations of the injector design later, the prototypes have survived over 1000 hours of testing due to improvements in the design and manufacturing.

The injector consists of 6 main components: the cage block, check block, vent block, tip, diesel needle, and CNG needle. In addition there are some secondary components such as the needle springs, alignment pins and plungers. These components are all shown in Figure 3.

![Design Pressures Over Engine Cycle](image)

*Figure 4 Design Pressures Over Engine Cycle*
The HPD injector is designed to inject both CNG and diesel fuels at set pressures and with a specific timing in order to achieve optimum combustion properties. Control is instituted within the injector using the hydraulic pressure of the diesel fuel. Since the injector is designed to replace existing diesel injectors, all of these features are incorporated into the same volume and shape as occupied by a regular diesel injector. Figure 4 shows the variation of the various input pressures over a typical engine cycle. The pilot diesel pressure is directly related to the control diesel pressure by a set ratio determined by the size of the plungers shown in Figure 3 and Figure 5. The control diesel pressure supplied to the injector increases linearly for approximately 1.5 ms during the engine cycle until it reaches a maximum pressure of 15 ksi (103 MPa). At this point, this pressure acting on the small differential area on the CNG needle creates a force sufficient to move the needle and compress the needle spring. This will open the seal between the tip and the CNG needle and allows the injection of the natural gas. The control diesel pressure will then
rapidly decreases to its initial level, allowing the CNG needle to seal again. The diesel pilot fuel is injected just before the opening of the seal between the tip and the CNG needle. The pilot diesel pressure will normally increase along with the control diesel pressure till it reaches a value of 10 ksi (69 MPa) at which the corresponding control diesel pressure will be approximately 6 ksi (41 MPa). Similar to the CNG cracking, the pilot diesel pressure acting on the differential area of the diesel needle creates a net upwards force and the diesel needle moves to allow the injection of the pilot fuel. Once the pilot fuel is released, the pilot pressure is relieved until the next engine cycle. The sealing diesel pressure remains constant throughout the engine cycle and is designed to keep the natural gas from leaking into the diesel pilot fuel. The CNG pressure also remains steady throughout the engine cycle.

Although these pressures are used in the design of the injector, they are much more complex in practice. Fluid effects such as pressure waves are difficult to predict and model but may be significant in the actual performance of the injector. Currently the fluid effects within the injector are being studied closely by others involved in the development of the injector.

1.3.3 Discussion of Failures

As discussed above, the original prototype of the injector failed after approximately five hours of testing in an engine test rig. Assuming an average speed of 1000-rpm, this would correlate to approximately 300,000 cycles. The tested components experienced failures due to cracking as well as excessive wear. The injector components were only produced in small numbers and only one full prototype had been tested for this length of time although due to mixing and matching, some components were tested up to nine hours.

Researchers of the Southwest Research Institute (SwRI) undertook a failure analysis of the prototype [2]. SwRI indicated that manufacturing difficulties were a significant cause of the problems with the injector. Specialised equipment and skills are necessary to produce the injector prototypes but the quality of the manufacturing was insufficient at that time. Much of the scuffing detected on the components was attributed to a poor grade lapping compound or insufficient
cleaning of the components between lapping steps. Recommendations were also made with respect to material choices and design specifications and tolerances.

M and H series Tool Steels were specified for each of the components of the injector. Table 1 indicates specifically what material was utilised for each component.

<table>
<thead>
<tr>
<th>Main Components</th>
<th>Materials Used</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cage Block</td>
<td>AISI H13</td>
</tr>
<tr>
<td>Check Block</td>
<td>AISI H13</td>
</tr>
<tr>
<td>Vent Block</td>
<td>AISI H13</td>
</tr>
<tr>
<td>Tip</td>
<td>AISI H13</td>
</tr>
<tr>
<td>CNG Needle</td>
<td>AISI M42</td>
</tr>
<tr>
<td>Diesel Needle</td>
<td>AISI M7</td>
</tr>
</tbody>
</table>

Table 1 Component Materials

The H class steels are designed for hot-work applications [3]. They are very resistant to deformation and wear at high temperatures. They also possess high resistance to mechanical and thermal shock and heat treatment deformation. H13 has the highest wear resistance of the 5% Cr. hot-work steels. It has high hardenability and can be hardened with a minimum amount of dimensional change. It also has good resistance to thermal fatigue. The need for high hardness, wear resistance, strength, and close tolerances in a high temperature environment make the H13 hot-work tool steel a good choice for the components of a fuel injector. H13 was primarily chosen as it could be surface hardened to a high degree and still retain toughness.

The properties of the H13 steel depend on the heat treatment of the steel. For the injector components, the steel was triple tempered with a specified hardness of 38-42 RHC. This treatment would result in a yield strength of approximately 187 ksi (1289 MPa) at room temperature or 146 ksi (1006 MPa) at 800°F (427°C). The plane-strain fracture toughness of this H13 steel is roughly 45 ksi-in$^{-1/2}$. 
Chapter 1: Introduction

In parts where a soft core was not necessary and the part could be through hardened, the M series steels were used, as they could be through hardened to a higher hardness (65-68 RHC). The M series steels are known as the 'high speed' tool steels. They are resistant to tempering at high temperatures, have good wear resistance and high hardness. Molybdenum is added to these steels in order to induce carbide formation and subsequently high hardness.

1.4 Research Plan

1.4.1 Brief Description of Analyses

Several different models are created in the course of this project in order to study different features or components of the design. These models may be grouped into five distinct categories. The first and largest model created represents the entire injector except for the diesel and compressed natural gas (CNG) needles. The analysis for this model is discussed in Chapter 2. Chapter 3 describes the remaining four analyses. These contain a number of models for various components of the injector. Certain simplifying assumptions are made to simulate the interaction between various components. The goal of these analyses is to be able to compare relative stresses among different features of the injector. Since the boundary conditions and results of the global model have associated uncertainty, with careful consideration, comparably accurate boundary conditions may be created directly on these models. The needles for example, would theoretically not interact with any other components than each other and the tip so it would be unnecessary to include any of the other components in the analysis. The effect of this assumption would be negligible at the contact locations between the needles and the tip. The error and uncertainty in each model will be discussed in more detail in the discussion of these models. In the following a brief outline of the five models and analyses is given.
Chapter 1: Introduction

1. Global Analysis

A coarse global model is used to identify a critical area in the vent block section of the injector. Submodelling techniques are then used to concentrate on this area to obtain more accurate results. Different design configurations are modelled to investigate if a significant reduction in the stress concentration may be achieved.

2. Needle and Tip

Two-dimensional (2-D) axisymmetric models of the needles and tip are created to study the interaction between the components using contact elements. The effective contact surface area between the CNG needle and the tip is examined as it is critical for sealing and the effect of changing the relative angle of the contact surfaces prior to contact is also investigated.

3. Transient Heat Transfer Analysis

The discovery of cracks in the tips of injector test specimens leads to a transient heat transfer analysis of the tip in order to determine if thermal fatigue stresses might be a significant cause of failure.

4. Stress Concentration at Holes in Tip

The cracks appear to originate at the inlet of a gas hole, so a three-dimensional (3-D) model of the injector tip is created to examine the stresses with the inclusion of the hole that is neglected for the 2-D analysis. A number of design modifications of the tip are also modelled to determine their effect on the maximum stress seen in this section.

5. Pressurised Spring Bore

Finally a model is created of the newest design of the cage and check block sections only in order to see the deformation which would occur in these sections should the spring bore be pressurised.

1.4.2 Other Aspects of the Study

An important consideration during the course of this project was the fact that the injector design is progressing independently of the results of these analyses. Although feedback is given
throughout, other design considerations result in many significant changes in the injector prototype. The prototype design progresses through 3 generations and alternative designs for different engine configurations are created. The finite element models adapt somewhat to the changes as they occur. The full injector analysis featuring the global model reflects the second generation of the design since the beginning of this project. Most of the different analyses feature this design although the most recent involves the latest configuration for the cage and check block. The implication of this is that the specific stress values determined by the analyses are less important than the stress effects that are noted. Even if the stress values are determined within a certain degree of uncertainty, changes in the design will most likely result in significant changes in these values. However, the design will not have changed so much that a design feature that acts to increase the stress in one model will still have the same effect in subsequent designs.

Experimental verification would supply important support and credibility to the finite element model results. However, most of the design data input that is available is a result of failure and wear analysis of the injectors after they have been removed from the engine test cell. A separate test rig designed to examine stress and deformation of the injectors would be difficult and expensive to create and the changes needed to accommodate measurement devices would cause the operating conditions to be far removed from the actual situation. Also, measurements could only be made at the outer surface of the injector, as the inner features would still be inaccessible. It should be noted that the same features that make it difficult to measure the stresses in the injectors also make it difficult to measure the pressures throughout the injector so that there is an uncertainty in the loading conditions in any type of analysis. A separate project was undertaken concurrently with this one whose aim was to predict the true pressure distribution within the injector.

1.5 Submodelling Techniques
1.5.1 Introduction

For large models with a coarse mesh, the accuracy in a local region of the model may not be satisfactory. In order to achieve better accuracy, a more refined mesh is necessary in the area of interest. It is inefficient and may not be possible (due to model size constraints) to use a
Chapter 1: Introduction

sufficiently refined mesh throughout the entire model. There are a number of techniques that can
be used to achieve this localised fine mesh such as transition elements, submodelling and
substructuring. Refer to Appendix B for more details regarding substructuring and other mesh
refinement methods. Appendix A contains a brief introduction to the finite element method.

The size of a model is limited by computer memory and storage capacity, time, and software
constraints. Although the numbers of elements, nodes, or degrees of freedom in the model are
good indicators of the size of a model, another important factor is the maximum wavefront of the
model. The element order has a significant effect on the wavefront value and commercial FEA
packages contain optimisation routines to reorder the element numbers to minimise the maximum
wavefront. In a more physical sense, the maximum wavefront is related to the number of
elements present along a cross-section of the model. The time to complete an analysis and the
memory required is proportional to the square of the wavefront size.

Most submodelling methods introduce certain approximations in the solution process. These
assumptions normally have little impact on the results and the accuracy with the addition of a
mesh refinement technique will usually still be much better than with the original coarse mesh. On
the other hand, one of the major benefits of these techniques is the potential to reduce the
computing power, resources and time needed to produce satisfactory results. The complexity of
different submodelling techniques varies greatly. For most applications, the use of commercial
FEA software is necessary. It is also important to note that some of the submodelling techniques
lend themselves better to the implementation of design modifications.

1.5.2 Submodelling Details

In the mesh refinement techniques of submodelling, a separate model of the critical area is
created with a more refined mesh. The results of the global analysis are used to create the
boundary conditions for the submodel. There are a number of different methods to achieve this
transfer of boundary conditions, two of which will be discussed briefly. This technique makes the
assumption that the effect of the change in stiffness between the submodel and the
corresponding section of the global model due to the mesh refinement would not significantly
change the response of the global model. The submodelling method is based on St. Venant’s principle: If an actual distribution of forces is replaced by a statically equivalent system, the distribution of stress and strain is altered only near the regions of load application. It is therefore important that the boundaries of the submodel are far enough away from the critical stress concentration so that the results at this location are reasonably accurate. The results of the submodel near the boundary cannot be used, as they will contain a high degree of uncertainty. If the nodes at the boundary of the submodel do not correspond with those at the same location of the global model, a method of interpolating the boundary conditions to the additional nodes is necessary. This step is the most critical as it is the primary source of uncertainty in this procedure but it also allows the creation of a much finer mesh in the submodel without the need for transition elements. Once the boundary conditions have been calculated from the global model, the submodel is analysed completely separately from the global model. An important benefit of this is that localised design alternatives in the submodel can be examined more easily. Different submodels may be created and run using the boundary conditions from the same global model. The same submodel can also easily be run with different mesh refinements in order to verify the adequacy of the refinement. Submodelling is more easily integrated with existing FEA code than other mesh refinement techniques, as it requires only the additional manipulation of results and boundary conditions. Some commercial codes have submodelling capabilities built into the package but even if they do not, usually the output can be used to calculate the boundary conditions separately and then input them back into the commercial code. In the following we describe two distinct methods of transferring boundary conditions from the global model to the local submodel that are utilised in this project.

(i) Displacement Method

This method of submodelling is often called the cut-boundary displacement method or the specified boundary displacement method (SBD) [6]. In this method, the displacements at the nodes on the boundary of the submodel are used to transfer the boundary conditions. If the location of nodes on the boundary of the submodel correspond to those in the global model, then this technique is exact. However, if the node locations correspond, then either the mesh of the
submodel is not very much more refined or transitional elements would have to be used. Usually there would be many more nodes on the boundary of the submodel due to the more refined mesh. The displacement method utilised by ANSYS®[6] interpolates the displacements from the nodes in the global model using the element shape functions. The trade-off with the use of this technique is an increase in the uncertainty of the results in the local areas near the submodel boundaries.

(ii) Nodal Force Boundary Condition Method

The nodal force boundary condition (NFBC) method uses the nodal force and moment results of the global model rather than nodal displacement data. The nodal data is extrapolated to the new nodes of the submodel by ensuring that force and moment equilibrium is satisfied in a global sense along designated cutting planes. This is a more literal utilisation of St. Venant’s principle than the displacement method. Both methods are based on this principle but the displacement method does not necessarily satisfy the global equilibrium equations on each cut plane. With the NFBC method, the forces and moments from the nodes on the coarse model are summed on these cutting planes and redistributed to nodes in the detailed submodel. An important benefit of this technique is that the nodal data can be transferred from a global shell model to a submodel formed from solid elements.

This redistribution takes place with three main steps for each cutting plane. First the new nodal forces are calculated simply by distributing the resultant forces from the global model evenly among the new nodes. This will introduce a resultant moment about the centroid of the cutting plane. In the second step, the resultant moment calculated from the global model is adjusted by the amount of moment generated in the first step. Then a set of linearly distributed forces is generated which produces this moment but no net resultant force. Finally in the third step, the principle of superposition is used to combine the nodal force results from the first two steps and these forces are applied to the submodel.
Chapter 1: Introduction

The creation of the distributed forces in step two is the key to the NFBC technique. It is a fairly complicated procedure involving a bounding box and an optimisation routine. Details of the method may be found in Reference [9].

1.5.3 Independent Verification of NFBC Technique

The NFBC method was developed, checked, and successfully applied to a previous project [9]. In the previous reference, two additional examples were included to further illustrate the use and accuracy of the method. The results of a global shell model of a core filter pipe used in the pulp and paper industry were transferred to a shell submodel with an identical mesh to the corresponding section of the global model. The maximum von Mises stress predicted at a location away from the boundaries of the submodel agreed with the results of the global model within 6%. The second example included a similar global shell model but the submodel was formed from solid elements and a much more refined mesh. In this case, the submodel would give a much better representation of the stress at the stress concentration in the models. However at specific locations chosen away from the stress concentrations and the submodel boundaries, the results of the global model and submodel still agreed within 8%.

Despite these encouraging results an additional independent test analysis was undertaken to better understand the implementation, application, and accuracy of the method before applying it to a problem in this project. A simple 3D test case was chosen as a suitable model for this evaluation. A linear static structural analysis was performed on a thin plate with a central hole although only one quarter of the plate was modelled due to symmetry. The ratio of hole diameter to plate width/length was small (1:50) so that the conditions approached the situation of an infinite plate. Therefore the results could be compared with the analytical solution that exists for this problem. A full model was run to determine the results from FEM without submodelling and then a cutting plane was introduced into the model to create a submodel. The values of the equivalent von Mises stresses at selected nodes were chosen for comparison.
Chapter 1: Introduction

Three separate submodels were created with different degrees of mesh refinement. The mesh of the first submodel was identical to that of the section of the full model it was replacing. The mesh of the second submodel contained 4 times as many elements as the first and the third submodel contained over 20 times as many elements. The mesh refinement has two distinct effects on the results. The refined mesh results in more accurate results locally and the increase in the number of nodes on the cutplane results in a better transfer of the boundary conditions. Details of the individual meshes and cutplanes are included in Figure 6.
Chapter 1: Introduction

The von Mises stress was calculated at various points in the different models for comparison. Table 2 lists and compares the stresses at three significant points at the hole and one internal point at the centre of the models (See Figure 5). The first comparison in the table is between the results of the full model and the analytical results (a) and then between the different submodels and the full model (b-d). The difference in the results of the analytical model and the full model can be explained as the analytical results are based on an infinite plate assumption and the mesh may not be sufficiently refined at the hole.

The results from submodel3 are very similar to those from the original full model except for the point at the compression zone of the hole. However, since the results of the analytical model and the full model also showed some discrepancy, it was postulated that much of the difference at the compression zone between submodel3 and the full model might be due to error in the full model due to the less refined mesh. This was further supported by the fact that the results of submodel3 were closer to the analytical results. In order to test this hypothesis, a full model with a more refined mesh (similar to submodel3) was created. The results of this model were generally closer to the analytical results (e) and very close to the results of submodel3 (f). At other points near the cutting planes of any of the submodels, the results are not very accurate. The algorithm satisfies the boundary conditions in a general sense and so the local effects near the cutplanes produce unacceptable errors in these areas. This effect of the NFBC submodelling routine is unavoidable due to the nature of the algorithm and so the cutplanes must be situated far enough away from the critical area of interest so that these local effects do not affect the critical results.
Table 2 Summary of Results of NFBC Verification Analyses

The results of these verification models indicate that under specific conditions, the NFBC submodelling technique can provide results with very little loss of accuracy. The NFBC algorithm is designed for the transfer of boundary conditions from one set of nodes to a much larger set of nodes on a cutplane. The increase in accuracy of the technique with the relative increase in nodes on the cutplane from the full model to the submodel is consistent with the findings of this
analysis. The models in this verification analyses represent a fairly simple geometry and the submodels utilise only a single cutplane. For models with more complex geometries and loading conditions and multiple cutplanes between the full model and the submodel, the error may be higher and compounded with each cutplane. However, with proper care in choosing the mesh configuration and cutting plane boundaries, the NFBC submodelling technique can be used to efficiently achieve good results.
2 Analysis of Global Structure

2.1 Introduction
The purpose of this analysis is to use a finite element model to improve the design of a fuel injector prototype. To achieve this, the stresses in the injector model are analysed to determine the local stress concentrations. A global model is created which includes all of the main components of the injector except for the diesel and CNG needles. Since the interaction between the needles and the other components is limited to a local area at the tip, it was decided to not include the needles in this analysis and include them in a separate analysis. Once a critical area has been identified, design modifications are modelled in order to predict how they will affect the stress state in the critical area.

Due to the loading situation (high cycle alternating loads) and the evidence of test failures, fatigue is the expected cause of failure. The optimal material and material treatments were investigated previously. The combustion properties and the nature of the design determine the pressures applied to the injector so that they can not be changed without a major change in the design of the injector. Since the design is largely constrained by the fact that the injector is designed to replace existing diesel injectors in retrofit, only relatively minor changes may be made to the design. The aim is to examine, therefore, the locations of maximum stress so that the design may be modified within the constraints to decrease the stress levels within the injector.

2.2 Global Modelling Procedure
2.2.1 General Concept
The injector prototype is modelled as a number of separate static (non-dynamic) systems. The idealised pressures over the engine cycle estimated by the designers of the injector are considered and the pressure cycle is decomposed into three static pressure 'snapshots'. These represent the critical pressure situations seen by the injector (See Figure 4). The validity of this assumption is discussed later in this chapter.

The first step in the creation of the model is to eliminate all of the non-critical features from the design drawings before creating the finite element model. Features that are removed from the
model include most fillets and chamfers as well as some larger design aspects. Slots are removed from the top face of the cage block and the alignment holes and pins as well as the relief ports in the vent block are ignored. Small valve seats are neglected and offset holes in the vent block are modelled as a single angled hole. The sealing groove is removed from the tip and the geometry of the end of the tip is simplified (see Figure 7).
2.2.2 Boolean Operators

First attempts to create the global model mesh used the 'Bottom-Up' method of mesh generation in the commercial finite element package NISA©[7]. In this method, the mesh is created manually by specifying points, lines, surfaces, and volumes that are eventually used to build geometric solids that define the boundaries of the elements generated. Unfortunately, even though the injector design is simplified, it is still not fully symmetric. Otherwise the global model would be created as a cyclic symmetric model that would be significantly smaller than a full solid model. Some of the less complicated components such as the cage and check blocks and the tip were meshed separately in NISA©. However it was not possible to join these meshed components as the elements and nodes at the interface boundaries did not correspond. Although a concerted effort was made to model the vent block component, the complexity of the geometry (as shown in Figure 8) proved too difficult to model with this technique.

![Figure 8 Examples of Complex Hole Intersections in the Injector](image)

At this point an alternative commercial finite element package (ANSYS©) was investigated as it provided the ability to model from the top down although this feature is not unique to ANSYS©.

The 'Top-Down' method of mesh generation refers to the ability to create solid models using solid shapes and Boolean operators. Boolean operations include such operations as addition, subtraction, and intersection, etc. Once a solid model is created, automeshing algorithms are used to produce a mesh that matches the solid geometry (See Figure 9). This method has some significant advantages. Solid shapes may be created and then added or subtracted (among other Boolean operations) to each other to produce more complex solid shapes. Specifically, this makes the creation of the hole intersections in the vent block much simpler. The holes are simply
modelled as solid cylinders and then subtracted from the larger solid shape to create the holes. The automation of the mesh generation also allows for optimisation of the mesh as it is created which helps create a more uniform mesh. The resultant mesh must be carefully inspected, however, as the user has not specifically created the mesh as in the previous method. Thus, the global model is created using the ‘Top-Down’ solid modelling approach available in ANSYS®.

Once the solid modelling techniques are used to create a solid geometric model, then automeshing techniques are used to convert the solid model geometry into a finite element mesh. For such an irregular geometry, the automeshing algorithms of ANSYS® require the use of tetrahedral elements to create a solid model. Since the model is represented by many curved lines, 10 node quadratic tetrahedral elements are chosen rather than the alternative 4 node linear tetrahedral elements.

![Solid Model and Meshed Model](image)

Figure 9 Example of Solid Model of a Component and Corresponding Meshed Model in ANSYS®

Even with the above mentioned simplifications from the original design drawings, the number of elements and nodes needed to produce the global model is much larger than the limits set by the license agreement between ANSYS® and the university. The number of nodes and elements available for a single model are limited to 32,000 nodes. When the global model is complete, it requires over 55,000 nodes. The version of NISA® available in the U.B.C. Finite Element Laboratory has no such restriction but the solid modelling capabilities of ANSYS® are needed to
create the model. In order to solve this problem, a translation program is created which will translate nodal and element information from a mesh created in ANSYS© into a format which can be input into NISA©. Small parts of the global model can then be modelled in ANSYS© and then translated into NISA© where they can be merged into a single model.

2.2.3 ANSYS© To NISA© Translation

A translation program was written that converts element and node data from ANSYS© into a format that could be read by NISA©. Since this program will be used many times, a user-friendly interface was designed to facilitate its use (See Figure 10).

The most critical part of this procedure is the merging of the meshed sections. The nodes of adjacent sections must be in exactly the same position before they can be joined. To achieve this, the interface boundaries of the meshed sections are meshed with quadratic shell elements. The shell elements share the same node locations as the faces of the solid elements at the boundary. The solid elements are then deleted so that only the shell elements remained. The next section can then be meshed without exceeding the node or element restrictions mentioned previously. The nodes on the interface boundaries with shell elements are shared with the shell elements and therefore continuity is ensured. The shell elements are deleted before exporting the node and element lists and then the procedure is repeated for each section to be meshed. Once all of the meshed sections are imported into NISA©, the nodes on the interface boundaries are merged.
2.3 Global Model Description

2.3.1 Geometry

Once completely assembled and merged into a single model, the number of nodes in the global model is over 55,000 and the number of elements is more than 34,000 for a total of over 165,000 degrees of freedom. Not only are the meshed sections of the same components merged but they are also merged with the meshed sections of the other components to create a single model. This action assumes that the various components do not move relative to each other and essentially act as if they are manufactured as a single component. This assumption unfortunately makes it impossible to analyse the interface between components that might be of interest in terms of
wear and sealing. However, the areas away from the interfaces are largely unaffected by this assumption.

The alternative to this assumption requires that contact or gap elements be employed at the component interfaces. The size of this model already makes it difficult to run as a static linear model without exceeding the hardware constraints of the machine. The inclusion of contact elements requires a non-linear analysis and many contact elements would be needed to cover all of the component interfaces. Such an analysis requires more memory and hard drive space than can be provided with available workstations. Based on experience with other large models, the analysis would also require many days to run and although such an investment in computing time may be made once, in reality the analysis would have to be run many times in order to deal with the usually inevitable changes and corrections to the model and the iterative nature of the analysis. If an analysis of the interface were warranted, a submodel of the specific interface would be a much more efficient use of resources.

2.3.2 Boundary Conditions

The boundary conditions applied to the model include pressures and displacement constraints as well as nodal temperatures. The pressures applied to the injector are divided into three separate load cases, each representing a different time snapshot within the engine cycle. The model is run separately for each load case. There are four main pressures defined for each load case as shown in Table 3 and Figure 11. The natural gas pressure remains constant throughout the engine cycle at approximately 3000 psi (21 MPa). The diesel fuel sealing pressure that is used to prevent the natural gas from bleeding into the diesel fuel that controls the lifting of the injection needles is also kept at a uniform pressure of 3500 psi (24 MPa). The diesel fuel control pressure will increase from a pressure of 50 psi (0.3 MPa) to a maximum of 15000 psi (103 MPa). The diesel pilot fuel injection pressure will vary from 3500 psi (24 MPa) to 10,000 psi (69 MPa). Load case three is the worst case of the three as it represents the highest overall pressures with the exception of the pilot pressure.
Chapter 2: Analysis of Global Structure

<table>
<thead>
<tr>
<th></th>
<th>Load Case 1</th>
<th>Load Case 2</th>
<th>Load Case 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>CNG Pressure (psi/MPa)</td>
<td>3000</td>
<td>3000</td>
<td>3000</td>
</tr>
<tr>
<td></td>
<td>(21)</td>
<td>(21)</td>
<td>(21)</td>
</tr>
<tr>
<td>Sealing Diesel Pressure (psi/MPa)</td>
<td>3500</td>
<td>3500</td>
<td>3500</td>
</tr>
<tr>
<td></td>
<td>(24)</td>
<td>(24)</td>
<td>(24)</td>
</tr>
<tr>
<td>Control Diesel Pressure (psi/MPa)</td>
<td>50</td>
<td>6030</td>
<td>15000</td>
</tr>
<tr>
<td></td>
<td>(0.3)</td>
<td>(42)</td>
<td>(103)</td>
</tr>
<tr>
<td>Pilot Diesel Pressure (psi/MPa)</td>
<td>3500</td>
<td>10000</td>
<td>3500</td>
</tr>
<tr>
<td></td>
<td>(24)</td>
<td>(64)</td>
<td>(24)</td>
</tr>
</tbody>
</table>

Table 3 Load Case Pressures

The injector components are constrained by a nut (Figure 12), which surrounds the entire injector and is threaded to accept the bolt load.
Chapter 2: Analysis of Global Structure

The components are stacked upon each other and the shoulder of the tip component contacts this nut. The model components are already attached to each other and the constraint at the tip is modelled by nodal constraints at the contact surface, which fix this surface in 3-D space and prevent rigid body motion.

At the top of the injector, a threaded bolt supplies a load to the top face of the cage block component. This load in conjunction with the nut contact at the tip is what maintains the alignment and sealing between the stacked components of the injector. The magnitude of the axial load on the face of the cage block due to the bolt load is roughly calculated from the bolt torque as follows:

\[ F_{axial} = \frac{T}{(0.2 \cdot d)} \]  

(2.1)

where \( T \) is the measured bolt torque and \( d \) is the thread diameter. The pressure is then determined by dividing this force by the area of the top face of the injector.

Therefore, in the model a pressure of 10 ksi (69 MPa) is applied to the element faces on the top face of the cage block. The bolt load more closely represents a displacement constraint rather than a pressure load, as the pressure at the top face will increase during operation due to thermal expansion. A load constraint will not result in this increase in pressure, as the top face will be free to expand past the point where the bolt will provide constraint. In order to predict this displacement load, the pressure load calculated above is applied to the model and the average displacement of the top face due only to the bolt load is calculated as \( 10.8 \times 10^{-4} \) in (0.027 mm).
Chapter 2: Analysis of Global Structure

For the remainder of the analysis, the bolt pressure load is substituted by this displacement constraint.

In order to determine the nodal temperatures to be input into the structural model, a preliminary heat transfer analysis is run using the global model. The temperatures output from this analysis provide the input for the thermal effects in the structural model.

2.3.3 Temperature Distribution

Nodal temperatures are determined using a simple heat transfer analysis with very simple temperature boundary conditions. The same model is used as created for the stress analysis but the elements are automatically altered to become heat transfer elements. A good estimate of the temperature of the gas in the engine cylinder is supplied and for this analysis, the exposed outer surface of the tip of the injector is assumed to have the same temperature. The temperature of the water jacket surrounding the body of the injector as a coolant is known to be a fairly steady 80°C. Therefore this temperature is assumed to be the temperature at the outer surface of the body of the injector.

The following relationship is calculated in order to determine if the fluctuating temperature at the tip will produce a fluctuating temperature gradient in the injector or if the temperatures will remain fairly constant once the system reaches steady-state. The following rough non-dimensional calculation based on the Fourier number supports this assumption.

\[ \delta = \sqrt{\alpha \cdot t_c} \]  

(2.2)

where:

\( \delta \) represents the distance travelled by the temperature wave;

\( \alpha \) is the thermal diffusivity which, for steel, is approximately \( 12 \times 10^{-6} \) m\(^2\)/s;

and \( t_c \) is the time in which the temperature wave travels.
For a time of 30 ms which corresponds to the approximate time for one engine cycle to complete, the distance travelled by the temperature wave is only \(\sim 0.6\) mm. Therefore transient effects are limited to the area very close to the engine cylinder.

Although this is a rough approximate check, it clearly indicates that steady-state is a good assumption for this heat transfer analysis and so transient effects are ignored. As other temperature effects are neglected, the result of this analysis is a smooth temperature gradient, which decreases from the hot tip to the cooled body of the injector.

The absence of accurate and detailed temperature related boundary conditions means that there is some degree of uncertainty with the temperatures predicted by this analysis. However, relevant experimental temperature measurements are not available (for the same reasons as mentioned for pressure data) and therefore introducing more complex boundary conditions such as convection coefficients and contact elements is unnecessary, as they would not improve the accuracy of this analysis. Furthermore, due to the steady-state condition of the temperature, the thermal loads are a steady load and so do not contribute significantly to thermal fatigue. Once the nodal temperatures are determined as a result of this analysis they are input into the stress analysis model and, with the specification of a thermal expansion coefficient, introduce thermal stresses to the model.

2.4 Submodelling of Critical Area

2.4.1 The Vent Block Submodel

Although the global model is large and contains many elements, the element mesh is still fairly coarse. The global model is adequate in providing a general sense of stresses and strains within the model but the coarseness of the mesh introduces additional uncertainty into the model. A finer mesh, especially at important areas, is needed to provide more accurate stress data. The results of the global model analysis are discussed in further detail in this chapter.

The first submodel is created for the vent block section. This model consists of more than 29,000 nodes and 18,000 elements for a total of over 89,000 degrees of freedom. The results of the global model analysis indicate that the critical area is within this vent block section. This
submodel is created in a manner similar to that of the global model as it is meshed first in ANSYS© and then translated to NISA©. The mesh in the submodel is more refined than the mesh in the vent block section of the global model, especially surrounding the area around an important hole intersection. The vent block is chosen as a submodel because the boundaries between the vent block and the cage block and tip components provide natural cut-plane boundaries for the submodel. Cut-planes may not be chosen arbitrarily due to the nature of tetrahedral elements (See Figure 13).

The NFBC algorithm method is first attempted to convert the results of the global model into boundary conditions for the submodel. However, as discussed previously (Section 1.5.2), the NFBC algorithm is designed to transfer nodal data from a relatively small number of nodes (<100) on a cut-plane boundary of a global model to a much larger number of nodes on the same cut-plane of the submodel. In this case, on the cut-plane dividing the vent block from the check block of the global model, the number of nodes in the submodel is very similar to that of the global model. The same is true of the cut-plane between the vent block at the tip section. Consequently, the NFBC algorithm is not ideally suited to this application. As a result of the similar number of nodes on the cut-planes, the algorithm is unable to adequately balance the forces and moments on each cut-plane.

Since the number of nodes on the cut-plane of both models are similar, it was decided to recreate the submodel so that the nodes on the cut-plane are identical to that of the global model. This is achieved by the shell element method described previously (Section 2.2.3). Although the mesh is similar to the global model at the cut-planes, the submodel still contains many more elements than the same section of the global model as the mesh is more refined elsewhere in important areas. Now that the nodes on the cut-plane boundaries between the global model and submodel have an approximately one-to-one ratio, the nodal displacements resulting from the global analysis may be transferred directly to the associated nodes of the submodel as boundary conditions. The only remaining difficulty is matching the nodes from the global model with their counterparts in the submodel.
2.4.2 Fine Submodel

The submodel of the vent block is still a fairly large model and the decision was made to create a further submodel of the submodel, which for brevity was called the 'fine-submodel'. Figure 14 shows the submodelling procedure from the submodel to the fine-submodel. The fine-submodel consists of a wedge shaped section of the submodel that encompasses the critical hole intersection. The boundaries of the fine-submodel are only far enough away from the hole intersection to allow the local effects of the boundary condition transfer to diminish so as to be insignificant at the hole intersection. The solid model and mesh of the submodel were recreated in order to achieve planar surfaces at the boundaries of the fine-submodel as otherwise the automeshed tetrahedral elements would never yield a smooth surface for a cut-plane except at the solid model boundaries (Figure 13). The fine-submodel consists of over 15,000 nodes and 10,000 elements for a total of more than 46,000 degrees of freedom.

Figure 13 Rough Surface Vs. Smooth with Tetrahedral Elements

The fine-submodel may be created directly from the global model but this would require a finer mesh for the global model. Not only would it be more difficult to recreate the entire global model than it would the submodel, the changes would increase the number of elements and nodes of an already large model. Aside from allowing for the creation of a more refined mesh at the critical
Figure 14 The Fine Submodelling Procedure
area, a very important aspect of the fine-submodel is that it greatly facilitates the simulation of alternative hole configurations arising during the iterative design modification phase.

Since the number of nodes on the cut-planes of the fine-submodel are much greater than those on the corresponding cut-planes of the submodel (e.g. 303 nodes from 55 nodes), the NFBC algorithm is used as this was the type of application for which it was designed. The procedure to implement the NFBC algorithm involves many steps that must be repeated for each cut-plane separating a submodel from a global model. In order to reduce the implementation time and the possibility of mistakes during this procedure, a special software code was written to aid in automating most of these steps. The details of this procedure are outlined in Appendix C.

2.5 Design Modifications for Hole Configurations

Since the configuration of the hole intersection in the fine-submodel creates a stress concentration that results in the highest stress predicted in the injector model and this is an area of previous failures, alternative hole configurations are modelled to see if this maximum predicted stress may be decreased. Manufacturability is also a concern when considering alternative hole configurations. The position of the opposite ends of the holes may not be changed and so the number of options for alternative hole intersections is limited.

The first alternative considered increases the diameter of one of the holes and decreases the diameter of the other to significantly change the geometry of the intersection. In the original design, the horizontal hole is drilled through from the outer radius of the injector and then plugged with a weld. In this case the same procedure may be followed or the horizontal hole may be formed by an Electro-Discharge Machine (EDM) from the inner radius.
The second alternative that was modelled eliminates the hole intersection completely by drilling on an angle through the component. However, this approach creates other stress concentrations at both ends of this angled hole. Specifically, new areas of concern are now the intersection of the angled hole and the inner groove in the injector and at the upper interface of the hole.
Both of these hole configurations are modelled and the same boundary conditions are applied as are applied to the fine-submodel using the NFBC submodelling procedure. The results of these models and the comparisons to the original hole configuration are given in the next section.

2.6 Global Analysis Results

2.6.1 Overview

The results of the global model indicate that the highest stress is located at a hole intersection in

Figure 18 Crack Initiation Sites on a Failed Injector
Chapter 2: Analysis of Global Structure

Figure 19 Von Mises Stresses in Global Model

Figure 20 Von Mises Stresses in Submodel
Chapter 2: Analysis of Global Structure

the passage for the high-pressure diesel control fuel (See Figure 19). This intersection is located in the vent block component of the injector. This result is consistent with the failure analysis of an injector test specimen, which failed as a result of fatigue cracks initiating at this location (Figure 18) [2].

Although this agreement in prediction of the critical area of the model supports the FE results, the actual values predicted by the global model may not be accurate due to the coarseness of the mesh. The submodel representing the vent block component contains more elements than the same section of the global model but is still a fairly coarse model. The displacement technique used to translate the boundary conditions from the global model to the submodel does not introduce any significant error. The only assumption made with this technique is that the global response at the boundaries of the submodel are the same regardless of whether the mesh in the vent block is the same as that in the global model or as in the submodel (See Figure 20).

2.6.2 Critical area

The fine-submodel consists of the local area around the critical hole intersection, which it models with a much more refined mesh than that in the submodel. The NFBC algorithm used to translate the boundary conditions in this case is not as exact as the previous displacement method but does allow the creation of this fine mesh. Since there is a large difference in the number of nodes at each cutplane, experience has shown that the NFBC yields good results. The main source of uncertainty is associated with the use of the idealised pressure loads. Maximum von Mises stress predicted by this fine-submodel reaches a value of 76.4 ksi (527 MPa) (See Figure 21).

The alternative hole configuration designs modelled in place of the original fine-submodel were discussed previously. The results of the analyses of these models are shown in Figure 21. The second alternative hole configuration predicts a maximum stress that is slightly higher than the original (+1.0%) whereas the configuration of the first alternative predicts a maximum stress that is slightly lower (-4.6%). Although this indicates that the hole configuration of first alternative is a better alternative to the existing configuration, the difference between the two predicted stresses
Figure 21 Von Mises Stresses in Alternative Hole Configurations
is not large enough to confidently declare one as clearly superior. The results of this analysis may not be strong enough, therefore, to warrant a design change. It may be concluded from this analysis that more significant changes to the design than are limited to simply changing the hole configuration are necessary if this type of failure is repeated. Although there were other injector failures during testing, the expense, time and resources necessary to manufacture an injector prototype as well as conduct an engine durability test means that there is not a large sample of injector failures to compare. The material and failure analysis was conducted on only one of each injector components along with an untested component. For these reasons, it cannot be assumed that the failure mode this analysis is trying to avoid is positively a repeatable failure.

The benefit of a fatigue and fracture analysis in this situation is questionable. The static results we have discussed are suitable for relative comparison only as the actual values are highly dependant on the idealised loading conditions used in the model. Additional assumptions associated with the fatigue and fracture aspects such as crack growth, shape and direction only increase the uncertainty in the analysis. It was, therefore, decided not to perform a FE fatigue and fracture analysis and rather spend the effort in comparing more design modifications via static analysis results.

2.7 Discussion of Error

The potential sources of error in the above analysis are grouped under the following three headings:

**Error Due to Modelling**: related to any differences between the actual injector and the injector model; **Error Due to Boundary Conditions**: related to any differences between the actual injector environment and operating conditions and the injector model boundary conditions; and finally, **Error Due to Analysis**: related to features of the numerical finite element analysis.

In the following, we briefly discuss the above sources of error.

1. Errors Due to Modelling

The simplifications of the design in order to create the global model (i.e. removal of alignment holes) should only have a local effect on the global model and should not significantly affect the
Chapter 2: Analysis of Global Structure

final local results of any submodel. Features removed from the global model that may possibly have an effect at the hole intersection are included in the fine-submodel.

Manufacturing or material flaws are neglected completely in this analysis. Every component is assumed to be created with zero tolerances and the material is completely homogeneous and isotropic. It is important to note that these may be the most significant factors in the failure of the component. Flaws and manufacturing defects may be modelled but it is difficult to predict where these flaws will occur and what exactly they will look like. Burrs and other flaws can be stress risers, crack initiation sites, or cause significant wear. For very large components, these types of factors were much less significant but with the very small scale of the injector components, manufacturing quality becomes very important. This could potentially be a significant source of error but it is difficult to quantify due to its unknown nature. Theoretically the needles do not contact with the components included in the global model except at the sealing surface at the tip component and so the needles are neglected from this part of the analysis. However, in reality there will be some contact and scuffing between the components. This could be significant in a separate analysis of the needles but does not affect the critical area framed by the fine-submodel.

The components of the injector are all created with ideal dimensions and tolerances are ignored. Since the tolerances in the injector are small and assuming they are properly specified to avoid compression fits and ensure good assembly, they do not have a significant effect on the results. However if the manufacturing tolerances are greater than specified on the design drawings, they may render to be significant. A tolerance study may help predict which tolerances might be critical should they not be achieved and the possibility of such an analysis was discussed with Westport Innovations. The effect of this source of error is negligible as long as the design tolerances are achieved in manufacture.

The merging of the separate components in the global model means that the components interface perfectly and do not move relative to each other. In reality the interface between the components is a source of leakage if the components do not mate and seal perfectly. Since these
interfaces are not in the same local area as the critical section, this is a reasonable assumption in the analysis of the fine-submodel.

2. Error Due to Boundary Conditions

The largest degree of uncertainty in this analysis most likely results from the use of the idealised pressures as boundary conditions. Although the pressures shown in Figure 4 represent the best estimate of the actual pressures at optimum operation, these pressures are largely dependent on the operation of the injectors. If the needles are not lifting as high as expected, the relief port that serves to decrease the control pressure may not be fully exposed and much higher pressures may result. From the limited experimental pressure data taken from the injector, measurements indicate that the maximum pressures may be more than 50% higher than the expected values used as input in the model [15]. Other fluid effects such as pressure waves are currently being investigated by Westport through advanced fluid modelling techniques. Due to the complexity of the operation of these injectors, it is very difficult to predict the pressures in the injector except under specifically defined conditions that may or may not represent the current operation of the injectors. As the injectors are prototypes, they are continually evolving and changing so that their operation is changing as well, increasing the difficulty of obtaining consistent experimental data.

Aside from the pressure values used, there is also uncertainty in the location of the pressures. Leakage is not taken into account and a linear pressure distribution is assumed along the inner bore of the injector between one set pressure and the next. Considering these factors with input from Westport Innovations [15], this source of error could produce an uncertainty of more than 50%.

There is also significant uncertainty in the specified nodal temperatures used to create the thermal stresses. The thermal stress is basically a steady stress, as the temperatures in the injector do not vary significantly once steady-state is reached. The largest sources of uncertainty with respect to the temperature inputs are the temperatures in the engine cylinder and the convection coefficient values used in the heat analysis model. However, the results of the steady-state heat analysis indicate that the effect of the engine cylinder temperature is limited to the tip of
the injector. The temperature of the remaining body of the injector is consistent with the
temperature of the coolant fluid surrounding the injector. This temperature is known as it is easily
measured.

A sensitivity analysis was performed to examine the effect of a change in the input temperatures
on the resultant stresses in the global model. The global steady-state heat analysis was repeated
with the input temperatures increased by 10%. The nodal temperatures determined from this
analysis are used as input for thermal effects for a global stress analysis. The resulting von Mises
stresses are compared to the original global analysis results at specific locations in the model.
The maximum increase in stress measured in the model is 7% but the effect at the critical hole
intersection is much less. Since the stress is dominated by the applied pressure at this location,
the increase in stress due to the increased thermal expansion is less than 1%. Allowing for the
effect of possible unaccounted factors (such as friction and heat loss through fluid) and assuming
a possible uncertainty in the coolant fluid of 10%, the amount of uncertainty due to thermal effects
in the results at the critical area should still be less than 2%.

The bolt load is initially applied as a pressure that assumes that the load is distributed perfectly
onto the top face of the cage block. Although this is ideal, it is not a critical assumption as the
actual case is most likely close to this ideal one. Uncertainty may be caused as the load may be
concentrated away from the central axis enough to create an additional bending moment in the
injector. Considering how the injector is constrained in the engine block (see Figure 12) it is
reasonable to conclude that the magnitude of this additional load is small. Any potential effect of
this on the maximum stress at the hole intersection is insignificant as that stress is primarily due
to the pressures at the hole intersection.

The nodal constraints at the tip are a good representation of the actual constraint due to the
injector nut. Since the nodes are constrained in both positive and negative axial directions, a
small local error might be introduced due to minor bending in that area but it does not have any
effect away from the constraints. If the injector should be slightly misaligned, it is likely that it will
contact the nut at additional areas other than at the constraints but this would be difficult to predict.

3. Errors Due to Analysis

Commercial codes such as ANSYS© and NISA© are extensively tested and the results of these programs are compared to analytical results for numerous standardised benchmark analyses with very good correlation. However, the accuracy of the results are dependant on the models created by the user.

The error relative to the mesh itself is considered to be minor. The automeshing techniques have many controls and checks to consider mesh distortion and shape testing. For tetrahedral solid elements, these checks include aspect ratio, maximum corner angle, and the Jacobian. The mesh is optimised as it is created in order to create satisfactory element shapes. If warnings occur as a result of element shape testing, additional input (such as local element size specification) is given to the model and it is remeshed until no further warnings occur. In order to verify this statement, a sensitivity analysis was performed on the fine-submodel to gauge the effect of the mesh configuration. Two additional fine-submodels were created with identical geometries but with different mesh configurations. One of the models was created with a relatively coarse mesh (4500 elements, 7000 nodes) compared to the original (10 000 elements, 15 000 nodes) and the other was created with a much more refined mesh (18 500 elements, 27 000 nodes). The maximum von Mises stresses in the most refined model agrees with the original fine-submodel within 8%. Even the coarsely mesh model differs from the original by less than 14%. Therefore it can be asserted with some confidence that the error due to discretization and mesh configuration in this model is less than 10%.

The submodelling techniques add to the complexity of the analysis but should not have a large effect on the accuracy of the results. The displacement submodelling method used to create the submodel does not introduce any significant error into the results as the mesh of the submodel and the corresponding section of the global model are similar. The NFBC algorithm may have introduced a small amount of uncertainty but previous tests of the method have indicated a very
Chapter 2: Analysis of Global Structure

small error when used in the types of applications for which it was designed. (See Section on
Submodelling methods) From this previous experience, the uncertainty introduced by the
submodelling techniques should safely be below 10% and is likely much less.

This analysis was run as a series of linear static analyses. This approach is valid assuming that
the dynamic effects are negligible. There are no moving components in this analysis of the global
model and the deformations are small. A calculation of the stress wave speed considering the
speed of sound in a metal, indicates that the time to propagate a stress wave through the injector
is much smaller than 1 ms. Therefore this is a reasonable approach which does not introduce any
additional uncertainty into the analysis.

2.8 Verification of Global Model

Due to the complexity of both the geometry of the injector and the loading scenario, it is not
possible to perform a significant check on the finite element analysis by analytic means.
Unfortunately, the small size, inaccessibility, and harsh operating environment of the injector also
prevents verification by experimental means. Considering the previous discussion of error it is
difficult to place a high degree of confidence in the absolute values of the predicted stress. There
should be enough confidence, however, to quantify the effect of changes in the design in a
comparative sense. Unfortunately in this case the changes in stress due to the modelled changes
in the design are too small to be effectively differentiated.

In order to check for modelling problems, simple loading cases such as tension and bending
cases were created. The stress profiles caused by such simple load cases may roughly be
predicted due to the geometry. Therefore problems related to bad elements, improper merging, or
improper input of material properties may be identified with these types of checks. The results of
the simple load cases of the global model revealed no gaps in the model, no discontinuous stress
or displacements, and generally agreed with the expected results.
2.9 Conclusions and Recommendations

The results of the global model indicate that the highest stresses are primarily due to the pressure load at a hole intersection in the vent block. This is consistent with the location of cracking that was observed in an analysis of an injector specimen. The fine-submodel allows a more refined mesh and the inclusion of small details in the model of the critical area. Alternative hole intersections are modelled at the same location for comparison purposes in order to try to improve the design by decreasing the stress levels. With the uncertainty in the loading conditions, these results can not be clearly differentiated from each other. Therefore, it may be concluded that the maximum stress is not greatly affected by these small changes in orientation and sizing. The best design choice should then be defined by other criteria such as manufacturability, ease of assembly, and maintenance.

The global model is used to indicate the important areas to be further analysed. The submodelling techniques allow the results from the global analysis to be translated to the detailed fine-submodel. This is an attempt to achieve the most accurate boundary conditions as possible in this analysis. Since the results of the alternative hole designs are used for comparison purposes, it may be possible to achieve adequate results much more efficiently by creating the fine-submodels directly and assuming the boundary conditions. The accuracy of these boundary conditions would certainly be significantly less than the loading determined through the submodelling technique but it would require only a fraction of the time to implement. For an analysis where the accuracy of the results is important, the combination of global model and submodels is necessary. However when a direct comparison is needed and time is more of a limitation, a more direct approach to this analysis with assumed boundary conditions may be sufficient.
3 Component Analyses

Needle and Tip Interaction

Introduction

In the previous global analysis, the diesel and CNG needle are excluded from the models, as theoretically their interaction with the other components is mostly limited to contact with the tip of the injector. In order to investigate the stresses in the needles as well as their interaction with each other and the tip, a separate analysis is undertaken in which models consisting of only the needles and tip are created. These models are created in ANSYS©. In order to model the interaction of these components, contact elements are specified and the analysis is carried out as a non-linear one.

One purpose of the stress analysis using this model is to predict the maximum stresses and deformations in these components. Although the vent block is not included in this analysis, the deformations calculated indicate the likelihood of contact and wear between it and the needles. Aside from looking at the general stresses and strains in these components, there are some specific features to be investigated related to the sealing contact between the components.

The CNG needle and tip interaction as well as the CNG and diesel needle interaction are similar to two cones with different slopes, one which fits inside the other. The areas where contact elements were added are shown in Figure 22.

![Figure 22 3-D Diesel Needle - CNG Needle – Tip Interaction](image-url)
Chapter 3: Component Analyses

When these components come into initial sealing contact, it is a line contact that circumscribes a circle. As a result of elastic deformation, this line contact becomes a surface contact. The designers of the injector currently have no way of predicting, other than looking at wear surfaces, how large this sealing surface gets as a result of the spring load on the needles. Therefore one of the aims of this analysis is to predict the area and location of these sealing surfaces.

The actual relative difference in slopes between the component contact surfaces is 0.5 degrees. The model is changed by altering this relative angle difference and then re-run to investigate the effect of the change on the resultant sealing surface.

Static Analysis Assumption

Unlike the global model, this model contains components that move during the operation of the injector, namely the needles. In order for this model to be run as a static analysis, the assumption is made that the inertial effects are insignificant. This was supported by the following rough calculation of inertial forces (assuming constant acceleration):

\[
Vol_{cng} = 2 \cdot \frac{\pi}{4} \cdot ((0.18 \cdot in)^2 - (0.096 \cdot in)^2) = 0.036 \cdot in^3
\]

\[
Vol_{diesel} = 2 \cdot \frac{\pi}{4} \cdot (0.076 \cdot in)^2 = 9.073 \cdot 10^{-3} \cdot in^3
\]

\[
Mass_{cng} = \frac{Vol_{cng} \cdot \frac{kg}{m^3}}{7700 \cdot \frac{lb}{m^3}} = 0.01 \cdot lb
\]

\[
Mass_{diesel} = \frac{Vol_{diesel} \cdot \frac{kg}{m^3}}{7700 \cdot \frac{lb}{m^3}} = 0.01 \cdot lb
\]

\[
d(\text{needle travel dist.}) = 0.03 \cdot in
\]

\[
t(\text{needle travel time}) = 0.001 \cdot s
\]

\[
vel_{ave} = \frac{d}{t} = 30 \cdot \frac{m}{s}
\]

\[
vel_{max} = 2 \cdot vel_{ave} = 60 \cdot \frac{m}{s}
\]

\[
accel_{ave} = \frac{vel_{max}}{t} = 6 \cdot 10^4 \cdot \frac{m}{s^2}
\]

\[
InertialForce_{cng} = Mass_{cng} \cdot accel_{ave} = 1.57 \cdot lbf
\]
Chapter 3: Component Analyses

\[ \text{Inertial Force}_{\text{diesel}} = \text{Mass}_{\text{diesel}} \cdot \text{accel}_{\text{ave}} = 0.392 \cdot \text{lbf} \] (3.11)

These calculated inertial forces are much less than the spring forces at contact (< 4%). A rough calculation is also used to check the possibility of resonance between the needles and spring and the engine frequency.

\[ k_{\text{cng}} \text{ (spring const.)} = 251 \cdot \frac{\text{lbf}}{\text{in}} \] (3.12)
\[ k_{\text{diesel}} \text{ (spring const.)} = 268 \cdot \frac{\text{lbf}}{\text{in}} \] (3.13)

\[ \text{freq}_{\text{eng}} = \sqrt{\frac{k_{\text{cng}}}{\text{Mass}_{\text{cng}}}} = 3.09 \cdot 10^3 \cdot \frac{1}{\text{s}} \] (3.14)

\[ \text{freq}_{\text{diesel}} = \sqrt{\frac{k_{\text{diesel}}}{\text{Mass}_{\text{diesel}}}} = 6.40 \cdot 10^3 \cdot \frac{1}{\text{s}} \] (3.15)

\[ \text{freq}_{\text{engine}} = 1200 \cdot \text{rpm} = 7.2 \cdot 10^4 \cdot \frac{1}{\text{s}} \] (3.16)

where \( k \) represents the stiffness of the springs and the resonant frequency is calculated as the square root of the stiffness over the mass. The predicted natural frequencies of the needles are at most approximately 10 times less than the average operating frequency of the engine (~1200 rpm) and therefore it is not likely that resonant effects are significant.

Boundary Conditions

The operation of the injector is discussed in section 1.3.2. The critical loading condition in this model is when the CNG and diesel needles are seated and the compression force on the needles is greatest. The load case when the CNG needle is not in contact with the tip is also modelled for comparison purposes. Constraints include the symmetry constraints at the lines and planes of symmetry. The nodes on the tip at the boundary interface where it would connect to the vent block are fixed. Originally, the needles were constrained only by the contact elements. This was a potentially unstable situation in ANSYS© as it often leads to an error in the solution. In order for this arrangement to have a solution, the surfaces must be in contact at the beginning of the
analysis and the parameters of the contact elements changed until a solution is achieved. Instead, an additional section is added to the top of both the needles where the spring loads are applied (See Figure 23). These sections are given different material properties, specifically, very low modulus of elasticity. The tops of these sections are then fully constrained. As a result of the low modulus of elasticity, these sections offer no significant resistance to the spring loads but still allow the needles to be constrained so that the solution does not cause an error.
Comparison of 2-D and 3-D Models of Tip and Needles

A 3-D cyclic symmetric model is created which requires that one sixth of the full solid model be created as this model is fully axisymmetric except for the presence of six injection holes in the tip and CNG needle (Figure 24).

Figure 25 2-D Axisymmetric Model
A 2-D axisymmetric model is also created which contains all of the same features as the 3-D model except for the injection holes (Figure 25). Although the analysis is based on small deformations, it is still non-linear as a result of the presence of the contact elements. The 3-D model is much larger than the 2-D model and as a non-linear analysis is required, the solution of the 3-D model takes much more time to process than the 2-D model. The 2-D model can more easily be modelled with a refined mesh at important areas and the analysis may be run multiple times which is helpful in checking and optimising the model. The calculation time needed for the solution of the 3-D model is prohibitively long relative to that of the 2-D model (more than 10 times greater). When the critical area is not significantly affected by the presence of the injection holes, the 2-D model is preferred as the local results are equivalent to those achieved by the 3-D model.

Contact Surface

Since the only difference between the 2-D and 3-D models is the local stress effects at the injection holes, it is preferable to use the 2-D model when looking at the sealing surface between the CNG needle and the tip. The mesh is refined along the contact surface until the maximum length of each of the contact elements is less than 0.002 in (0.05 mm) along the contact surface (See Figure 26). When the maximum loads are applied, the analysis predicts that only one of the contact element pairs are in contact at each interface. The amount of deformation at contact is minimal so that the contact surface predicted has a width of less than 0.002 in (0.05 mm). The maximum von Mises stress predicted at contact is 30.6 ksi (214 MPa). In order to test the model, the spring load on the components is increased by a factor of 10. The predicted contact surfaces are then increased to 0.013 in (0.33 mm) for the diesel/CNG needle interface and 0.012 in (0.30 mm) for the CNG needle/tip interface. The maximum von Mises stress is also increased to 140 ksi (980 MPa).
These results appear reasonable, however it is difficult to judge how accurately they represent reality. The results are affected by the stiffness of the model that in turn is a function of the material properties and geometry, element types, and mesh density. The best estimate of the injector designers is that the sealing surface is approximately 0.010 in (0.25 mm) wide. This value is estimated by examining the wear on the two surfaces with a microscope after injector operation.

This model predicts a much smaller width for the same load. Therefore, either this estimate is too large or the model is too stiff or it is a combination of these factors. Since there is not very much uncertainty in the modulus of elasticity of the material and the added sections at the top of the injector have no significant effect, the mesh must be chosen carefully. Quadratic elements were chosen which are more flexible than linear elements and the mesh is refined significantly at the contact surface. If the contact width is approximately 0.01 in (0.25 mm), then the contact elements size of < 0.002 in (0.05 mm) should be adequate for this analysis. Any possible
additional friction or misalignment that is not considered in this model would act to decrease the sealing surface but may increase the width of the wear band. The width of the wear pattern may be due to progressive wear. Very little material needs to be removed due to wear to extend the contact width since the relative angle is very small. The added width may also be related to the added deformation that occurs due to the inertial force of the needle as it is forced to decelerate at contact. The deformation in this analysis would represent the deformation once the needle has rebounded slightly and is static. Therefore it is suggested that the sealing surfaces in the injector could be smaller than are currently estimated.

Contact Angle Variation

As an additional step in this analysis, code was written within ANSYS®, which could automatically alter the geometry of the tip and CNG needle and remesh them so that various angles of contact may be examined (See Figure 27).

Figure 27 Contact Angle

However, as the amount of deformation at the contact surface is minimal, the angle does not have a significant effect on the width of the sealing surface. When a relative angle difference of
Chapter 3: Component Analyses

0.25° is modelled instead of the original 0.5°, the predicted contact surface is still limited to one contact element pair. When the contact surfaces are made parallel, a large contact surface of 0.04 in (1.0 mm) is predicted as expected, as no deformation is necessary to create the contact surface. The CNG spring load is increased by a factor of 10 in both the model representing the original relative angle difference of 0.5° as well as the model with the angle difference of 0.25° in order to achieve sufficient contact for comparison purposes. As expected, the contact surface area is greatest in the model with the smaller relative angle difference. The width of the contact surface predicted is 0.014 in (0.36 mm) versus 0.006 in (0.15 mm) for the original configuration. Therefore in this case, decreasing the relative angle by 50% results in an increase of the contact width of more than 100%.

Conclusions

The results of the contact surface analysis indicate a much smaller sealing surface than is currently assumed. When the appropriate spring loads and pressures are applied, the models indicate that the width of the sealing surface is less than 0.002 in (0.005 mm) and a higher load is necessary to cause significant elastic deformation and increase the sealing area. At the loads currently applied to the injector needles, changing the contact angle does not significantly affect the size of the sealing area. However, by increasing the spring load in the models, an inverse relationship is predicted between the change in contact angle and the width of the contact surface.

3.1 Transient Heat Analysis

3.2.1 Introduction

In the global model analysis, the brief investigation into transient heat analysis indicates that significant thermal effects in general are limited to the tip region of the injector and that transient effects are insignificant. These transient effects refer to the effect of the change of the temperature in the injector over an engine cycle and not the change in temperature in the injector during engine start-up before it reaches a pseudo steady-state. A report of cracking at the injection holes of the tip component caused a re-examination of the transient effects as this critical area is very close to the engine cylinder and therefore the source of the heat variations.
Chapter 3: Component Analyses

Since these cracks are formed in a location in which the transient effects may still be significant, then thermal fatigue may possibly be significant. The stresses in this region due to other loading conditions are relatively low in relation to other important areas of the injector, which reinforces the need to examine the transient effects more closely.

These boundary conditions are applied to a 2-D axisymmetric model of the tip that was created using 3 and 4 node axisymmetric heat transfer elements. The temperature of the gas in the engine cylinder at the tip of the injector is input as four different set temperatures at defined times in the engine cycle. A loop was created so that the analysis can be run for a predetermined number of engine cycles. Initial conditions are created by applying an appropriate gas temperature and allowing the injector to reach quasi-steady-state before beginning the engine cycle loops. This loading scenario is described in detail in the flowchart of Figure 31.

3.2.2 Boundary Conditions

The temperatures in the engine cylinder at the injector tip are estimated using pressure and temperature data from a similar engine test (Figure 28).

**Predicted 6V92 Engine Gas Temperatures At Tip**

![Graph showing predicted engine gas temperatures at tip](image)

**Figure 28 Engine Temperature Cycle**
Chapter 3: Component Analyses

These temperatures are significantly higher than previously estimated as they are more representative of the temperature of the gas at the injection tip than the average temperature in the cylinder or the temperature of the tip itself.

Instead of assuming temperatures at the boundaries of the tip component, convection coefficients are used to simulate the heat transfer from the engine cylinder to the injector and from the injector to the coolant fluid. The conduction of heat from the tip to the needles is neglected, as the contact between these is small and variable. The convection between the tip and the CNG gas is also ignored. Loss through radiation is negligible. A value of 0.000575 Btu/sec-°F-inch (43 W/m-K) is used for the conduction coefficient. The specific heat capacity is 0.1099 Btu/lb-°F (460 J/kg-K).

The convection coefficient at the part of the tip that is exposed to the engine cylinder is taken to be $1.7 \times 10^{-5}$ Btu/sec-°F-inch$^2$ (50 W/m$^2$K) for the surface to gas convection. The surface area from exposed tip to where the tip begins to expand radially is assumed to have a convection coefficient of $3.4 \times 10^{-5}$ Btu/sec-°F-inch$^2$ (1000 W/m$^2$K) for the convection from the surface to the coolant fluid. Convection from the remaining surface of the tip is ignored. There is a very large degree of uncertainty with respect to the choice of convection coefficients; however, the values chosen are conservative as the purpose of this analysis is to determine the significance of transient thermal effects. Conservative values for the coefficients were found by performing a sensitivity analysis.

![Figure 29 Transient Heat Analysis Boundary Condition Locations](image)

3.2.3 Results

The first interesting aspect of the results is the difference in temperature between the gas temperature at the tip and the tip itself. There is a significant thermal resistance across that interface. Although the temperature in the engine cylinder at the tip is input with a range of 415°F
to 1264°F (213°C to 684°C), the steady-state temperature of the tip itself is predicted in the range of 515°F (268°C) with only slight variation. At the tip, a plot of temperature vs. time shows clearly the variation of temperature at the tip over the engine cycle (Figure 28).

Even at the very tip where the variation will be greatest, the variation in temperature is still limited to approximately ± 1°F. Further along the tip, the variation in temperature quickly decreases until the temperature may be considered steady. The time required to reach the quasi steady-state condition (when the average temperature is constant) varies depending upon the initial condition specified. A simple one-dimensional transient heat flux analysis of a semi-infinite steel solid using a finite difference spreadsheet model was performed using the same temperature inputs and
coefficients. This analysis confirms the results of the finite element model as even the most conservative cases predict very little temperature variation after only a depth of a millimetre from the surface exposed to the engine temperature.

3.2.4 Conclusions

Without experimental data to back up the results or even to supply more accurate boundary conditions, there is a great deal of uncertainty when dealing with such a complex situation. The simplification of the boundary conditions most likely yields a reasonably good solution providing that the values chosen for the coefficients are appropriate. Unfortunately, without supporting experimental data, the results of this analysis must be handled with appropriate reservations. The boundary conditions chosen are designed to be conservative, so that even if the results are not accurate, they would predict a larger effect of the transient temperature variation than might actually occur. Therefore since this analysis predicts that the transient effects are small, it may be reasonably suggested that the actual effects of the transient temperature load are not significant. This conclusion is supported by the results of a separate finite difference analysis. It should be noted that, although the number of cycles is much smaller, the influence of start-up and shut-down transient effects may be important, especially with respect to crack initiation. This is not addressed in this analysis and may be a subject for future work.
Chapter 3: Component Analyses

Figure 31 Simple Flowchart of Loading Loop

LOADCASE1
Set Engine Temp to 725°F
Set Load Step to End at Time: 600 sec
Specify as STEP Load
Write Load Step: 1
N = 0

LOADCASE2
Set Engine Temp to 415°F
Set Load Step to End at Time: 600.005 + 0.03*N sec
Specify as STEP Load
Write Load Step = 2 + 4*N

LOADCASE3
Set Engine Temp to 794°F
Set Load Step to End at Time: 600.012 + 0.03*N sec
Specify as RAMP Load
Write Load Step = 3 + 4*N

LOADCASE4
Set Engine Temp to 1264°F
Set Load Step to End at Time: 600.015 + 0.03*N sec
Specify as RAMP Load
Write Load Step = 4 + 4*N

LOADCASE5
Set Engine Temp to 415°F
Set Load Step to End at Time: 600.03 + 0.03*N sec
Specify as RAMP Load
Write Load Step = 5 + 4*N

N < 100?  
Yes  
N = N + 1  
No  
Solve for Load Steps
3.2 Stress Concentration at Holes in Tip

3.2.1 Introduction

A crack evaluation of six injector tips that were subjected to a 1000-hour durability test revealed the presence of cracks at the inlet of the injection holes [16]. A 3-D cyclic symmetric model is created for an analysis of the critical section at the injection holes. The purpose of this analysis is to try to quantify the effect of design changes in the end of the tip component. Therefore different design modifications are modelled separately and in combination in order to compare the resultant maximum stress seen in the critical area.

The durability test consisted of the injectors running within an engine block at an engine speed of approximately 1800 rpm for 667 hours as part of an accelerated wear duty cycle equivalent to a conventional duty cycle of 1000 hours at 1200 rpm. The components experienced more than 70 million cycles in the course of the full test. The conventional duty cycle was based on the New York City cycle and the Urban Dynamometer Driving Schedule for Heavy Duty vehicles (HUDDS). The tests were conducted at the Idaho National Engineering and Environmental Laboratory (INEEL)[17]. One of the injector tips exhibited significant cracking as cracks were found initiating at the inlet of the holes and extending along the hole and through the needle contact seat while in the other five injector tips, only very small cracks were found.
Figure 32 shows a fractograph of the tip with the most significant cracking. The sample was immersed in liquid nitrogen and then broken open along the crack front. Note the magnification as even this crack is in the micro-crack range (~0.2mm). Note the secondary particle at the point...
where the crack split along the boundaries. It is likely that the crack grew as a quarter elliptical crack until the secondary particle caused it to arrest in the radial direction. The crack then continued to grow around the secondary particle along the injection hole and needle seat. The presence of this secondary particle and the others visible in the figure may be a factor in why this sample had propagated the cracks much more than the others. The other half of this cracked tip was mounted in epoxy and ground to expose the cracks perpendicular to their propagation paths. All three injection holes exposed in this process exhibit similar length cracks. This indicates that the cause of the cracks is not a random material flaw. A manufacturing flaw would only be possible if it is the type to be repeated in the manufacture of each injection hole.

The crack evaluation also noted striations and cyclic propagation features on the crack surfaces at high magnifications. This evidence and the high number of cycles to which the tips were subjected clearly indicate a high cycle, low load fatigue failure. Such a fatigue failure may be caused by cyclic thermal or pressure stresses; however, the previous transient heat transfer analysis indicates that effects of cyclic thermal stresses is small. Most of the small cracks are limited to the white nitride surface layer on the components (See Figure 33). This nitride layer is a probable source of initiation of the cracks. Although the nitride layer induces favourable compressive stresses at the surface, if it cracks, the residual tensile stresses in the adjacent material may help to grow the crack. The initiation point of the cracks is susceptible because it is the location of the highest local stress and an excess of white nitride. The high surface area at the intersection of the hole with the inner surface of the injector causes the nitride layer to penetrate deeper in that location and results in a deeper layer of the brittle white nitride.
3.2.2 Fatigue Crack and Failure Analysis

At first glance, this seems to be a good candidate for a fatigue analysis with a fracture mechanics approach. However, the cracks in this analysis were too small to model with normal finite elements. A micro-crack approach would have to be undertaken which would be difficult to integrate with the finite element model. Factors such as grain size and boundaries would have to be considered and these are currently ignored in this finite element model. Furthermore, accurate boundary conditions would be necessary if the aim of the analysis was to predict the life of the injector tips. This would require the use of global model again as well as contact elements to model the interface with the needles as the contact surface is too close to this critical area to ignore. The result would be a very complex model and analysis that would be beyond the scope of this project.

Even without the fracture analysis component, trying to accurately predict the resultant stress values in the tip would be questionable. At least with the fatigue analysis, the final crack size
could be compared to that found in the specimen although the chance of them agreeing would be small. As with the previous analyses, there were no experimental values for confirmation of stress values.

It is therefore a more efficient use of time to compare the relative stress results of design modifications with boundary conditions that are close to those in the actual injector. The ratio of maximum stress values determined in this analysis should be very similar to those that might be calculated if the actual stresses could be measured.

3.2.3 Model Description and Boundary Conditions

Although the hole is critical to this analysis, 2-D models of the loading scenarios are created first. The corresponding 3-D models are then created which include the injection holes and the maximum stresses are found to be much higher. In order to verify that this rise in stress is due to the presence of the hole and not an error in the model, another 3-D model of the tip only is created without the hole. The results of this model agrees with the 2-D model and so verifies that the rise in stress is due to the stress riser effect of the hole. For the models featuring the design modifications, the simpler loading scenario dealing only with the tip is used.

Since the critical area is limited to the end of the tip of the injector, only the bottom sections of the tip and needles are included in this model. The two main loading conditions considered are one with both needles seated and the other without needles (needles not seated). The nodes at the uppermost section of the tip model are fully constrained.
3.2.4 Discussion of Design Options

A number of design alternatives are modelled in order to assess their effectiveness in reducing the stress concentration at the inlet of the injection hole. These changes are all limited to the local area of the tip. In this analysis, the modifications are only compared in terms of stress effects; however other important considerations such as the effect on the injected jet would also have to be considered by the injector designers.

Figure 34 shows the pressures applied on the faces of the model and their locations.
(i) Extend Tip Radius

The first modification consists of extending the outer radius at the very tip of the tip component. The radius is extended by 0.015" and then further extended so that the radius corresponds to the larger radius elsewhere on the tip (See Figure 35).

(ii) Round Edge of Holes

Another design modification is to round the edges of the inlet hole. In practice, this would be achieved by forcing a diamond paste through the hole with a high pressure that would wear away the sharp edges and produce the round. This process is known as Abrasive Flow Machining (AFM). Models are created with 0.0025 in (0.06 mm) and 0.005 in (0.13 mm) radius rounds (See Figure 36).
(iii) Change Size of Holes

Changing the size of the inlet holes themselves from 0.02 in (0.51 mm) diameter to 0.018 in (0.46 mm) diameter has a double effect on the stresses in the tip. If the hole diameter is decreased, then the number of holes must be increased to achieve the same flow rate. Therefore the stress is affected by the smaller hole diameter as well as the fact that there are now eight holes rather than the original six. Although in practice these features would always be seen together, they are also modelled separately so that their individual and relative effects may also be seen (See Figure 37).

![Figure 37 Models with 8 Smaller Holes](image)

(iv) Adding Material Above Holes

A slightly more complex design change causes the seat of the CNG needle to be moved slightly up on the injector tip in order to add material above the hole (See Figure 38).
3.2.5 Results

Before investigating the results of the different design alternatives, an approximation of the alternating stress due to prescribed load cases is considered. Comparing the two load cases discussed previously for both the 2-D and 3-D models, the alternating stress varies from 3 to 5 ksi (21 to 34 MPa). This is a small alternating stress but the cracks detected are in the micro-crack range and may be propagated with a small load. Although this is a very rough approximation of the actual alternating stress, it is important to investigate this value as it is the alternating stress which would cause the cracks to propagate. The alternating stress is mostly a function of the loading, which is not a design parameter that may be changed. The design alternatives considered in this analysis are designed to reduce the maximum stress in the tip but it is likely that they would not significantly change the alternating stress. However, if the maximum and mean stresses experienced in the tip can be reduced, this would have the potential to increase the fatigue life of the component.
(i) Extend Tip Radius

The added material adds stiffness to the tip of the injector and reduces the stresses seen at the hole inlet. More material results in a greater decrease in the maximum stress. See Table 4 for numerical comparisons of effects.

(ii) Round Edge of Holes

The round is supposed to reduce the stress by eliminating some of the edge effects at the hole inlet and reducing the stress concentration. However, all three models that included the rounded inlet hole indicate that instead of reducing the maximum stress, the stress is actually slightly increased. The orientation of the hole with respect to the stresses in the tip is such that the expected benefits do not occur. Both radii of round that are tested result in a higher maximum stress although the larger radius round has less negative effect. Another consideration with this modification is that the elimination of the sharp edges would also reduce the formation of excess white nitride at that location. Since this was likely a significant factor in crack initiation, the benefits of the round with respect to crack initiation may be greater than the negative effect on the stress concentration.

(iii) Change Size of Holes

Decreasing the hole diameter has a slight beneficial effect as there is more material around the smaller hole and so the added stiffness again acts to reduce the maximum stress. The larger number of holes actually has a more significant effect. Since there are now 8 holes, the cyclic symmetric model consists of 45 degrees or 1/8th of the full solid model rather than 1/6th. The model is now smaller and therefore the symmetry constraints are closer to the injection hole. This also has a stiffening effect and causes a decrease in the maximum stress.

(iv) Add Material Above Holes

Raising the location of the needle seat on the tip also has a beneficial effect as there is now more material above the inlet hole and the maximum stress is decreased.


(\textit{v}) \textbf{Combinations}

All of the design modifications have some benefits as expected. However, in terms of stress only, the rounding of the inlet of the hole may actually increase the maximum stress at the hole. The maximum decrease in stress occurs when all of the potentially helpful design modifications are instituted at the same time. The benefits of each modification are independent and so the greatest overall benefit represents the sum of the effects (See Figure 39). Table 4 shows the different combinations of design modifications and features that are modelled and the relative change in the maximum stress from the original design.
### Table 4 Design Modifications and Features and the Effect on the Maximum Stress

<table>
<thead>
<tr>
<th>Model Name</th>
<th>0.0025&quot; Round</th>
<th>0.005&quot; Round</th>
<th>Extend by 0.015&quot;</th>
<th>Extend to Intersection</th>
<th>6 Inject. Holes</th>
<th>8 Inject. Holes</th>
<th>&lt; 0.02&quot; Dia. Hole</th>
<th>0.016&quot; Dia. Hole</th>
<th>Move CNG</th>
<th>Seat Up</th>
<th>Max Stress</th>
<th>% Change in Max Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>holetip3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>15909</td>
<td>Base</td>
</tr>
<tr>
<td>holetip4</td>
<td>✔</td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>16717</td>
<td>+5.1%</td>
</tr>
<tr>
<td>holetip14</td>
<td></td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>16285</td>
<td>+2.4%</td>
</tr>
<tr>
<td>holetip7</td>
<td></td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>15206</td>
<td>-4.4%</td>
</tr>
<tr>
<td>holetip13</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>14765</td>
<td>-7.2%</td>
</tr>
<tr>
<td>holetip8</td>
<td></td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>14584</td>
<td>-8.3%</td>
</tr>
<tr>
<td>holetip16</td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>14498</td>
<td>-8.9%</td>
</tr>
<tr>
<td>holetip12</td>
<td></td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>13453</td>
<td>-15.4%</td>
</tr>
<tr>
<td>holetip15</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>13101</td>
<td>-17.7%</td>
</tr>
<tr>
<td>holetip17</td>
<td>✔</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>12349</td>
<td>-22.4%</td>
</tr>
</tbody>
</table>

3.2.6 Conclusions

An approximate value for the alternating stress is determined by varying the loading conditions. The alternating stress is primarily due to the loading conditions and cannot be significantly changed by geometry modifications. Although not as critical, the maximum stress is a feature that may be controlled by design geometry and still have an effect on the fatigue life of the component. If the maximum stress were the only design consideration, the best alternative would be to implement all of the design modifications except for the rounding of the hole inlet. However, the elimination of the sharp corner at the inlet may be advantageous due to the suppression of crack initiation at that location. Some of the design modifications may have a significant effect on the injection performance. For example, the extension of the outer radius of the tip would cause the inlet hole to become longer. This would change the length-diameter ratio and potentially change the characteristics of the gas as it exits the injection holes. This of course would be an
important consideration in the decision on which design modifications to implement. Once the ramification of each of the design modifications is considered, Table 4 would be a valuable resource in assessing each of them individually.

### 3.3 Effect of Pressurised Spring Bore on New Design

#### 3.3.1 Introduction

In the newest version of the injector design, the spring bore inside the cage block and check block components of the injector is pressurised. The consequent deformation of the plunger bore adjacent to the spring bore may impede the travel of the plunger. A new model of the cage and check blocks is created in order to predict the deformations that occur under these conditions.

#### 3.3.2 Model Description and Boundary Conditions

Although the check and cage blocks were modelled as part of the original global model, a completely new model is created which reflected the most recent design. Care is taken to produce a mesh that requires few enough elements and nodes so that the model does not exceed the limits imposed by the ANSYS© license agreement. The cage and check blocks are meshed together as a single model so that the interface between the two components may be neglected. The current design of these components is such that they could almost be created by extruding a 2-D mesh along the longitudinal axis. However, two of the gas holes are not parallel to the longitudinal axis and so this method cannot be used. Fortunately, a new feature of the latest version of ANSYS© allows the creation of a very uniform mesh by allowing the extrusion of a 2-D mesh along an existing volume. This feature enables the creation of a very uniform mesh using quadratic brick elements and allows the model to be created within the imposed nodal limits (See Figure 40).
The nodes at the top and bottom faces of this model are completely constrained. This reasonably represents the actual constraints on these components. The top of the cage block caps off the spring bore so that it does not directly experience the spring bore pressure and the friction of the bolt load interface constrains it securely. The bottom face of the check block is constrained by the friction of the interface of the more rigid vent block component. As the check and cage blocks are meshed together, it is assumed that the friction at their interface would keep the components from moving relative to each other. The gas and diesel holes in the model are loaded with the appropriate pressures as in previous models, however now the spring bore is loaded with a pressure of 3000 psi (21 MPa).
3.3.3 Results

The deformation of the model as shown in Figure 41 is greatly exaggerated but it does follow the expected pattern, as does the representation in Figure 42. The maximum deformation occurs at the portion of the plunger bore where the wall between the spring bore and the plunger bore is thinnest. A graph of the deformation at the inner and outer walls of the plunger bore is shown in Figure 44. The difference between the two deformations is the deformed 'diameter' and removes the relative displacement at both locations due to radial expansion of the components. In terms of clearance of the plunger inside the plunger bore, the maximum difference between the min and max deflection at each location along the bore gives the best indication if there is cause for concern. The maximum deformation occurring at the plunger bore was 0.0039 mm ($1.5 \times 10^{-4}$ in).
Chapter 3: Component Analyses

Max deformation at thin wall of plunger

Figure 42 Displacement Contours in Spring Bore

Max stress at thin wall of large plunger hole

Figure 43 Von Mises Stress in Spring Bore
Chapter 3: Component Analyses

Graph of Deflection at Inner Wall of Plunger Holes.

Note: Almost a magnitude less deflection and now more deflection in check than cage compared to inner wall.

Deformation at Outer Wall of Plunger Holes:

Figure 44 Displacement Along Plunger Bore
Due to the relative deformation of the outer wall of the plunger bore, the maximum predicted decrease in the size of the plunger bore is $3.2 \times 10^{-3}$ mm ($1.3 \times 10^{-4}$ in). The acceptable change in clearance provided by the injector designers is one micron. Therefore under the conditions of this model, the deformation is more than three times the acceptable amount.

### 3.3.4 Analytical Check

Although there are no experimental data with which to check this model, the geometry may be reasonably approximated by a more simple geometry and checked with an analytical analysis. If the plunger bore and gas and diesel holes are ignored, the model resembles a simple thick walled cylinder. In order to deal with the constrained ends however, a thin walled analysis is used based on infinite beam theory. A mean radius value ($r_m$) was used for the analytical equation for radial expansion.

\[ y(x) = \frac{P \cdot r_m^2}{E \cdot t} \cdot \left(1 - e^{-\beta x} \cdot (\cos(\beta x) + \sin(\beta x))\right) \]  

where $E$ is the modulus of elasticity, $\nu$ is Poisson's ratio, $t$ is the thickness, $P$ is the internal pressure, $x$ is the axial distance from the constrained end, and $y$ is the radial expansion. The radial expansion predicted by this equation is compared to the radial expansion seen at the inner and outer radius of the model. A location on the circumference of the model is chosen away from the presence of any of the gas or diesel holes. A plot of this comparison is shown in Figure 45 which indicates a close correlation between the two methods, especially considering the simplifications involved in the analytical model.
Chapter 3: Component Analyses

3.3.5 Conclusions

Given the limitations of the analytical equation, the analytical check of the deformation of the model supports the numerical results well. With the present configuration of the injector model, the deformation of the walls of the cage and check block components at the plunger bore, caused by pressurising the spring bore, is predicted as more than three times the given limit. There is insufficient clearance for the plungers to operate without restrictive contact with the cage and check blocks. Therefore if the spring bore is pressurised, the walls at the plunger bore would have to be thicker or subjected to a lower pressure.
4 Summary and Future Work

HPD fuel injector prototypes have developed significantly in the last few years but have not yet achieved an optimal degree of reliability. Fatigue cracks were identified during the failure analysis of components subjected to engine testing for durability. Numerical finite element models are created to gain insight and improve the design, since prototyping is expensive and experimental testing of the injector is limited.

The sources of error in the different analyses may be grouped into three specific types: error due to modelling, error related to the boundary conditions, and error specific to the finite element analysis. The amount of uncertainty in the global analysis due to these sources of error is estimated in the range of 60% to 75%, most of which is due to the uncertainty in the boundary conditions. The use of the global model and submodelling techniques of the global analysis results in boundary conditions at the critical area that are more representative of the real situation than if the boundary conditions are assumed. However, assumed boundary conditions for a model of a critical area are usually sufficient if the purpose is primarily to compare different design alternatives.

A global analysis of the entire injector without the needles is used to predict the most important areas of high stress in the design. Through the use of submodelling techniques, a critical area involving an intersection of pressurised holes is examined in more detail and alternative design options of the same area are also modelled and compared to the original design configuration. Changes to the hole intersection do not significantly affect the maximum stress predicted by the submodels. Therefore, any localised modifications to the hole intersection would be insufficient to achieve a significant decrease in the maximum stress in the critical section. Failures at this critical location may be more sensitive to the quality of the manufacturing techniques used to produce the hole intersection than any modifications to the local geometry and therefore the local design should be chosen to facilitate manufacture. Larger changes to the loading or geometry of the injector that are not limited to the critical area are necessary to further improve the stress state in this region.
In a separate design analysis, the interaction between the needles and the tip is studied with use of contact elements. The size of the contact surface formed between the components is predicted as less than 0.002 in. (0.05 mm) which is significantly less than the anticipated value of 0.01 in. (0.25 mm) used by the injector designers and determined by microscopic observation of the wear surface. Changing the relative angle between these contact surfaces does not have any appreciable effect on the size of the contact surfaces under the proper specified loading conditions as the model still predicts minimal contact. However, when the spring load is increased by a factor of 10, decreasing the relative angle between the contact surfaces by half results in the doubling in size of the contact surface.

The results of a transient heat transfer analysis verify the assumption that the thermal variation over the engine cycle is not a significant factor in the cause of the fatigue failures. In a conservative model, the maximum temperature deviation in the tip is limited to approximately one degree at the engine cylinder. The size of this temperature range quickly decreases away from the engine cylinder, supporting the steady state temperature assumption made in the stress analyses of the injector. The results of this analysis are supported by a simple one-dimensional finite difference heat transfer analysis that also predicts these minimal temperature variations.

As the very tip of the injector is exposed in the engine cylinder, the possibility of a catastrophic failure exists in this area. The importance of analysis of this section is further emphasised by the discovery of cracks at the injection holes of a tip. Therefore several theoretical design improvements to this section geometry are modelled and quantitatively compared. Most of the design changes result in a decrease in the maximum stress in the model as was anticipated. Rounding the inlet of the injection holes that formed the major stress concentration in the area, however, does not result in the expected decrease in the stress range in the model. Aside from verifying that the individual design changes improve the stress condition in the area, this analysis allows for a quantitative comparison of the different options. Since the design changes also impact other aspects of the design such as manufacturability and function, the comparison table generated in this analysis is valuable in the decision of which design changes to implement.
Another analysis is performed to check the ramifications of pressurising the spring bore of the injector. This analysis reveals that the degree of deformation in the plunger bores adjacent to the spring bore results in restriction of movement of the plungers. Therefore the thickness of the walls of the spring bore must be increased to reduce the effects of the pressurisation.

Future work associated with the design analysis of the HPD fuel injector using finite element models would be focused on newer designs that contain significant changes from the versions analysed in this project. These newer injector prototypes have been designed for different diesel engines and applications such as heavy-duty trucks and power generators. The analyses would be similar in nature to those performed in this project but customised to suit the latest design requirements.

Another consideration for future work is in the area of using the commercial finite element models to generate fluid dynamic models of the gas and diesel fuel in the injector. Such models have already been generated using specific computational fluid dynamics programs but a finite element code may have some advantages over these. It is also a method of verifying the results of these other models. Therefore an investigation into the applicability of finite element programs to model the fluid effects in the injectors would be valuable.

A fatigue and fracture analysis of the injector prototypes is possible although it would be highly theoretical which would be inconsistent with the practical approach taken in this project. The cracks investigated in the injector tip are in the micro-crack range and therefore features such as the grain size and configuration in the metal of the tip become very important. It would be very difficult to model such small scale features and still relate them to the existing finite element models. Such an analysis would require many assumptions and would be based on fairly abstract micro-crack fracture mechanics theories. Therefore it is not advisable to perform a fatigue and fracture analysis at this time.

The finite element models created in this project resulted a better understanding of the stress state in the injector components and the associated uncertainty. Insight was gained into the operating conditions of the injector and the analyses were valuable in allowing educated design
decisions based on the analyses to produce an improved design. Future work would be concentrated on newer prototype designs but would be facilitated by the lessons learned in the course of this project.
References

[1] U.S. Patent References:

  Patent 1,321,110, "Intensifier-Injector For Gaseous Fuel For Positive Displacement Engine," Aug. 10, 1993
  Patent 2,810,455, "Intensifier-Injector For Gaseous Fuel For Positive Displacement Engine," July 31, 1998


References


[15] Input from David Mumford of Westport Innovations and email message, June 20, 1999


Appendix A: The Finite Element Method

Introduction

The finite element method is a numerical technique used to analyse a wide range of engineering problems. Usually the problem is too complex to be solved using classical analytical methods. Field variables such as stress or displacement in a structure have an infinite number of values, as they are a function of the location in the structure. In a finite element model, this number is reduced to a finite number by expressing the field variable with an approximation function within each element. These approximation or interpolation functions are defined in terms of the values of the variables at specific points in the element called nodes. Nodes of adjacent elements are shared to ensure compatibility between elements. The following is a summary of the steps in a typical linear finite element stress analysis:

1. Preprocessing Stage:

   In this step, the geometry of the structure is prescribed into the program and the structure is then discretized into small elements. This process is called discretization and mesh generation or modelling. Model checks are also performed in this step. This stage would normally consume more than 50% of the analyst's time.

2. Solution Stage:

   Various actions are performed in this stage. This includes the generation of elementary characteristic equations, the assembly of the element matrices, and specification of material model and analysis type. Loading and boundary conditions are then specified before the solution of the global set of equations takes place.

3. Postprocessing Stage:

   In this stage relevant quantities (e.g. strain, stress) are calculated from the solution field (e.g. displacement). Graphical presentation and display of results are also performed in this stage. [4] [5].
Appendix A: The Finite Element Method

Commercial Finite Element Packages & CAD Software

There are a number of commercial finite element analysis (FEA) packages available such as ANSYS© [6], NISA© [7], NASTRAN©, and ABACUS©. They usually contain a large library of element types and can accommodate many different kinds of analysis. It is possible to create a custom finite element analysis to solve a specific problem but usually this is only done when a specialised formulation is necessary. Even then it is still often easier to integrate special features into existing codes if possible.

If the geometry of a component being analysed has already been produced in a commercial computer aided drafting (CAD) package, it is of interest to transfer this data into the FEA package rather than recreate the geometry in the pre-processor of the FEA package. The advantages of such a transfer were more apparent in past years when the pre-processors of the FEA software were much more difficult to use than they are today. In the past, such transfers of information were included as options in these packages but their usefulness was limited. The translation process would produce small gaps and discontinuities in the model that were often very hard to detect and eliminate. The representation of the geometry was usually much more complicated than if the model was created directly in the pre-processor.

The development of solid modelling techniques in both FEA and CAD software has improved this situation. With the use of standardised formats such as IGES (Initial Graphics Exchange Specification), fairly complicated solid models can be created in such programs as Solid Works© and ProENGINEER© and imported into FEA packages such as ANSYS©. However, the amount of work required to fix the model of the imported solid geometry is still usually considerable. Unlike a model created directly, an imported model may still contain deficiencies (e.g., gaps, cracks, and overlaps) that are difficult to detect and may lead to difficulties in the creation of the mesh or the obtaining of the solution.

Some CAD software developers have ventured into the domain of the FEA software developers to introduce a more seamless transfer of data from the CAD package to a finite element analysis.
Appendix A: The Finite Element Method

Jacobian Ratio

- 1 (Optimal)
- 30
- 1000 (Poor)

Maximum Corner Angle

- 60° (Optimal)
- 165°
- 90°
- 140°
- 180° (Poor)

Aspect Ratio

- 1 (Optimal)
- 20 (Poor)

Parallel Deviation

- 0° (Optimal)
- 70°
- 100°
- 150°
- 170° (Poor)

Figure A.1 Element Checking Features
Most of the finite element pre-processors and the CAD packages that provide modelling capabilities perform rigorous model checking. Generally accuracy is a function of mesh size and density but if the elements are poorly shaped, the solution can be significantly affected. One benefit of auto-meshing techniques is the ability to check the shape of the elements as they are generated to produce a somewhat optimised mesh. Even so, the mesh should be verified, as a finer mesh may be necessary to create more uniformly shaped elements. Common checks on element shape include aspect ratio, parallel deviation, max corner angle, and Jacobian ratio.

The shapes of solid elements are verified by checking each of the 2-D faces of the element as well as specific cross-sections through the element. Shapes associated with shape testing are depicted in Figure . Aspect ratio refers to the ratio of the longest edge of the 2-D section to the shortest edge. Parallel deviation refers to opposite edges of quadrilateral faces. It has been demonstrated that degradation of stress convergence will occur in linear displacement quadrilaterals as opposite edges become less parallel to each other. Maximum corner angle on the sections is checked as the element performance degrades when there are large angles that approach 180°. For isoparametric elements, a high Jacobian ratio indicates that the mapping between the element geometry and the ideal geometry is becoming computationally unreliable [6].
Appendix B: Other Mesh Refinement Techniques

Introduction

Besides submodelling, there are other techniques that can be used to achieve a localised fine mesh. Aside from using transition elements, substructuring is the main alternative to submodelling in finite element analysis. Recent advancements in this area are also discussed briefly.

Substructuring

In substructuring, the critical area is modelled separately as a substructure known as a superelement. The process of condensation is used to combine the internal elements of a substructure into a single superelement. The properties of the internal nodes are analysed and then represented by master nodes on the boundary of the superelement at the interface between the superelement and the global model. When the global model is run, the master nodes represent the substructure and the internal details do not have to be reanalysed. The bulk of the analysis of the substructure is already completed in the creation of the superelement. Therefore the analysis of the substructure is sort of decoupled from the analysis of the global model. Although this procedure may be fairly complex to implement, there can be significant advantages.

If the substructure is repeated within the global model, this method becomes very efficient as the section only has to be analysed once and the results copied to each location in the global model. Even if the superelements are not used more than once in a global model, this technique may be used to create a number of different substructures so that a model that is too large can be broken down into more manageable components. In a non-linear analysis, a linear portion of the model may be analysed as a linear substructure and thus require less computational resources.

Substructuring does not aid much in the analysis of different design alternatives, as the entire model would have to be rerun to determine the new results including the new superelement and the global model. In terms of accuracy, this technique is considered an 'exact' technique for static
Appendix B: Other Mesh Refinement Techniques

analysis, as the process of condensation does not require any additional assumptions. It is simply a reorganisation of the equations and therefore it does not affect the accuracy of the analysis.

The following is a simple illustration of the substructuring procedure in which \( k \) represents the stiffness matrix, \( d \) represents the degrees of freedom, and \( r \) represents the load vector:

Linear Static Equations (for superelement portion of model)

\[
[k]{d} = {r}
\]  \hspace{1cm} (a)

Upon partitioning the global stiffness matrix to reflect internal DOF \( d_i \) and external DOF \( d_b \) we get:

\[
\begin{bmatrix}
  k_{bb} & k_{bi} \\
  k_{ib} & k_{ii}
\end{bmatrix}
\begin{bmatrix}
  d_b \\
  d_i
\end{bmatrix}
= 
\begin{bmatrix}
  r_b \\
  r_i
\end{bmatrix}
\]  \hspace{1cm} (b)

or

\[ [k_{bb}]d_b + [k_{bi}]d_i = r_b \] \hspace{1cm} (c.i)

\[ [k_{ib}]d_b + [k_{ii}]d_i = r_i \] \hspace{1cm} (c.ii)

Rearrange (c.ii) and solve for \( d_i \)

\[ \{d_i\} = -[k_{ii}]^{-1}(r_i - [k_{ib}]d_b) \] \hspace{1cm} (d)

Substitute (d) into (c.i):

\[ ([k_{bb}] - [k_{bi}][k_{ii}]^{-1}[k_{ib}])d_b = r_b - [k_{bi}][k_{ii}]^{-1}(r_i) \] \hspace{1cm} (e)

or

\[ [k']d_b = r' \] \hspace{1cm} (f)

where \([k']\) and \([r']\) represent condensed versions of \([k]\) and \([r]\).

Once these are computed, they are used in place of \([k]\) and \([r]\) and this portion of the model is only a function of the boundary d.o.f.
Recent Advancements in Mesh Refinement Techniques

A literature search was undertaken to investigate recent advancements in the field of mesh refinement techniques besides the NFBC method. Most of the papers found from the last two decades dealt with the development of only two new techniques. It could be argued that these methods are not entirely new but merely new variations of existing methods such as substructuring. These two methods are namely the Zooming method [10-12] and the Two-Level Fractal Finite Element method [13-14]. The Two-Level Fractal Finite Element method is related to crack analysis and since this is unrelated to this project, only the Zooming method will be discussed further.

The Zooming method by Hirai et al. utilises static condensation to reduce the entire global model and then refines the submodel. New nodes are added to the submodel and then a sub-region of the submodel is then used to create a further submodel. This zooming process is continued until the desired degree of refinement is achieved. This is an exact method but the accuracy of the results of the model is still affected by the choice of mesh and zooming area. This method can result in considerable computational savings but it requires a specialised finite element program. Therefore it is not appropriate for integration with commercial FEA codes.
Appendix C: Procedure to Implement NFBC Submodelling Technique

Although the NFBC algorithm was created as an executable piece of code, the implementation of this code is fairly complex and time consuming. The following procedure was created to better illustrate the use of the algorithm. Additional code was created to automate most of the steps as described below in order to significantly decrease the time to implement the technique as well as decrease the potential for implementation error. Figures C.1 and C.2 show the user interface for some of this code and indicate how they would be utilised.

**Step 1)**

- Open Global (Full) Model
- Delete all except for sub-model section
- Save as *.dbs file (i.e. section.dbs)

**Step 2)**

- Place cutplanes as layers
- Colour layers to ensure that nodes exist in one layer only
- Delete all except for nodes on cutplanes
- Save as *.nis file (i.e. temp.nis)
- Print for future reference.

**Step 3) (or use nisa-to-dis.exe to automatically create temp.dis)**

- Edit output solution file from global model (global.out):
  - Delete all except for the displacement solution.
  - Delete all nodes except for cutplane nodes by referencing printout (temp.nis).
- Save as new file *.dis (i.e. temp.dis)
Appendix C: *Procedure to Implement NFBC Submodelling Technique*

**NISA TO DISP Edits Displacement Solution to include only nodes on cutting planes**

This code edits the global solution file to create a new file which contains only the displacements for the nodes on the cutplanes. It takes as its input the name of the files 'temp.nis' and 'global.out' as described in the instructions on the NFBC sub-modeling procedure (MSWord doc). This code automates step 3.

**Input Files**
- Edited NISA file containing only nodes on cutplanes
  - [temp.nis]
- Solution Output File from Global Model
  - [global.out]

**Output Files**
- The output file generated by this code is automatically named 'temp.dis'.
- This can then be input into 'dis-to-ses.exe' to produce a session file.

Figure C.1 User Interface for Nisa-to-Dis.exe code.

**Step 4) (or use dis-to-ses.exe to automatically create disp.ses)**

- Make temp.dis compatible with NISA© session file format:
  
  e.g. dis.add,node#,1,DOF1/DOF2/DOF3/.....DOF6

  where previously the format of temp.dis was:

  node#  DOF1  DOF2  DOF3  ....  DOF6

- Save as new file (disp.ses).

  Note: Some displacements may be ignored if the corresponding lines in the session file were too long. Split long lines into 2 by adding "& then enter".

**Step 5)**

- Open section database file (section.dbs)
- Read session file created in Step4 (disp.ses)
- Save as new NISA© file (section.nis)
- Run section.nis to verify that the results of this model are identical to the full model.
Appendix C: Procedure to Implement NFBC Submodelling Technique

Steps 6) to 10) are repeated for each cutting plane.

The Input File for NFBC has the following format:

<table>
<thead>
<tr>
<th>Description</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Step 6: #of nodes on section model:</td>
<td>10</td>
</tr>
<tr>
<td>node# xcoord ycoord zcoord:</td>
<td>100 x y z</td>
</tr>
<tr>
<td>200 &quot; &quot; &quot; &quot;</td>
<td></td>
</tr>
<tr>
<td>: : : :</td>
<td></td>
</tr>
<tr>
<td>1000 &quot; &quot; &quot; &quot;</td>
<td></td>
</tr>
<tr>
<td>Step 7: node# fx fy fz mx my mz:</td>
<td>100 fx fy fz mx my mz</td>
</tr>
<tr>
<td>200 &quot; &quot; &quot; &quot; &quot;</td>
<td></td>
</tr>
<tr>
<td>: : : : :</td>
<td></td>
</tr>
<tr>
<td>1000 &quot; &quot; &quot; &quot; &quot;</td>
<td></td>
</tr>
<tr>
<td>Step 8: #of nodes on final submodel:</td>
<td>2000</td>
</tr>
<tr>
<td>node# xcoord ycoord zcoord:</td>
<td>1</td>
</tr>
<tr>
<td>2 x y z</td>
<td></td>
</tr>
<tr>
<td>2 &quot; &quot; &quot; &quot;</td>
<td></td>
</tr>
<tr>
<td>: : : :</td>
<td></td>
</tr>
<tr>
<td>2000 &quot; &quot; &quot; &quot;</td>
<td></td>
</tr>
</tbody>
</table>

Step 6) (or use autoinput.exe for steps 6 to 9)

- Open NISA® file ‘temp.nis’ in Display3.
- Delete all layers except for one containing cutplane1.
- Save as cutplane1.nis.
- Edit NISA® file and delete all except for node numbers and co-ordinates
Appendix C: Procedure to Implement NFBC Submodelling Technique

- Copy into input file as shown above.

**Step 7)**

- Edit output solution file from section model (section.out).
- Delete all except for 'Reaction Forces and Moments at Nodes'.
- Delete all nodes except for those on the cutplane.
- Copy into input file as shown above.

**Step 8)**

- Create fine mesh in section model using Display3 to create final submodel and save as submodel.nis. *(Note: Maximum allowable number of nodes per cutplane is 1000.)*
- Delete all nodes except for those on cutplane1.
- Save newcutplane1.nis
- Edit NISA© file and delete all except for node numbers and co-ordinates.
- Copy into input file as shown above.

**Step 9)**

- Count number of nodes in cutplanes of section model and submodel and include in input file as shown above.
- Save input file as sm.dat.
Appendix C: Procedure to Implement NFBC Submodelling Technique

**Autoinput - Automates Input File For NFBC Algorithm**

This code creates the formatted input text file for the NFBC algorithm. It takes as its input the name of the files 'cutplanex.nis', 'section.out', and 'submodel.nis' as described in the instructions on the NFBC sub-modeling procedure (MSWord doc). This code automates steps 6 to 9 in these instructions and must be utilized for each cutplane in the model.

**Input Files**
- Section Cutplane NISA File (NISA file containing only orig. nodes on cutplane x)
- Section Model Output File (same file for all cutplanes in model)
- Sub-Model Cutplane NISA File (NISA file containing only new nodes on cutplane x)

**Output Files**
- The output file generated by this code is automatically named 'sm.dat' as that is the input file accepted by NFBC.exe. The output of NFBC.exe is 'sm.out' which should be renamed cutplane_x.ses where x is the number of the cutplane.

![Figure C.2 User Interface for Autoinput.exe Code.](image)

**Step 10)**

- Run NFBC.exe to execute algorithm.
- Output file is named 'sm.out' – delete extra information at end of the file and rename as cutplane1.ses.

**Step 11)**

- Open submodel.nis and read in all cutplane.ses files.
- Save.
- Run.
- Perform Post Analysis procedures.

**Notes:** If shell elements are used, cutplanes should be created as lines which vary in more than one dimension. e.g. (this will not work): x coordinate and y coordinate are fixed and only the z coordinate varies for all nodes in a particular cutplane.

Codes: 'nisa-to-disp.exe', 'dis-to-ses.exe' and 'autoinput.exe' automate most of the manual text manipulation described above. These programs were written and compiled in Borland C++.