

# Deformation and stress analysis of gate valves

Chua, Beng Yeow

2009

Chua, B. Y. (2009). Deformation and stress analysis of gate valves. Master's thesis,  
Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/47214>

<https://doi.org/10.32657/10356/47214>

---

Nanyang Technological University

*Downloaded on 03 Apr 2024 09:21:18 SGT*

# Deformation and Stress Analysis of Gate Valves

CHUA BENG YEOW

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

A thesis submitted to the Nanyang Technological University  
in fulfillment of the requirement for the degree of  
Master of Engineering

2009



---

## ACKNOWLEDGEMENT

The author would like to express his gratitude to the following people who have assisted him throughout the project:

Associate Professor Chai Gin Boay, his NTU supervisor, for his patience, guidance and valuable advice in making the project successful.

Staff of Cooper Cameron Singapore Pte Ltd, for providing adequate information, ideas and suggestions.

Cooper Cameron Singapore Pte Ltd, for giving the author the opportunity to undergo training at their facilities and participate in the project.

Other NTU staff and lecturers, for their kind assistance throughout the project.



## TABLE OF CONTENTS

Acknowledgement	i
Abstract	ii
List of Figures	iii
List of Tables	viii
List of Graphs	ix
CHAPTER 1	
PROJECT INTRODUCTION	1
1.1 Background	1
1.2 Objectives	2
1.3 Scope	2
1.4 Methodology	3
CHAPTER 2	
LITERATURE REVIEW	5
2.1 Standards (Existing Design Guides)	5
2.1.1 ANSI/API Specification 6A	5
2.1.2 ASME Boiler & Pressure Vessel Code, VIII, Division 1	13
2.2 Gate Valves	17
2.2.1 Types of gates	17
2.2.2 Types of gate valves	19
2.3 Block valves	24
CHAPTER 3	
AUTO-FEA PROGRAM	27
3.1 Introducing Auto-FEA	27






---

3.2	Running the Auto-FEA program (Launch Level)	29
3.3	Running the Auto-FEA program (Processing Level)	32
3.4	Running the Auto-FEA program (Post-Processing Level)	39
3.5	ANSYS Analysis Methodology	45
3.6	ANSYS details of sub-programs	49
 CHAPTER 4		
	ANALYSIS METHODOLOGY	77
4.1	Models	77
4.2	Objectives	78
4.3	Properties	80
4.4	Variables	83
4.5	Finite Element Analysis	84
 CHAPTER 5		
	RESULTS AND DISCUSSION	85
5.1	Parallel Bores	85
5.2	Valve Cavity & Perpendicular Intersecting Bores(non-through)	94
 CHAPTER 6		
	FRACTURE MECHANICS APPROACH TO ANALYSE AS-CLAD VALVES	100
6.1	Simulation of welds	102
6.2	Experimental determination of critical fracture toughness	106
6.3	FEA simulation of cracks	108
6.4	Fracture mechanics FEA model results & discussion	110
 CHAPTER 7		
	CONCLUSION	123
REFERENCES		

---



---

## ABSTRACT

Fluid transfer valves are used in petroleum installations worldwide such as subsea and surface operations. Gate valves utilize a sliding closure where a control member called a gate slides across a general passageway in order to control fluid flow. Gate valves are used when a straight-line flow of fluid and minimum restriction is desired.

Cooper Cameron Singapore Pte Ltd is an industry leader in the design, manufacture and installation of large-bore gate valves for subsea and surface applications. The company encounters certain issues regarding the manufacture and production of their gate valves and block valves. They require a program to evaluate code compliance and act as a design tool for their block valves based on finite element analysis. Hence, Cooper Cameron has tied up with the author in a project to come up with new parametric software.

The project aimed to design a program that interfaces with ANSYS and allows a user not trained in finite element modeling to carry out such analysis. Two block valve configurations were analyzed and parametric studies carried out to study the effects of temperature, pressure and wall thickness on the stresses in the block valve. Generally, when the internal temperatures and pressures of the bores increase, the Von Mises stress would increase. However, there are exceptions to this trend when the compressive stresses on the bores caused by internal pressure cancel out part of the thermal stresses. A consistent observation was made that when wall thickness between bores or between a seat and a bore decreases, the Von Mises stress increase.

In addition, an investigation was carried out on the possibility of cracking on Cameron As-clad valves if no machining was done to smoothen the weld clads. Results show that the interface between the weld beads do not propagate as cracks into the wall thickness of the valves. Manufacturing cost and time will be greatly reduced since this machining process is not required.



## LIST OF FIGURES

Figure 1.1	Design Methodology	3
Figure 2.1	Expanding style gate (left) and Slab style gate (right)	17
Figure 2.2	Open position (left) and closed position (right) of gate	18
Figure 2.3	M Gate Valve	19
Figure 2.4	M215 Gate Valve	20
Figure 2.5	FLS-R Gate Valve	21
Figure 2.6	FL Gate Valve	22
Figure 2.7	FLS Gate Valve	23
Figure 2.8	Christmas tree configuration	25
Figure 2.9	Block valve with other equipment attached	26
Figure 2.10	Block valve	26
Figure 3.1	Launcher	30
Figure 3.2	Main Menu of sub-program	32
Figure 3.3	Visual Reference Menu	33
Figure 3.4	Choosing classification of cavity	34
Figure 3.5	Textboxes defining classification of cavities	35
Figure 3.6	Section displaying material properties	36
Figure 3.7	Section displaying Loads	36
Figure 3.8	Choice of choosing parameter units	37
Figure 3.9	Output Window	38
Figure 3.10	Graphical plots	39
Figure 3.11	Listview showing results	40
Figure 3.12	Report Menu	42
Figure 3.13	Warning message	43
Figure 3.14	Prompt for choosing reference model	43
Figure 3.15	Prompt for choosing graphical plots	43
Figure 3.16	Prompt for choosing Excel file	43





Figure 3.17	Plane35 and Plane2 elements	46
Figure 3.18	Solid87 and Solid92 elements	47
Figure 3.19	Reference model for parallel bores	49
Figure 3.20	Meshing of model	50
Figure 3.21	Boundary conditions on model	51
Figure 3.22	Visual Reference for Perpendicular Out of Plane Bores	53
Figure 3.23	Meshing of model	54
Figure 3.24	Boundary conditions on model	55
Figure 3.25	Reference model for Perpendicular Out of Plane Bores	56
Figure 3.26	Meshing of model	57
Figure 3.27	Boundary conditions on model	58
Figure 3.28	Reference model for Valve Cavity & Perpendicular Intersecting Bores (Through)	59
Figure 3.29	Meshing of model	60
Figure 3.30	Boundary conditions on model	61
Figure 3.31	Reference model for Valve Cavity & Perpendicular Intersecting Bores (Non-through)	62
Figure 3.32	Meshing of model	63
Figure 3.33	Boundary conditions on model	64
Figure 3.34	Reference model for Valve Cavity & Offset Perpendicular Intersecting Bores	65
Figure 3.35	Meshing of model	66
Figure 3.36	Boundary conditions on model	67
Figure 3.37	Reference model for Valve Cavity & Angles Intersecting Bores	68
Figure 3.38	Meshing of model	69
Figure 3.39	Boundary conditions on model	70
Figure 3.40	Reference model for 2 Valve Cavities & Angled Intersecting Bores	71
Figure 3.41	Meshing of model	72
Figure 3.42	Boundary conditions on model	73



Figure 3.43	Reference model for 2 Valve Cavities & Offset Perpendicular Intersecting Bores	74
Figure 3.44	Meshing of model	75
Figure 3.45	Boundary conditions on model	76
Figure 4.1	Parallel Bores	77
Figure 4.2	Cavity & Perpendicular intersecting bores (non-through)	78
Figure 4.3	Wall thickness for Parallel bores	79
Figure 4.4	Wall thickness for Valve Cavity & Perpendicular intersecting bores (non-through)	79
Figure 4.5	Geometric parameter labels for Valve Cavity & Perpendicular intersecting bores (non-through)	81
Figure 5.1	Contour stress plot	85
Figure 5.2	Contour stress plot zoom in	86
Figure 5.3	Tensile thermal stresses	92
Figure 5.4	Tensile thermal stresses counteracted by compressive stress	92
Figure 5.5	Contour stress plot	94
Figure 5.6	Contour stress plot zoom in	95
Figure 6.1	Symmetrical view of As-clad valve	100
Figure 6.2(a)	Actual geometry	102
Figure 6.2(b)	Simplified geometry	102
Figure 6.3	Crack simulation	103
Figure 6.4	Comparison of the actual with simplified model	104
Figure 6.5	Effects of tension and shearing	104
Figure 6.6	Experimental setup	106
Figure 6.7	Specimen used in $J_{IC}$ testing	107
Figure 6.8	Axisymmetric model	108
Figure 6.9	Boundary constraints due to axisymmetry	109
Figure 6.10	Crack locations	110
Figure 6.11	Scenario 1 location 1	111



---

Figure 6.12	Scenario 1 location 2	112
Figure 6.13	Scenario 2 location 1	114
Figure 6.14	Scenario 2 location 2	115
Figure 6.15	Scenario 3 location 1	116
Figure 6.16	Scenario 3 location 2	117
Figure 6.17	Scenario 4 location 1	119
Figure 6.18	Scenario 4 location 2	120
Figure 6.19	Different modes of fracture	122



## LIST OF TABLES

Table 2.1	Maximum rated working pressures	6
Table 2.2	Temperature ratings	7
Table 2.3	Standard material applications for bodies, bonnets and end and outlet connections	12
Table 2.4	Standard material property requirements for bodies, bonnets and end and outlet connections	12
Table 4.1	Properties used for Parallel Bores	80
Table 4.2	Properties used for Valve Cavity & Perpendicular intersecting bores (non-through)	82
Table 4.3	Variables used in parametric study for Parallel Bores	83
Table 4.4	Variables used in parametric study for Valve Cavity & Perpendicular intersecting bores (non-through)	83
Table 6.1	Input parameters for Scenario 1	111
Table 6.2	Stress intensity values for Scenario 1 location 1	112
Table 6.3	Stress intensity values for Scenario 1 location 2	113
Table 6.4	Stress intensity values for Scenario 2 location 1	115
Table 6.5	Stress intensity values for Scenario 2 location 2	115
Table 6.6	Stress intensity values for Scenario 3 location 1	117
Table 6.7	Stress intensity values for Scenario 3 location 2	117
Table 6.8	Stress intensity values for Scenario 4 location 1	119
Table 6.9	Stress intensity values for Scenario 4 location 2	120
Table 6.10	Summary of results from different scenarios & locations	121





## LIST OF GRAPHS

Graph 5.1	Variation of Von Mises Stress vs Wall thickness (for 13.8 MPa bore pressure)	87
Graph 5.2	Variation of Von Mises Stress vs Wall thickness (for 34.5 MPa bore pressure)	87
Graph 5.3	Variation of Von Mises Stress vs Wall thickness (for 69 MPa bore pressure)	88
Graph 5.4	Variation of Von Mises Stress vs Wall thickness (for 138 MPa bore pressure)	88
Graph 5.5	Variation of Von Mises Stress vs Wall thickness (for 275K bore temperature)	89
Graph 5.6	Variation of Von Mises Stress vs Wall thickness (for 334.5K bore temperature)	89
Graph 5.7	Variation of Von Mises Stress vs Wall thickness (for 394K bore temperature)	90
Graph 5.8	Variation of Von Mises Stress vs Wall thickness (for 20.7 MPa flow bore pressure)	96
Graph 5.9	Variation of Von Mises Stress vs Wall thickness (for 69 MPa flow bore pressure)	96
Graph 5.10	Variation of Von Mises Stress vs Wall thickness (for 103.5 MPa flow bore pressure)	97
Graph 5.11	Variation of Von Mises Stress vs Wall thickness (for 300K flow bore temperature)	97
Graph 5.12	Variation of Von Mises Stress vs Wall thickness (for 345K flow bore temperature)	98
Graph 5.13	Variation of Von Mises Stress vs Wall thickness (for 394K flow bore temperature)	98





# 1 PROJECT INTRODUCTION

## 1.1 BACKGROUND

---

Fluid transfer valves are used in petroleum installations worldwide such as subsea and surface operations. Although many different types of valves are used to control the flow of fluids, the basic valve types can be divided into two general groups: stop valves and check valves. Stop valves are used to shut off the flow of fluid and are controlled by the movement of the valve stem. Stop valves can be divided into four general categories: globe, gate, butterfly, and ball valves.

Gate valves utilize a sliding closure where a control member called a gate slides across a general passageway in order to control fluid flow. Gate valves are used when a straight-line flow of fluid and minimum restriction is desired. When the valve is wide open, the gate is fully drawn up into the valve, leaving a straight-through, unobstructed passageway for flow through the valve. Therefore, there is little pressure drop, turbulence and flow restriction through the valve. Gate valves are not suitable for throttling purposes as the key valving components have a tendency to erode quickly due to the high velocity fluid passing through a narrowly restricted area.

Cooper Cameron Singapore Pte Ltd is an industry leader in the design, manufacture and installation of large-bore gate valves for subsea and surface applications. Cameron FLS and FLS-R gate valves have been used in demanding applications from deepwater subsea manifolds in the Gulf of Mexico, to harsh environments in the North Sea. They provide a wide range of gate valves and block valves depending on the type of gate design.

There are several issues raised by Cooper Cameron regarding the manufacture and production of their gate valves and block valves. One is regarding the code compliance



during production of valves. When some valves do not pass quality control due to certain dimensions, the whole valve is either re-evaluated by senior engineers on the reliability of the valve or it will be scrapped. This results in loss of material and high manufacturing costs and time. Secondly, it is important to reduce the dimensions of gate and block valves while still adhering to industrial standards. Reducing dimensions such as overall block size, seat and gate thickness will reduce the cost of manufacture. Therefore, a finite element analysis program would be useful in the design of future valves. Thirdly, they seek to develop a program where the user is able to carry out finite element analysis without knowing the basics of FEM or usage of FEA software.

These issues form the objectives and scope of this Master of Engineering thesis report.

## **1.2 OBJECTIVES**

---

The primary objective of the project is to create an analytical simulation tool or program using finite element modeling to solve technical problems arising from the design, manufacturing, servicing and repair of the gate or block valves. This program should be capable of evaluating code compliance and assisting in parametric studies. It should allow the user to manipulate input parameters and obtain relevant design outputs and information.

An additional minor objective of the project is to evaluate the possibility of cracking on Cameron As-clad valves if there is no machining done to smoothen the weld clads. Manufacturing cost and time can be greatly reduced if this machining process is not required.

## **1.3 SCOPE**

---

A wide range of disciplines was involved in this project. Learning and knowledge of a programming software such as Visual Basic was emphasized during the initial stages of



program development. The basis of the programs' finite element modeling were produced using ANSYS Parametric Design Language. Finite Element Analysis (FEA) was carried out on the block valves using ANSYS software to determine their structural stability. Fracture mechanics approach using FEA was then used to investigate possibility of crack propagation in the As-clad valves.

### 1.4 METHODOLOGY

The following flow chart below illustrates the methodology applied by the author to achieve the project objectives.

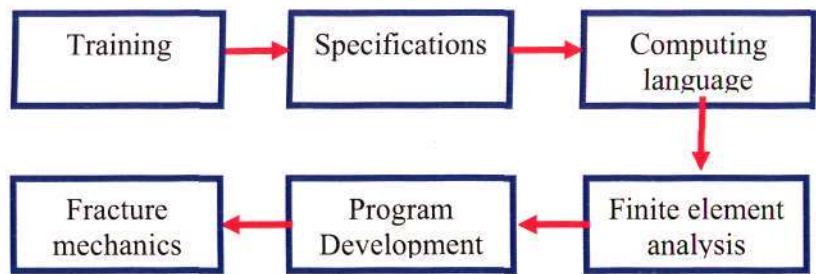


Figure 1.1 Design Methodology

#### ▪ Training

At the initial stage, the author worked in Cooper Cameron's valve assembly and test area. He studied the existing gate and block valve designs and gain an in-depth understanding of the functions, assembly and mechanism of the various components. He investigated the technical problems that could possibly arise throughout the development cycle of the gate and block valves. He included a thorough study of the relevant industry codes, specifically API Specification 6A/ISO 10423 "Specification for Wellhead and Christmas Tree Equipment" [1], and other relevant design codes referenced within, including key sections of the ASME BPVC.





- **Specifications**

For this stage, discussions were undertaken with engineers from Cooper Cameron Singapore Pte Ltd (R&D department) to determine the functional requirements and of the parametric software. The type of block valves to be analyzed and developed as part of the programs was decided.

- **Computing language**

During this stage, the different types of programming language that could be interfaced with ANSYS were explored. Visual Basic was chosen as the main programming software which can interface and run ANSYS in the background to give finite element analysis results.

- **Program development**

The Auto-FEA program was created with nine sub-programs inside to cater to different block valves. Additional features were created at the request of Cooper Cameron such as program menus and interfacing of Auto-FEA with local database systems.

- **Finite element Analysis (FEA)**

After consultation and discussion with engineers from Cooper Cameron, 2 main areas of the block valves were to be analyzed and parametric studies carried out. The models to be analyzed are the Parallel Bores and the Valve Cavity and Perpendicular intersecting bore. The effects of temperature, pressure and wall thickness were to be investigated.

- **Fracture mechanics using FEA**

The interfaces between weld beads on the weld clad section of the As-clad valves were modeled as cracks and the validity of this simulation was verified through FEA. After which, analysis was carried out to determine if crack propagation will be initiated at weld bead interfaces for current As-clad valve specifications.



---

## 2. LITERATURE REVIEW

---

A literature review was undertaken to obtain all relevant information useful to the cause of the project. The review included research journals, publications, brochures and established standards and information from the Internet. The important findings were summarized in the following sections:

- 1) Standards – These are existing design guides which helped the author in designing the structure.
- 2) Gate valves – All relevant gate valves to be used in the project were researched and their features explained.
- 3) Block valves – These special block configurations of gate valves were introduced.

### 2.1 Standards (Existing Design Guides)

---

#### 2.1.1 ANSI/API Specification 6A – Specification for Wellhead and Christmas Tree Equipment - ISO 10423:2003, (Modified) Petroleum and natural gas industries — Drilling and production equipment — Wellhead and Christmas tree equipment

- Clause 4.2 Service conditions
- Clause 4.2.1 Pressure ratings
- Clause 4.2.1.1 General

Equipment shall be designed to operate at only the following maximum rated working pressures:



MPa	Psi
13,8	2 000
20,7	3 000
34,5	5 000
69,0	10 000
103,5	15 000
138,0	20 000

**Table 2.1      Maximum rated working pressures**

- Clause 4.2.2    Temperature ratings
- Clause 4.2.2.1    General

Equipment shall be designed to operate in one or more of the specified temperature ratings with minimum and maximum temperatures as shown in Table 2.2.

Minimum temperature is the lowest ambient temperature to which the equipment may be subjected. Maximum temperature is the highest temperature of the fluid that may directly contact the equipment.

- Clause 4.2.2.2    Design considerations

The design shall consider the effects of differential thermal expansion from temperature changes and temperature gradients which the equipment would experience in service. Design for high temperature rating, e.g. X and Y, shall take into consideration the effects of temperature on strength levels, see annex G for guidelines.

- Clause 4.2.2.3    Temperature rating considerations

Choosing the temperature rating is the ultimate responsibility of the user. In making these selections, the user should consider the temperature the equipment would experience in drilling and/or production services.





Temperature classification	Operating range			
	°C		°F	
	min.	max.	min.	max.
K	-60	82	-75	180
L	-46	82	-50	180
P	-29	82	-20	180
R	Room temperature		Room temperature	
S	-18	66	0	150
T	-18	82	0	180
U	-18	121	0	250
V	2	121	35	250

Table 2.2      Temperature ratings

- Clause 4.3            Design methods
- Clause 4.3.3        Other end connectors, bodies and bonnets
- Clause 4.3.3.1    General

Other end connectors, bodies and bonnets that utilize standard materials (in designs other than those specified in this International Standard) shall be designed in accordance with one or more of the following methods. Standard materials are those materials whose properties meet or exceed the requirements of Table 2.4.

Other end connectors, bodies and bonnets that utilize non-standard materials shall be designed in accordance with the requirements of 4.3.3.6. Non-standard materials are materials with specified minimum yield strength in excess of 517 MPa (75 000 psi) that do not meet the ductility requirements of Table 2.4 for standard 75K materials.



In the event stress levels calculated by the methods in 4.3.3.2 to 4.3.3.6 exceed the allowable stresses, other methods identified by the manufacturer shall be used to justify these stresses. Fatigue analysis and localized bearing stress values are beyond the scope of this International Standard.

▪ Clause 4.3.3.2 ASME method

The design methodology as described in ASME, Section VIII, Division 2, Appendix 4, may be used for design calculations for pressure-containing equipment. Design-allowable stresses shall be limited by the following criteria:

$$S_T = 0,83 S_Y \text{ and } S_m = \frac{2S_Y}{3}$$

where

$S_m$  = design stress intensity at rated working pressure;

$S_T$  = maximum allowable general primary membrane stress intensity at hydrostatic test pressure;

$S_Y$  = material specified minimum yield strength.

▪ Clause 4.3.3.3 Theory of constant energy of distortion

The theory of constant energy of distortion, also known as the Von Mises Law, may be used for design calculations for pressure-containing equipment. Rules for the consideration of discontinuities and stress concentrations are beyond the scope of this international Standard. However, the basic pressure-vessel wall thickness may be sized by combining triaxial stresses based on hydrostatic test pressure and limited by the following criterion:





$$S_E = S_Y$$

where

$S_E$  is the maximum allowable equivalent stress at the most highly stressed distance into the pressure vessel wall, computed by the distortion energy theory method;

$S_Y$  is the material-specified minimum yield strength.

- Clause 4.3.3.4 Experimental stress analysis

Experimental stress analysis as described in ASME, Section VIII, Division 2, Appendix 6 may be used as an alternative method to those described in 4.3.3.2 and 4.3.3.3.

- Clause 4.3.3.5 Design qualification by proof test
- Clause 4.3.3.5.1 General

As an alternative to the analytical methods above, the pressure rating of equipment may be determined by the use of a hydrostatic test at elevated pressure. A test vessel, or vessel part, is made from the equipment for which the maximum allowable working pressure is to be established. It shall not previously have been subjected to a pressure greater than 1.5 times the desired or anticipated maximum allowable working pressure.

- Clause 4.3.3.5.2 Determination of yield strength

- a) Method

The yield strength of the material in the part tested shall be determined in accordance with the method prescribed in the applicable material specification.

- b) Specimen preparation

Yield strength so determined shall be the average from three or four specimens cut from the part tested after the test is completed. The specimens shall be cut from a location



here the stress during the test has not exceeded the yield strength. The specimens shall not be flame-cut because this might affect the strength of the material.

c) Alternative specimens

If excess stock from the same piece of material is available and has been given the same heat treatment as the pressure part, the test specimens may be cut from this excess stock. The specimen shall not be removed by flame cutting or any other method involving sufficient heat to affect the properties of the specimen.

d) Exemption

If yield strength is not determined by test specimens, an alternative method is given in 4.3.3.5.3 for evaluation of proof test results to establish the maximum allowable working pressure.

▪ Clause 4.3.3.6 Non-standard materials design requirements

The design methodology as described in ASME, Section VIII, Division 2, Appendix 4, shall be used for design and calculations for pressure-containing equipment utilizing non-standard materials. Design allowable stresses shall be limited by the following criteria:

$$S_T = \text{the smaller of } \frac{5}{6} S_Y \text{ or } \frac{2}{3} R_{m,\min}$$

$$S_m = \text{the smaller of } \frac{2}{3} S_Y \text{ or } \frac{1}{2} R_{m,\min}$$

$$S_S = \text{the smaller of } 2 S_Y \text{ or } R_{M,\min}$$

where



$S_m$  is the design stress intensity at rated working pressure;

$S_S$  is the maximum combined primary and secondary stress intensity;

$S_T$  is the maximum allowable general primary membrane stress intensity at hydrostatic test pressure;

$R_{m,min}$  is the material-specified minimum ultimate tensile strength;

$S_Y$  is the material-specified minimum yield strength.

- Clause 5.4 Bodies, bonnets, end and outlet connections
- Clause 5.4.1 Material

a) Tensile property requirements

All bodies, bonnets, end and outlet connections shall be fabricated from standard or non-standard materials as specified in Table 2.3. Standard materials shall meet the applicable properties shown in Table 2.4. Non-standard material shall conform to the manufacturer's written specification. The specification shall include minimum requirements for tensile strength, yield strength, elongation, reduction of area, toughness and hardness applicable for the specific alloy. All non-standard materials shall exceed 75K minimum yield strength and meet a minimum of 15 % elongation and 20 % reduction of area.



Part	Pressure ratings					
	MPa (psi)					
	13,8 (2 000)	20,7 (3 000)	34,5 (5 000)	69,0 (10 000)	103,5 (15 000)	138,0 (20 000)
Body, <sup>a</sup> bonnet	Material designation					
	36K, 45K	36K, 45K	36K, 45K	36K, 45K	45K, 60K	60K, 75K
	60K, 75K	60K, 75K	60K, 75K	60K, 75K	75K, NS	NS
	NS <sup>b</sup>	NS	NS	NS		

Table 2.3      Standard material applications for bodies, bonnets and end and outlet connections

Material designation	0,2 % Yield strength min.  MPa (psi)	Tensile strength min.  MPa (psi)	Elongated in 50mm (2 in) min.  %	Reduction in area min.  %
36K	248 (36 000)	483 (70 000)	21	No requirement
45K	310 (45 000)	483 (70 000)	19	32
60K	414 (60 000)	586 (85 000)	18	35
75K	517 (75 000)	655 (95 000)	17	35

Table 2.4      Standard material property requirements for bodies, bonnets and end and outlet connections





## 2.1.2 ASME Boiler & Pressure Vessel Code, VIII, Division 1, Rules for Construction of Pressure Vessels

- Clause UG-16 GENERAL

- (a) The design of pressure vessels and vessel parts shall conform to the general design requirements in the following paragraphs and in addition to the specific requirements for Design given in the applicable Parts of Subsections B and C.
- (b) *Minimum Thickness of Pressure Retaining Components.* Except for the special provisions listed below, the minimum thickness permitted for shells and heads, after forming and regardless of product form and material, shall be 1/16 in. (1.6mm) exclusive of any corrosion allowance. Exceptions are:
  - (1) the minimum thickness does not apply to heat transfer plates of plate-type heat exchangers;
  - (2) this minimum thickness does not apply to the inner pipe of double pipe heat exchange nor to tubes in shell-and-tube heat exchangers, where such pipes or tubes are NPS 6 (DN 150) and less. This exemption applies whether or not the outer pipe or shell is constructed to Code rules. All other pressure parts of these heat exchangers which are constructed to Code rules must meet the 1/16 in. (1.6mm) minimum thickness requirements.
  - (3) the minimum thickness of shells and heads of unfired steam boilers shall be 1/4in. (6mm) exclusive of any corrosion allowance;
  - (4) the minimum thickness of shells and heads used in compressed air service, steam service, and water service, made from materials listed in Table UCS-23, shall be 3/23 in. (2.4mm) exclusive of any corrosion allowance.

- Clause UG-20 DESIGN TEMPERATURE

- (a) *Maximum.* Except as required in UW-2(d)(3), the maximum temperature used in design shall be not less than the mean metal temperature (through the thickness)



expected under operating conditions for the part considered (see 3-2). If necessary, the metal temperature shall be determined by computation or by measurement from equipment in service under equivalent operating conditions.

- (b) *Minimum.* The minimum metal temperature used in design shall be the lowest expected in service except when lower temperatures are permitted by the rules of this Division (see UCS-66 and UCS-160). The minimum mean metal temperature shall be determined by the principles described in (a) above. Consideration shall include the lowest operating temperature, operational upsets, autorefrigeration, atmospheric temperature, and any other sources of cooling [except as permitted in (f)(3) below for vessels meeting the requirements of (f) below]. The MDMT marked on the nameplate shall correspond to a coincident pressure equal to the MAWP. When there are multiple MAWP's, the largest value shall be used to establish the MDMT marked on the nameplate. Additional MDMT's corresponding with other MAWP's may also be marked on the nameplate.

▪ Clause UG-21 DESIGN PRESSURE

Vessels covered by this Division of Section VIII shall be designed for at least the most severe condition of coincident pressure and temperature expected in normal operation. For this condition and for test conditions, the maximum difference in pressure between the inside and outside of a vessel, or between any two chambers of a combination unit, shall be considered [see UG-98, UG-99(e), and 3-2].

▪ Clause UG-22 LOADINGS

The loadings to be considered in designing a vessel shall include those from:

- (a) internal or external design pressure (as defined in UG-21);  
(b) weight of the vessel and normal contents under operating or test conditions (this includes additional pressure due to static head of liquids);





- (c) superimposed static reactions from weight of attached equipment, such as motors, machinery, other vessels, piping, linings, and insulations;
- (d) the attachment of:
  - (1) internals (see Appendix D);
  - (2) vessel supports, such as lugs, rings, skirts, saddles, and legs (see Appendix G);
- (e) cyclic and dynamic reactions due to pressure or thermal variations, or from equipment mounted on a vessel, and mechanical loadings;
- (f) wind, snow, and seismic reactions, where required;
- (g) impact reactions such as those due to fluid shock;
- (h) temperature gradients and differential thermal expansions;
- (i) abnormal pressure, such as those caused by deflagration.

▪ Clause UG-27 THICKNESS OF SHELLS UNDER INTERNAL PRESSURE

- (a) The thickness of shells under internal pressure shall be not less than that computed by the following formulas. In addition, provision shall be made for any of the other loadings listed in UG-22, when such loadings are expected. (See UG-16.)
- (b) The symbols defined below are used in the formulas of this paragraph.

$t$  = minimum required thickness of shell, in.

$P$  = internal design pressure (see UG-21), psi

$R$  = inside radius of the shell occurs under consideration, in.

$S$  = maximum allowable stress value, psi (see UG-23 and the stress limitations specified in UG-24)

$E$  = joint efficiency for, or the efficiency of, appropriate joint in cylindrical or spherical shells, or the efficiency of ligaments between openings, whichever is less.

For welded vessels, use the efficiency specified in UW-12.

For ligaments between openings, use the efficiency calculated by the rules given in UG-53.



- (c) *Cylindrical Shells*. The minimum thickness or maximum allowable working pressure of cylindrical shells shall be the greater thickness or lesser pressure as given by (1) or (2) below.

(1) *Circumferential Stress (Longitudinal Joints)*. When the thickness does not exceed one-half of the inside radius, or  $P$  does not exceed  $0.385SE$ , the following formulas shall apply:

$$t = PR/(SE - 0.6P) \text{ or } P = SEt/(R + 0.6t) \quad (1)$$

(2) *Longitudinal Stress (Circumferential Joints)*. When the thickness does not exceed one-half of the inside radius, or  $P$  does not exceed  $1.25SE$ , the following formulas shall apply:

$$t = PR/(2SE + 0.4P) \text{ or } P = 2SEt/(R - 0.4t) \quad (2)$$

- (d) *Spherical Shells*. When the thickness of the shell of a wholly spherical vessel does not exceed  $0.356R$ , or  $P$  does not exceed  $0.665SE$ , the following formulas shall apply:

$$t = PR/(2SE - 0.2P) \text{ or } P = 2SEt/(R + 0.2t) \quad (3)$$

- (e) When necessary, vessels shall be provided with stiffness or other additional means of support to prevent overstress or large distortions under the external loadings listed in UG-22 other than pressure and temperature.
- (f) A stayed jacket shell that extends completely around a cylindrical or spherical vessel shall also meet the requirements of UG-47(c).
- (g) Any reduction in thickness within a shell course or spherical shell shall be in accordance with UW-9.





## 2.2 Gate Valves

The different types of API 6A gate valves manufactured by Cameron were introduced in this section. The gate valves can be classified according to the type of gates used, expanding gates and slab gates (Figure 2.1).

### 2.2.1 Types of Gate

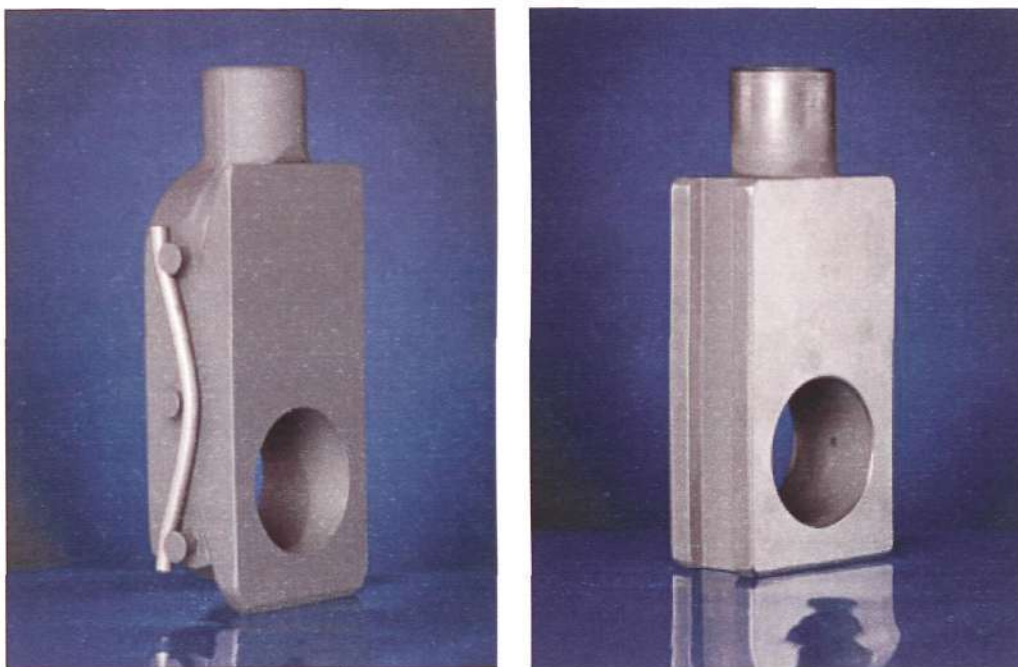
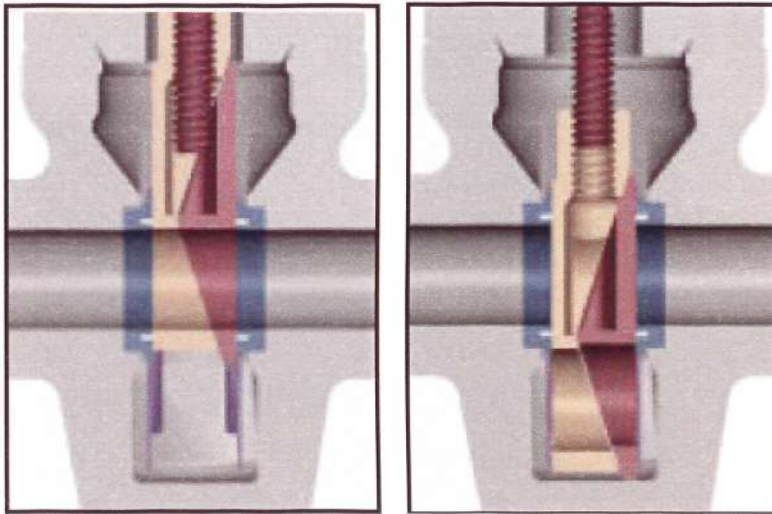


Figure 2.1 Expanding style gate (left) and Slab style gate (right)

#### ▪ Expanding Gate

Cameron M and M215 Gate Valves use expanding-style gates. When the hand wheel is tightened in manual valves, the expanding gate design creates a high seating force against both the upstream and downstream seats simultaneously as. A tight mechanical seal is then created which is unaffected by line pressure fluctuations or vibration. The expanding gate allows a positive mechanical seal across both seats both upstream and downstream, with or without line pressure.



**Figure 2.2** Open position (left) and closed position (right) of gate

The whole gate assembly uses an angular gate face which is collapsed during travel. Expanding gates provide a mechanical sealing method that ensures low pressure sealing. When the gate is closed, downward travel of the loose gate segment is stopped by the body. As the main gate segment continues downward, the two halves of the gate are forced apart by their tapered mating surfaces. Therefore, the faces of the gate assembly are forced outwards to effect a positive line flow seal.

When it is opened, upward travel of the loose gate segment is stopped by the bonnet. As the main gate segment continues upward, the two halves of the gate are forced apart by their tapered mating surfaces. With the bottom faces expanding, the gate is then sealed against the seats to isolate flow from the valve body cavity.

#### ▪ Slab Gate

Cameron FL, FLS and FLS-R Gate Valves use slab-style gates for both manual and actuated applications. These slab-style gates are one-piece, solid gates and are applicable for a wide range of bore sizes, rated working pressures and environments.



The slab-style gate design uses a metal-to-metal seal on the flow stream as the fluid pressure on one face of the gate forces the opposite face to contact the seat. It is ideal to be used for both low and high-pressure sealing situations where particulate matter could pose a sediment problem. When wireline cutting is a requirement, slab gates may also be used.

### 2.2.2 Types of gate valves

- **M Gate Valves**

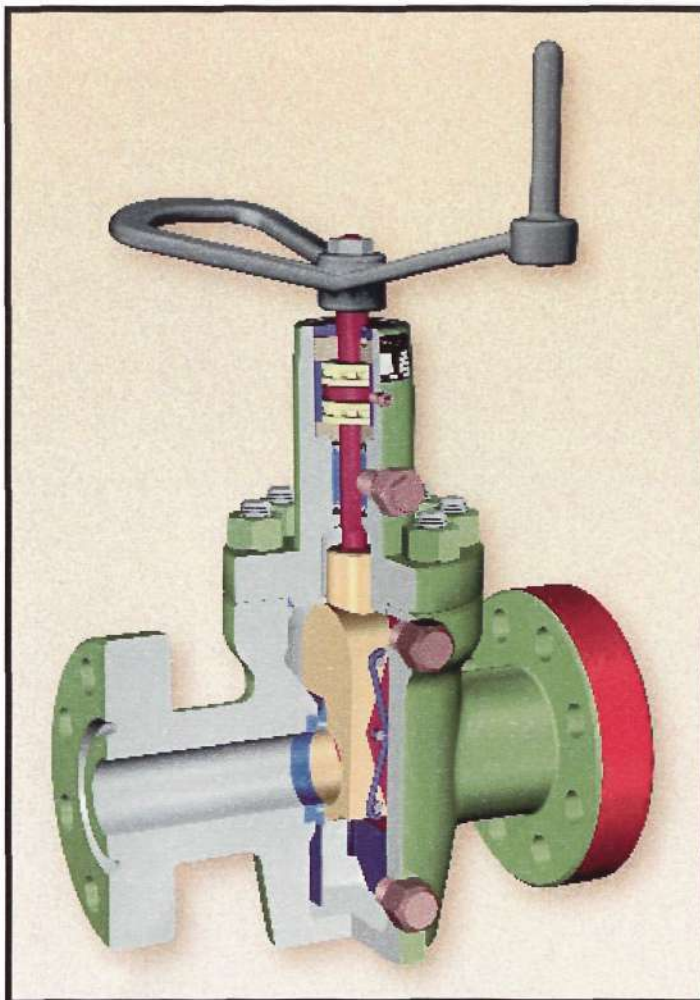


Figure 2.3 M Gate Valve

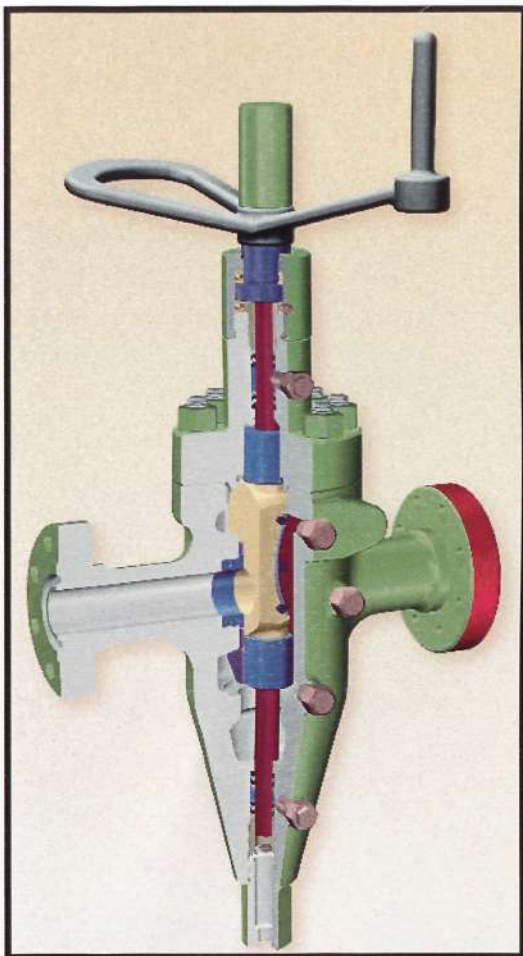




The Cameron M Gate Valve uses an expanding gate design with a non-rising stem and metal-to-metal sealing (gate-to-seat and seat-to-body). The non-rising stem allows the valve to be installed in close quarters with space constraints. The full-bore, through-conduit gate-to-seat seal almost eliminates turbulence and pressure drop, in turn maximizing the valve life. Torque is minimized as the upper and lower roller thrust bearings are isolated from well fluid.

It can be used in conditions of 13.8 to 34.5 MPa working pressure and available in sizes from 2-1/16" (0.0524 m) through 4-1/16" (0.1032 m) with either threaded or flanged ends.

▪ **M215 Gate Valve**



**Figure 2.4** M215 Gate Valve



The Cameron M215 Gate Valve uses an expanding gate design with a pressure balanced stem and metal-to-metal sealing (gate-to-seat and seat-to-body). The pressure balancing lower stem reduces load on the bearings and stem threads, giving a lower operating torque. Stem threads are outside the valve body, isolated from bore fluids. Regardless of the open/closed position of the valve, the positive mechanical seal of the gates isolates the body cavity from bore pressure, permitting pressure venting. Similar to the M Gate valves, the full-bore, through-conduit design virtually gives no turbulence and pressure drop.

It is used in applications with 69 MPa working pressure and is available in sizes from 1-13/16" (0.046m) through 3-1/16" (0.0778m) with flanged ends.

#### ▪ FLS-R Gate Valve

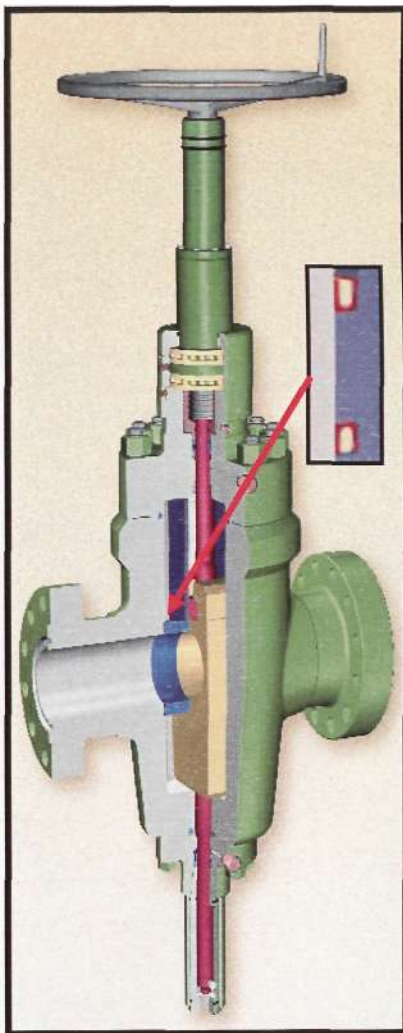


Figure 2.5 FLS-R Gate Valve



The Cameron FLS-R Gate Valve was designed for use as a manual valve in high pressure, large bore applications. It has a positive metal-to-metal sealing (gate-to-seat and seat-to-body) and its bi-directional design provides flow direction versatility. This valve incorporates a lower balancing stem and unique ball screw mechanism for ease of operation in the field. The lower stem balances the pressure thrust on the upper stem to reduce operating torque. This prevents body cavity pressure build-up during operation, and provides position indication. Two spring-loaded, pressure energized, non-elastomeric lip-seals between each seat and body assist in low pressure sealing. They also protect against intrusion of particle contaminants into the body cavity and seal areas. The FLS-R has many of the same features as the FLS including the gate and seat design.

#### ▪ FL Gate Valve

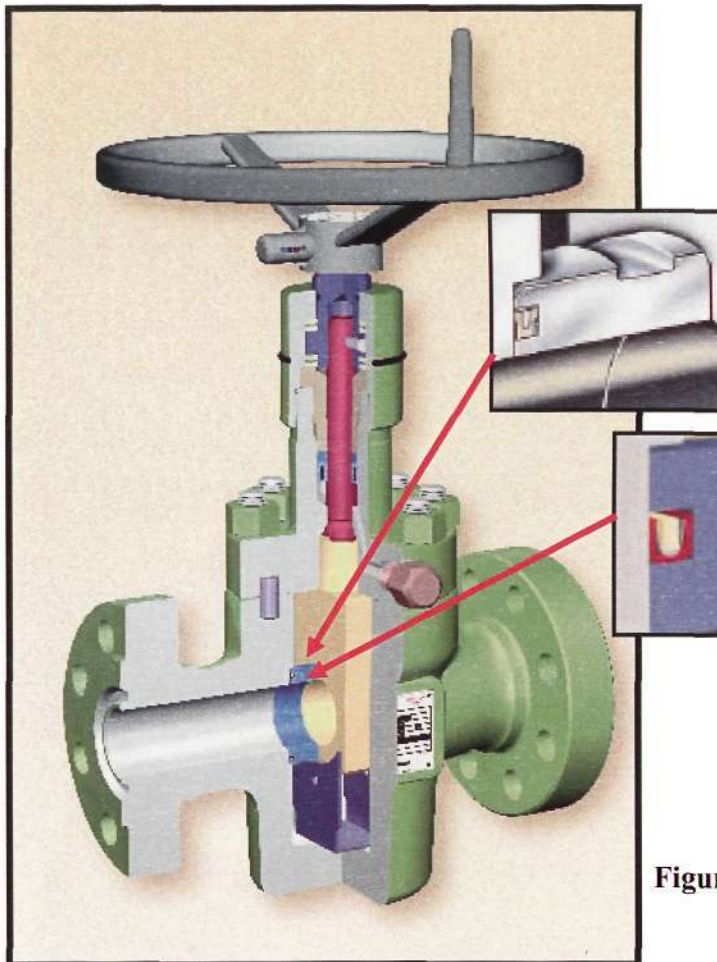


Figure 2.6 FL Gate Valve





The FL Gate Valve is a full-bore, through-conduit valve available in standard double flange, threaded-end and special block body configurations. It has a positive metal-to-metal sealing (gate-to-seat and seat-to-body) and its bi-directional design provides flow direction versatility. A single spring-loaded, pressure energized, non-elastomeric lip-seal between each seat and body assists in low pressure sealing. It also protects against intrusion of particle contaminants into the body cavity and seal areas. The stem shoulder can be backseated against the bonnet to seal off the stuffing box to allow stem seal replacement while the valve is under pressure.

It is a forged valve which is available in 13.8, 20.7 and 34.5 MPa working pressure and in bore sizes from 2-1/16"(0.0524m) to 4-1/16" (0.1032m).

#### ▪ FLS Gate Valve

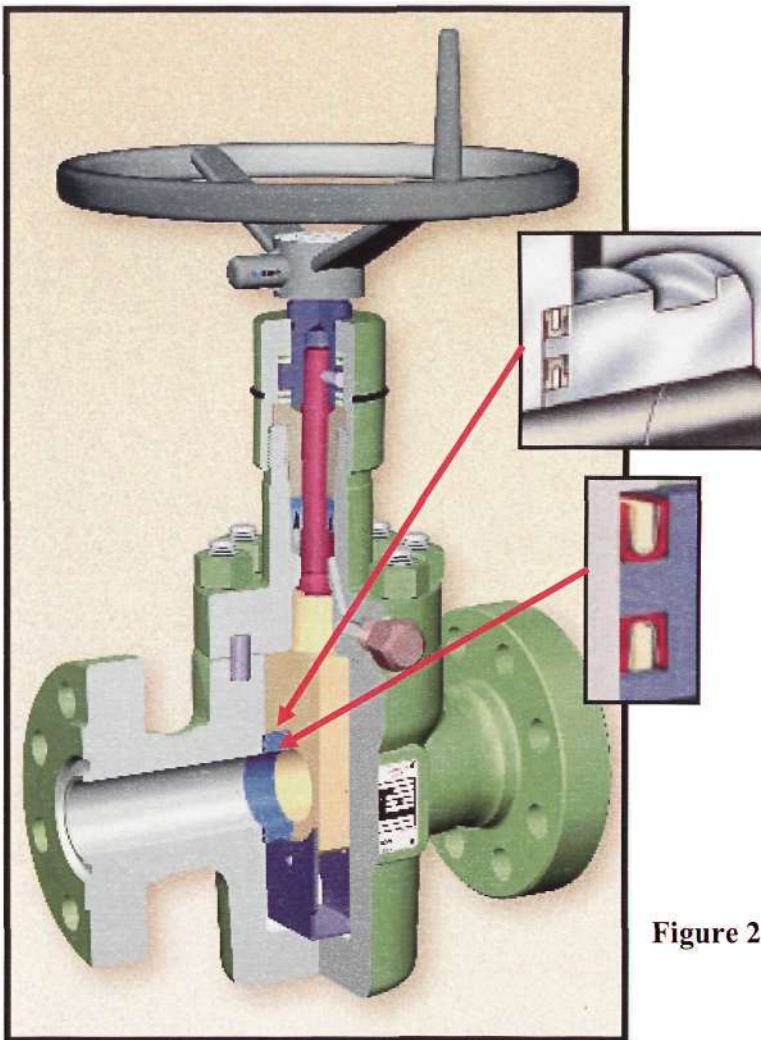


Figure 2.7 FLS Gate Valve



The FLS Gate Valve is a full-bore, through-conduit valve available in standard double flange, threaded-end and special block body configurations. Sealing at the gate-to-seat and seat-to-body interface is metal-to-metal. Different from the FL gate valve, it has two spring-loaded, pressure energized, non-elastomeric lip-seals between each seat and body to assist in low pressure sealing. They also protect against intrusion of particle contaminants into the body cavity and seal areas. The gate-and-seat assembly can be sealed in both directions, providing flow direction versatility and increased service life. The stem can also be backseated to allow stem seal replacement with the valve under pressure.

It is a forged valve available in pressure ratings from 13.8 to 138 MPa and bore sizes from 1-13/16" (0.046m) to 9" (0.2286m).

### **2.3 Block valves**

---

Block valves are essentially block configurations of gate valves. They include manual gate valves, remote gate valves and station block valves (suction valves and discharge valves). When closed, the valve can block oil flow in both directions. Block valves act as a connector between wellhead housings and other valves and equipment in a Christmas tree configuration (Figure 2.8 and 2.9). Its configuration can be changed by varying the number of bores through it, number and location of valve cavities. Figure 2.10 shows a 'raw' block valve with no gate valves or other equipment fixed on. These block valves will be the subjects for the programs to be created in Chapter 4 and analysed in Chapter 5.



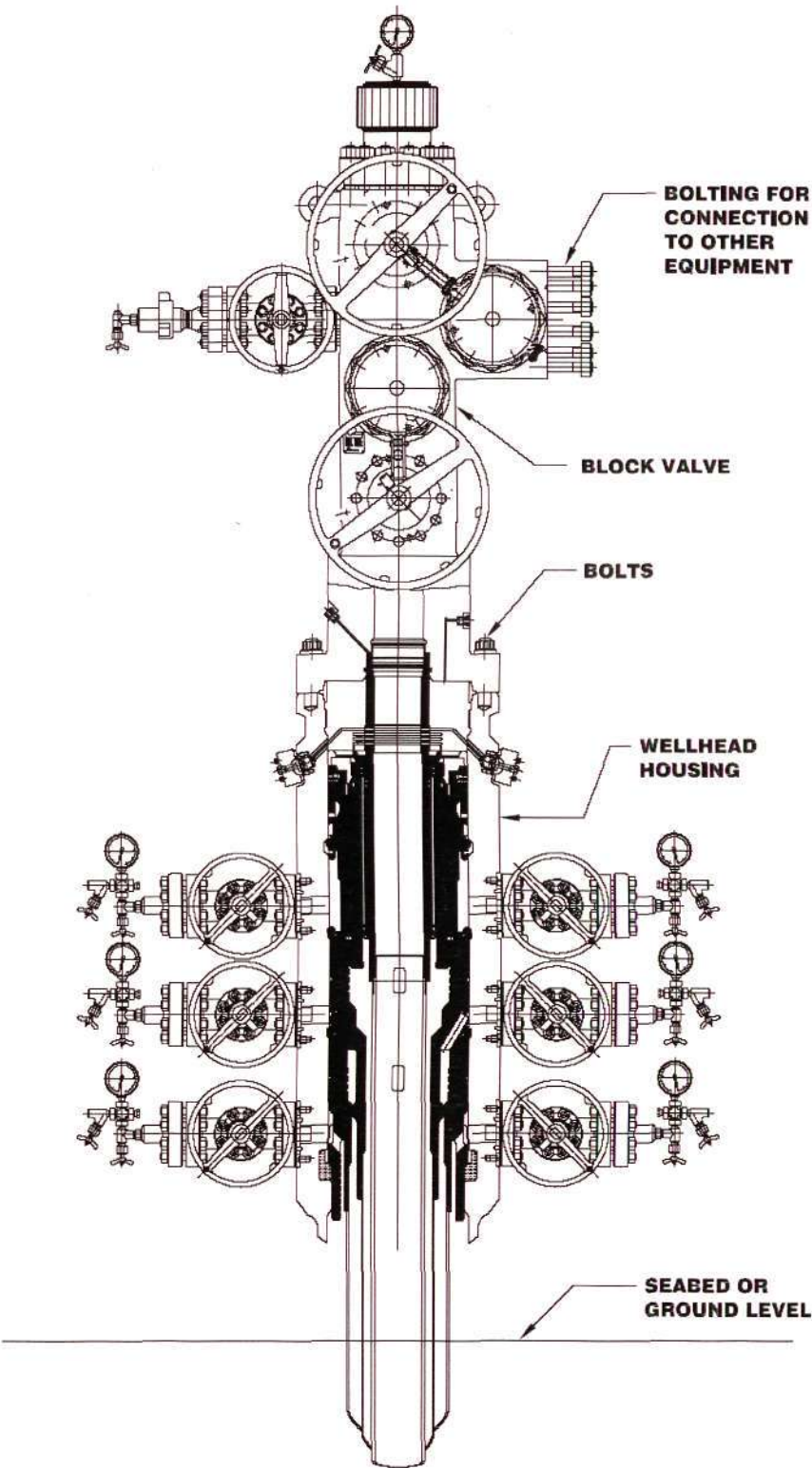


Figure 2.8 Christmas tree configuration

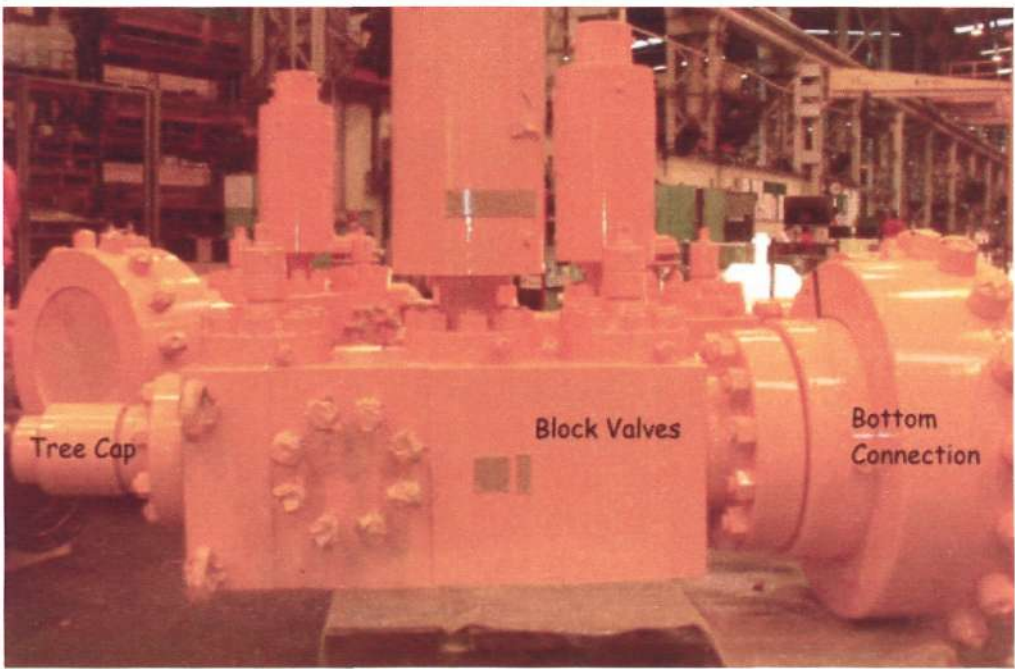


Figure 2.9 Block valve with other equipment attached

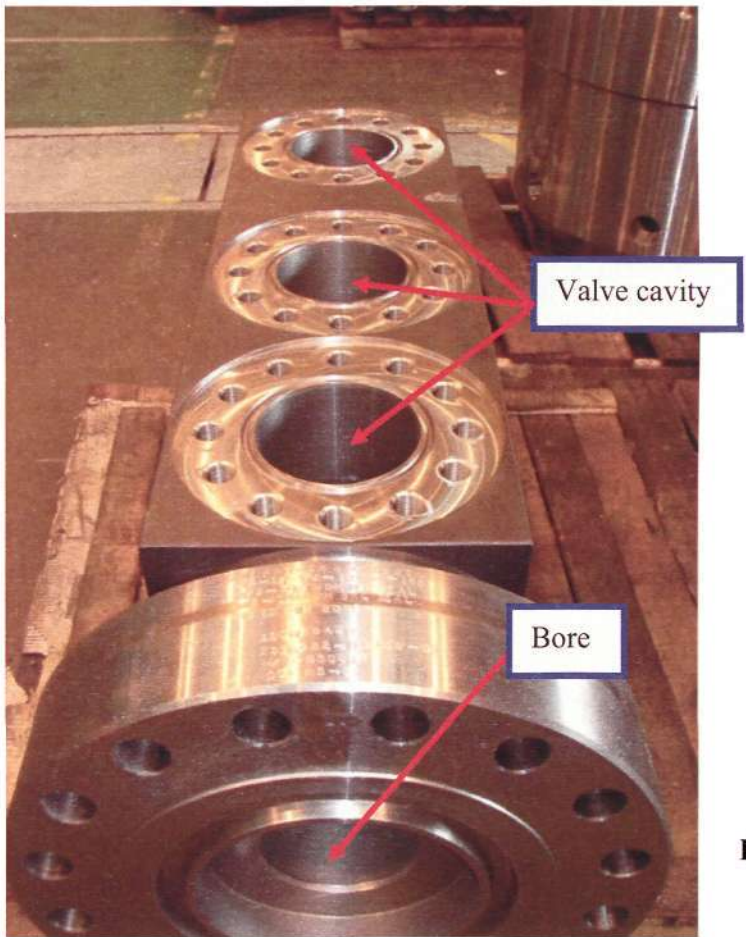


Figure 2.10 Block valve



### 3. Auto-FEA program

---

This chapter provides basic instructions for operating the Auto-FEA program: starting and stopping the program, understanding and using its GUI, etc. It also provides information on the FEA models: modeling and meshing details, types of elements used, etc.

*Note: Dimensions mentioned in this chapter will be in imperial units due to the program requirements of Cooper Cameron.*

#### 3.1 Introducing Auto-FEA

---

##### ▪ What is Auto-FEA?

Auto-FEA is a program developed using Visual Basic and ANSYS finite element analysis software. It allows a user, who is unfamiliar with any FEA package, to carry out finite element analysis on a particular model without being subjected to the technicalities of a FEA package. It enables the engineer to perform the following tasks:

- Modify dimensions of pre-defined models.
- Apply operating loads or other design performance conditions.
- Study physical responses such as stress levels and safety factors.
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible

The Auto-FEA program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands and documentation. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both.





### ▪ What is GUI?

The Graphical User Interface (GUI) is a simple, user-friendly and intuitive way to communicate with Auto-FEA. The GUI is a menu system that lets beginners and experienced users perform virtually all the software operations interactively. The GUI mode is also referred to as “interactive mode” in the documentation.

Each GUI interaction manipulates Visual Basic and ANSYS commands to perform the operation. The GUI allows the user to perform an analysis with little or no knowledge of the Visual Basic or ANSYS commands.

### ▪ Organization of the Auto-FEA program

The Auto-FEA program is organized into three basic levels.

- Launch level
- Processing level
- Post-process/Evaluation level

The *Launch level* acts as a gateway into the Auto-FEA program. The user is at the Launch level when he first enters the program. At this stage, the user chooses between different sub-programs to launch and carry out the processing.

At the *Processing level*, one of the sub-programs is activated. Each sub-program has a set of functions that perform a specific analysis task.

At the *Post-processing/Evaluation level*, results from the analysis are retrieved and evaluated. The user can create reports here using input parameter values and results obtained from the previous analysis.





---

## 3.2 Running the Auto-FEA program (Launch Level)

---

### ▪ Auto-FEA launcher / Start Menu

The launcher /Start Menu will give the user access to the sub-programs. The sub-programs are:

- Parallel Bores
- Perpendicular Out of plane Bores
- Perpendicular In plane Bores
- Valve Cavity and Perpendicular intersecting bores (through)
- Valve Cavity and Perpendicular intersecting bores (non-through)
- Valve Cavity & Offset Perpendicular intersecting bores
- Valve Cavity & Angled intersecting bores
- 2 Valve Cavities & Angled intersecting bores
- 2 Valve Cavities & Offset Perpendicular intersecting bores

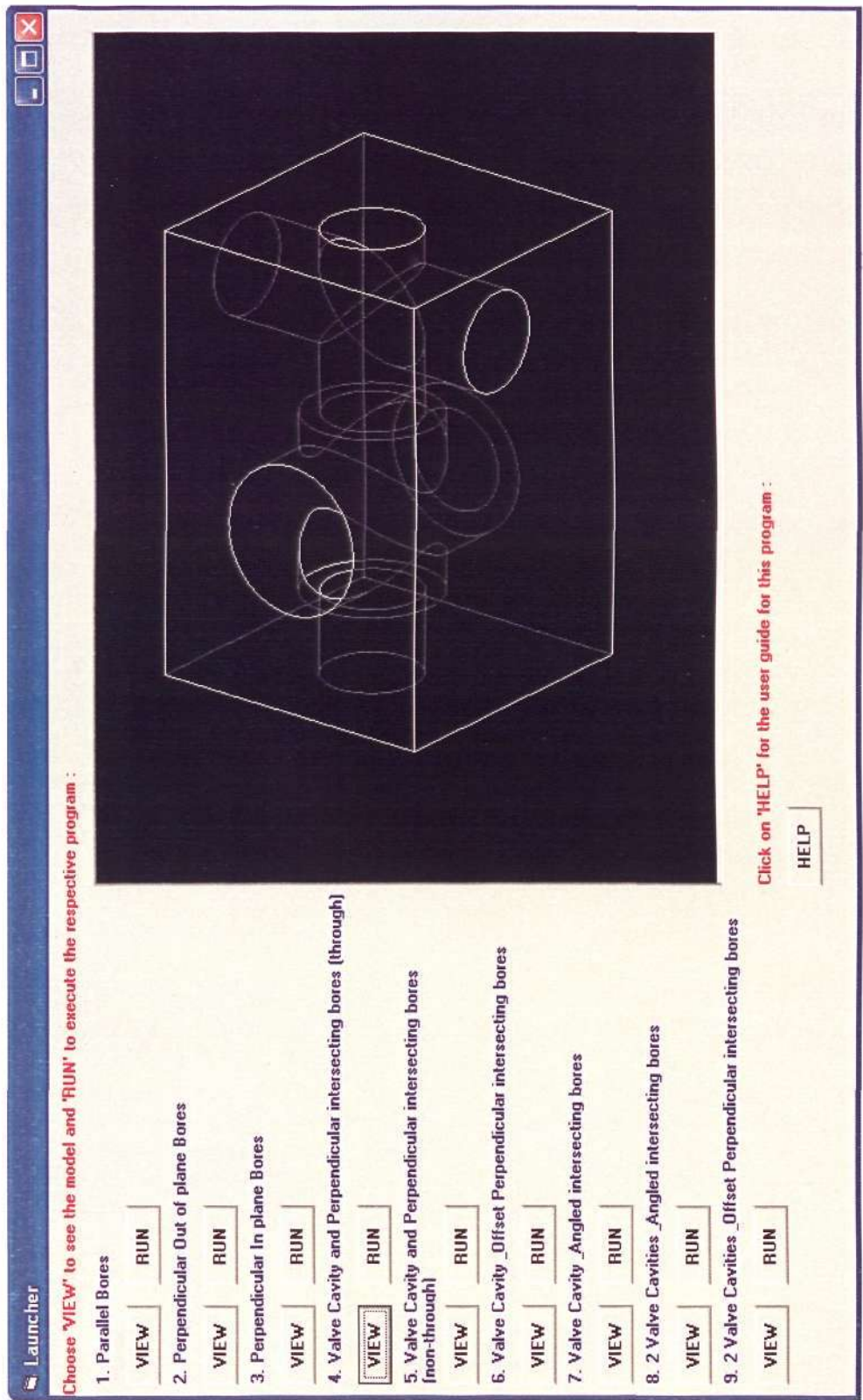


Figure 3.1 Launcher



### **Viewing reference models**

Each sub-program will have its own specific model to be analyzed. To view the different reference models under each sub-program, click on **VIEW** button. The picture box displays the different models and will vary accordingly to the sub-program selected.

### **Running the sub-programs**

Click **RUN** button once the model in which the analysis is to be carried out has been identified. More than one program can be launched at one time.

### **Launching the user manual /guide**

The user manual/guide is available as a PDF file which contains all the documentation for the Auto-FEA program. Click on the **HELP** button to access it. The user manual will appear in a separate window.

### **Exiting the Launcher**

The user has the option to minimize the Launcher window or close it once the sub-programs are activated. Exiting the Launcher will not affect the operation of the sub-programs.



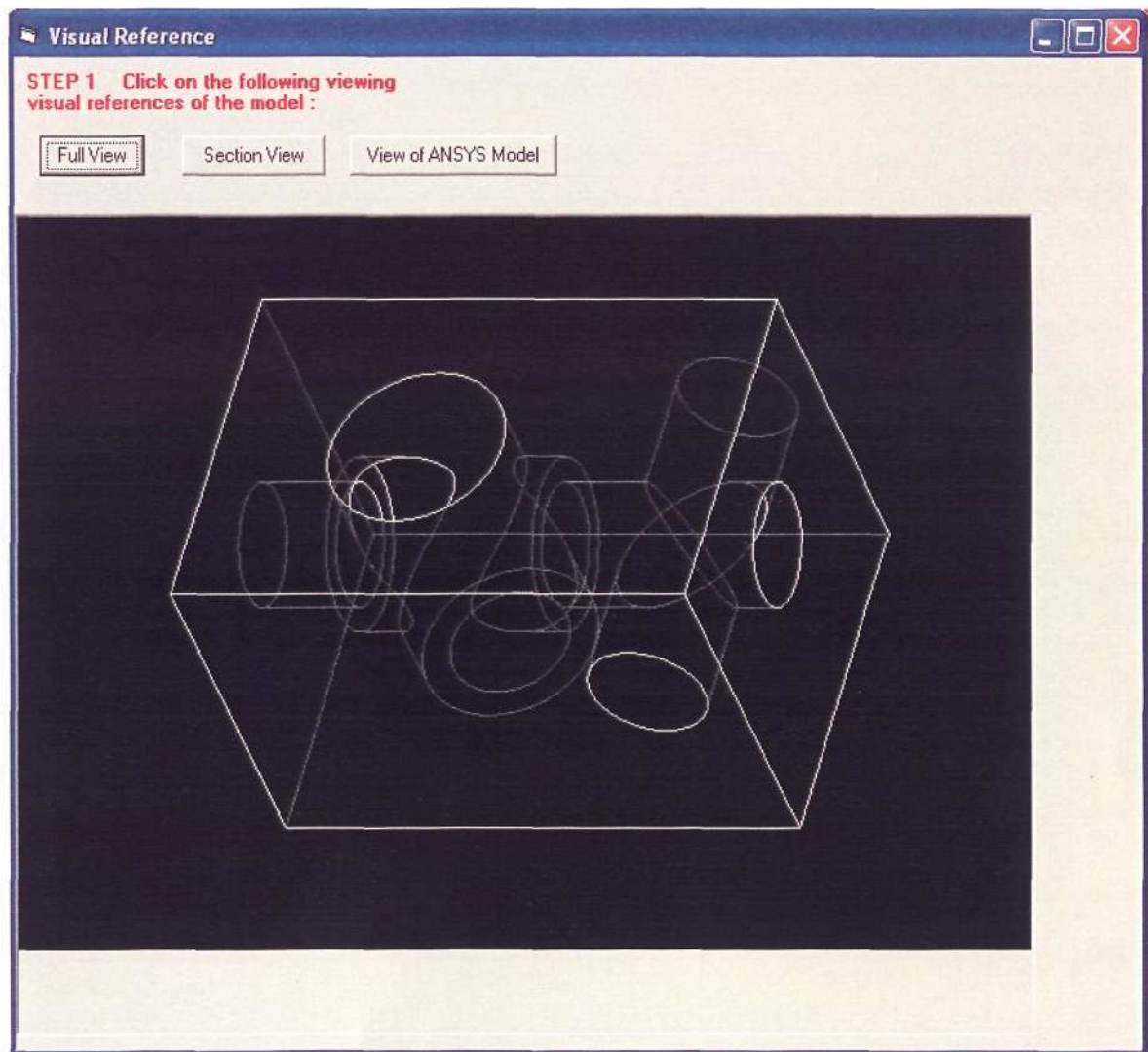






### Visual reference models

To view the specific model to be analyzed under the sub-program, click on **SHOW** button. Another menu will appear as shown in Figure 3.3.



**Figure 3.3 Visual Reference Menu**

There are 3 buttons available for different viewing choices:

- *Full View* displays the overall view of the model in 3D.
- *Section View* shows the 2D sectional views with the respective dimensions and labels.



- *View of ANSYS Model* illustrates the model ANSYS uses to carry out the analysis.

The picture on display will change according to the button clicked.

### **Choosing classification of cavity**

This is an optional feature of the program which saves the user the hassle of entering parameters related to a cavity repeatedly. The user will choose the classification of the cavity by selecting the Type, Pressure and Size of the cavity and clicking on the **SELECT** button shown in Figure 3.4.

**Choose Classification of cavity (Optional)**

Type : FL ▼

Pressure : 3K ▼

Size : 2-1/16" ▼

**SELECT**

**Figure 3.4** Choosing classification of cavity

The options available for Type are FL and FLS.

The options available for Pressure are 2K, 3K, 5K, 10K and 15K.

The options available for Size are 1-13/16", 2-1/16", 2-9/16", 3-1/16", 3-1/8", 4-1/16", 4-1/8", 5-1/8" and 6-3/8".

The geometric parameters defining the different classifications of cavities are stored in a MSACCESS database. Once the **SELECT** button is clicked, the program will access this database, search for the parameters pertaining to the classification chosen, retrieve the values, return and display them in the textboxes for the respective parameters. This is illustrated in Figure 3.5. To help the user identify the geometric parameters defining a cavity, the textboxes containing these parameters will have their backgrounds turned to black once the new values are displayed.



*Note: It is the responsibility of the user to confirm the geometric properties values are correct. The user can overwrite the default values as required to correct any discrepancies.*

Geometric Properties	
Length L :	1 in
Length W :	0.6 in
Length H :	0.8 in
Length A :	0.3 in
Cavity Bore Diameter C :	0.35 in
Length D :	0.5 in
Drill Point Diameter E :	0.2 in
Drill Point Angle F :	2.8 °
Length P :	0.2 in
Flow Bore Diameter Q :	0.15 in
Length R :	0.4 in
Seat Pocket Diameter T :	0.3 in
Length V :	0.4 in
Cross Bore Diameter Y :	0.15 in

Geometric Properties	
Length L :	1 in
Length W :	0.6 in
Length H :	0.8 in
Length A :	0.3 in
Cavity Bore Diameter C :	4.51 in
Length D :	7.891 in
Drill Point Diameter E :	2.81 in
Drill Point Angle F :	115 °
Length P :	0.2 in
Flow Bore Diameter Q :	0.15 in
Length R :	6.879 in
Seat Pocket Diameter T :	3.55 in
Length V :	0.4 in
Cross Bore Diameter Y :	0.15 in

Figure 3.5     Textboxes defining classification of cavities





**Entering parameter values**

All parameters to be defined for carrying out the analysis are categorized into 3 sections, Geometric Properties (Figure 3.5), Material Properties (Figure 3.6) and Loads (Figure 3.7).

**Material Properties**  
Young's Modulus :    
Poisson's Ratio :   
Thermal Conductivity :    
Coefficeint of thermal expansion :    
Yield Strength :

**Figure 3.6** Section displaying material properties

**Loads**  
External Pressure :    
Pressure in Cavity :    
Pressure in Flow Bore :    
Pressure in Cross Bore :    
Cavity temperature :    
Flow Bore temperature :    
Cross Bore temperature :    
External temperature :    
Reference temperature :

**Figure 3.7** Section displaying Loads

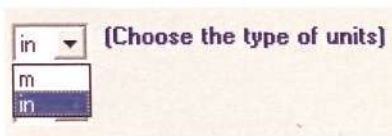
Default values in the textboxes will appear initially and these can be changed according to the user's specifications. Without exiting the main menu, the textboxes will take on the



previously entered values. If the main menu is closed and reopened/activated, the textboxes will take on the default values again.

### **Choosing parameter units**

Metric or Imperial Units are available for selection. Depending on the type of units used, the parameter values will take on different physical values.



**Figure 3.8** Choice of choosing parameter units

The user will make the selection of the type of units in the first dropdown box (Figure 3.8). The units for the other parameters will follow this selection.

### **Running the FEA analysis**

Click the **RUN** button to carry out the analysis after all parameter values have been entered. This will activate the ANSYS program in the background and carry out one run using the values entered. While ANSYS is running in the background, the user can look at the ANSYS output window (Figure 3.9) to monitor the progress.



```

ANSYS 8.1 Output Window
00265231      VERSION=INTEL NT      RELEASE= 8.1      UP20040329
CURRENT JOBNAME=file 23:53:42 OCT 25, 2005 CP=      0.781

/SHOW SET WITH DRIVER NAME= WIN32 , RASTER MODE, GRAPHIC PLANES =
RUN SETUP PROCEDURE FROM FILE= C:\Program Files\Ansys Inc\v81\ANSYS\ans
ans
/INPUT FILE= menust.tnp LINE=      0
/INPUT FILE= C:\Program Files\Ansys Inc\v81\ANSYS\apdl\start81.ans L
0
ACTIVATING THE GRAPHICAL USER INTERFACE <GUI>. PLEASE WAIT...
Creating the GUI panes
Creating the GUI menubar
Creating the GUI status
Creating the GUI Graphics Context
Creating the Application Context
Creating the GUI toolbar
Creating the GUI toolbar - "ANSYS Standard Toolbar"
Creating the GUI toolbar - "ANSYS Main Menu"
Creating the GUI toolbar - "ANSYS Toolbar"
Creating the GUI toolbar - "ANSYS Graphical View"

CUTTING PLANE SET TO THE WORKING PLANE
Loading ANSYS Help System
Welcome to ANSYS

PRODUCE NODAL PLOT IN DSYS= 0
TURN OFF WORKING PLANE DISPLAY
  
```

Figure 3.9 Output Window

The Output Window receives all text output from the ANSYS program - command responses, notes, warnings, errors, and any other messages. It is usually minimized in the taskbar but it can be brought up when necessary.

**Warning:** Closing the Output Window in the midst of the run, will terminate the ANSYS session and is not advisable.

Once one complete run is carried out, the ANSYS program running in the background will exit automatically and the Output Window will disappear. The user can then change the parameter values and re-run the analysis or move on to other options. After the end of each run, there will be 2 forms of outputs in the respective folder:

1. graphical plots saved as jpeg files, file000.jpg and file001.jpg
2. Nodal results saved in a Notepad format named listview.txt.





### 3.4 Running the Auto-FEA program (Post-processing Level)

This section details the operation guide for the different sub-programs during the post-processing level.

#### Retrieving results

Click the **GET** button to view results of the ANSYS run. Graphical displays of the model and its contour plots of **safety factors** will appear (Figure 3.10). Safety factor is defined as the yield strength divided by the Von Mises stress. The user has the option to save this graphical result by clicking on the **SAVE** button on the top right corner.

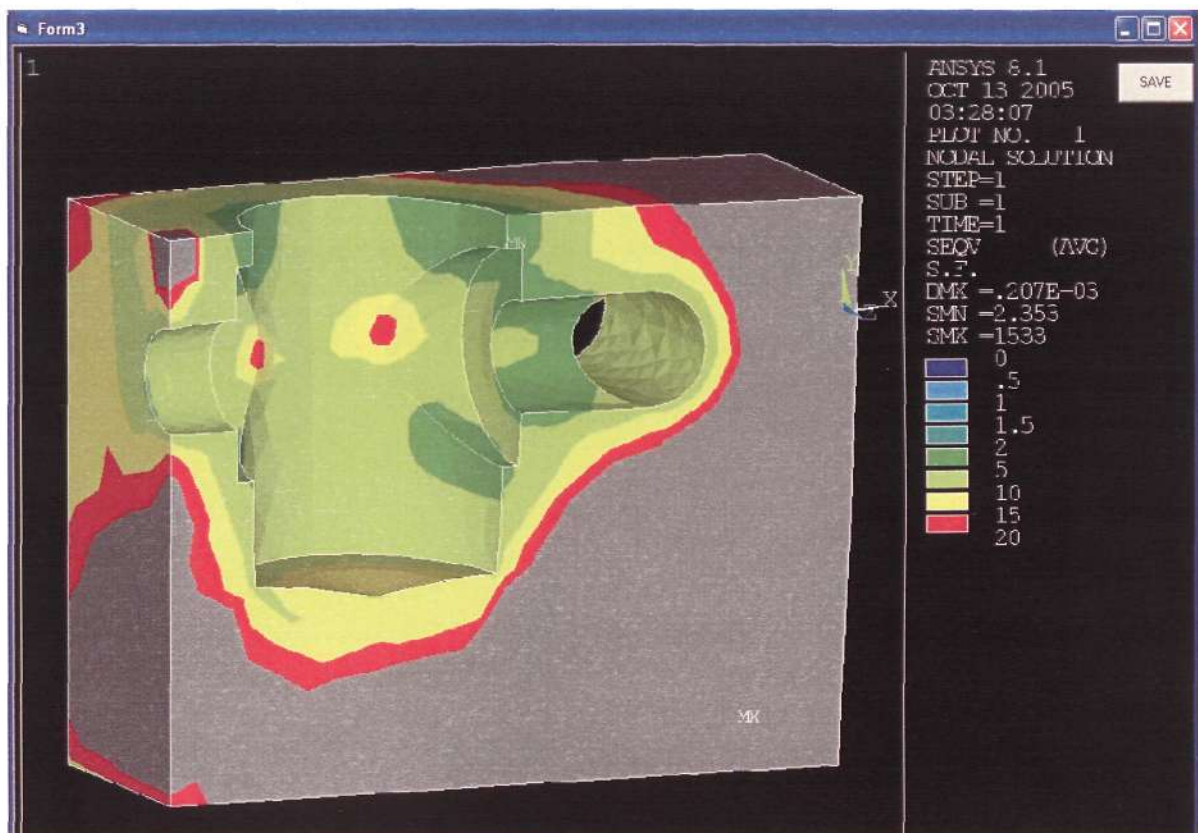


Figure 3.10 Graphical plots

### Abbreviations used in graphical plots

SEQV – Von Mises Stress

S.F – Safety factor

DMX – Maximum deformation

SMN – Minimum stress

SMX – Maximum stress

At the same time, on the main menu, the list view box (Figure 3.11) will list out all the nodal numbers, nodal values and safety factors. This is done by importing the values from listview.txt into the list view box. The **GET** button will always display the results of the previous analysis/ANSYS run, regardless of exiting the main menu.

[illegible]

Node no.	S1	S2	S3	SINT	Safety Factor
PRINT	S	NODAL	SOLUTION	PER	NODE
1	95.06839	103.6786	22.87716	18.43982	21.06300
2	47.05659	34.15445	10.13240	12.91283	13.55806
3	2907.602	2798.531	304.8425	275.9147	300.3422
4	3287.296	6534.687	2257.977	1338.553	1532.856
5	46.51769	29.44327	11.02058	14.44208	15.64280
6	6157.940	4029.408	215.0144	222.7936	224.9117
7	3668.933	289.7589	64.01384	65.15055	71.26354
8	31.41458	46.61188	97.79253	46.28210	53.42820
9	20.87464	1471.009	12.84720	7.952728	9.104996
10	9.969535	825.1201	13.47588	5.730258	6.587467
11	7.726024	1773.986	13.43638	4.905387	5.599435
12	30.74451	821.7792	12.88460	9.079506	10.23149
13	8.355713	52.55601	14.09841	5.246353	6.054791

**Figure 3.11** Listview showing results



### **Abbreviations used in list view**

SINT – Stress Intensity

S1 – 1<sup>st</sup> Principal Stress

S2 – 2<sup>nd</sup> Principal Stress

S3 – 3<sup>rd</sup> Principal Stress

### **Saving results as EXCEL file**

All the values in the list view box (Figure 3.11) can be saved as a spreadsheet in MS EXCEL format. This is an optional step. Clicking on the **SAVE** button on the main menu will activate MS EXCEL and import the values from the list view box into the spreadsheet. The user has a choice to format the cells in MS EXCEL, save the file and exit MS EXCEL.

### **Creating a report**

This is an optional step for users who wish to create a report of the analysis using a MS WORD template. When the **REPORT** button is clicked, another separate report menu will appear (Figure 3.12). The parameters and its values are the same as those entered on the sub-program main menu.





Report

<p><b>Geometric Properties</b></p> <p>Length L : <input type="text" value="1"/> <input type="button" value="in"/></p> <p>Length W : <input type="text" value="0.6"/> <input type="button" value="in"/></p> <p>Length H : <input type="text" value="0.8"/> <input type="button" value="in"/></p> <p>Length A : <input type="text" value="0.3"/> <input type="button" value="in"/></p> <p>Cavity Bore Diameter C : <input type="text" value="0.35"/> <input type="button" value="in"/></p> <p>Length D : <input type="text" value="0.5"/> <input type="button" value="in"/></p> <p>Drill Point Diameter E : <input type="text" value="0.2"/> <input type="button" value="in"/></p> <p>Drill Point Angle F : <input type="text" value="2.8"/> <input type="button" value="rad"/></p> <p>Length P : <input type="text" value="0.2"/> <input type="button" value="in"/></p> <p>Flow Bore Diameter Q : <input type="text" value="0.15"/> <input type="button" value="in"/></p> <p>Length R : <input type="text" value="0.4"/> <input type="button" value="in"/></p> <p>Seat Pocket Diameter T : <input type="text" value="0.3"/> <input type="button" value="in"/></p> <p>Length V : <input type="text" value="0.4"/> <input type="button" value="in"/></p> <p>Cross Bore Diameter Y : <input type="text" value="0.15"/> <input type="button" value="in"/></p> <p><b>Material Properties</b></p> <p>Young's Modulus : <input type="text" value="29000000"/> <input type="button" value="Psi"/></p> <p>Poisson's Ratio : <input type="text" value="0.3"/></p> <p>Thermal Conductivity : <input type="text" value="296"/> <input type="button" value="BTU-in/hr-ft²-F"/></p> <p>Coefficient of thermal expansion : <input type="text" value="6.3"/> <input type="button" value="µin/in-F"/></p> <p>Yield Strength : <input type="text" value="65000"/> <input type="button" value="Psi"/></p>	<p><b>Loads</b></p> <p>External Pressure : <input type="text" value="15"/> <input type="button" value="Psi"/></p> <p>Pressure in Cavity : <input type="text" value="5000"/> <input type="button" value="Psi"/></p> <p>Pressure in Flow Bore : <input type="text" value="5000"/> <input type="button" value="Psi"/></p> <p>Pressure in Cross Bore : <input type="text" value="5000"/> <input type="button" value="Psi"/></p> <p>Cavity temperature : <input type="text" value="453"/> <input type="button" value="°F"/></p> <p>Flow Bore temperature : <input type="text" value="453"/> <input type="button" value="°F"/></p> <p>Cross Bore temperature : <input type="text" value="453"/> <input type="button" value="°F"/></p> <p>External temperature : <input type="text" value="453"/> <input type="button" value="°F"/></p> <p>Reference temperature : <input type="text" value="453"/> <input type="button" value="°F"/></p> <p>Subject : <input type="text"/></p> <p>File No. : <input type="text"/></p> <p>Prepared by : <input type="text"/> Date : <input type="text"/></p> <p>Checked by : <input type="text"/> Date : <input type="text"/></p> <p>Comment : <div style="border: 1px solid black; height: 100px; width: 100%;"></div></p> <p style="color: red; text-align: center;">Click 'GENERATE' to create a WORD report.</p> <p style="text-align: center;"> <input type="button" value="Generate"/> </p>
---	--

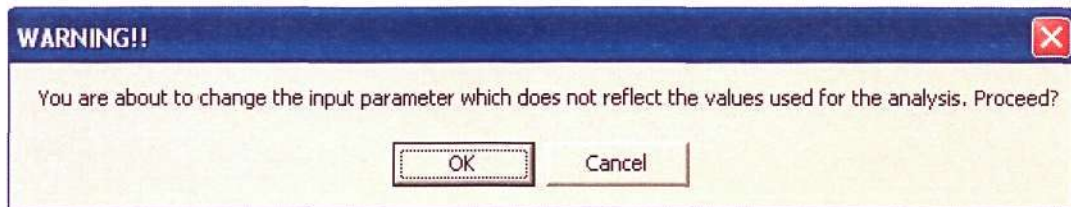
**Figure 3.12 Report Menu**

There are textboxes for entering the Subject, File number, Prepared By, Checked By, respective dates and comments.

An additional feature has been added to this report menu so that data entered will not be lost if the user 'accidentally' closes the report menu. By clicking on the **REPORT** button on the main menu again, the report form will reappear and the previous data entered intact.

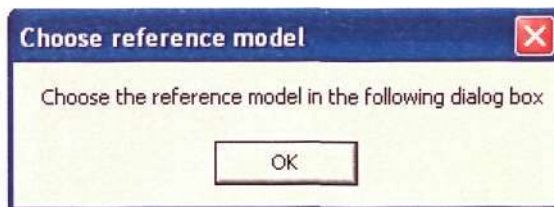


*Note: If the user changes the values in the textboxes, a warning message will appear.*



**Figure 3.13** Warning message

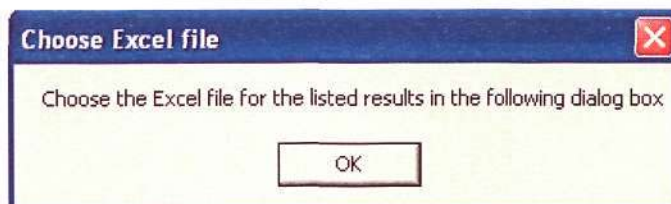
Once all data are entered, the user will click on the **GENERATE** button which opens the MICROSOFT WORD template in the respective folders. The user will be prompted to choose the visual reference model (Figure 3.14), graphical plots (Figure 3.15) and MS EXCEL file (Figure 3.16) to be shown on the MS WORD document.



**Figure 3.14** Prompt for choosing reference model



**Figure 3.15** Prompt for choosing graphical plots



**Figure 3.16** Prompt for choosing Excel file



All the input parameter values and additional data will be imported into bookmarks in the WORD template.

- **Linking Visual Basic to ANSYS**

APDL stands for ANSYS Parametric Design Language, a scripting language that is used to build the models in terms of parameters (variables) and automate common analysis tasks.

Visual Basic will write all the APDL to a Notepad file called **testfile.txt** in C:\ drive. The full path is "C:\testfile.txt". This testfile.txt gets overwritten every time the user clicks on the **RUN** button in the main menu. It contains the APDL for the particular sub-program that is activated and the input parameter values that are used for the ANSYS run. Users can make use of the preprogrammed solid models and APDL and make their own modifications to the APDL code. This can be done by running the sub-program once and accessing "testfile.txt". Experienced ANSYS users would then be able to understand the APDL used in testfile.txt and manipulate the code to suit their own use. This saves the user time to recreate the solid models themselves and the rest of the APDL sequence.





---

### 3.5 ANSYS Analysis Methodology

---

- **General**

Combined thermal and structural analyses can be carried out on all the models in the sub-programs. Similarly, by keeping certain parameters constant, individual thermal stress analysis or structural analysis can be performed. The sequentially-coupled physics analysis approach using physics environments is used to solve the thermal structural analyses.

- **Thermal Structural Analysis using Physics Environments**

The basic procedure for the physics environment approach is shown below:

1. Define the thermal problem.
2. Write the thermal physics file.
3. Clear boundary conditions and options.
4. Define the structural problem.
5. Write the structural physics file.
6. Read the thermal physics file.
7. Solve and post process the thermal problem.
8. Read the structural physics file.
9. Read the temperatures from the thermal results file.
10. Solve and post process the physics file.



## ▪ Element Types

Element types must maintain a consistent base geometry across multiphysics environments. For 2-D analysis, Plane35 and Plane2 elements are compatible element types across physics environments. For 3-D analysis, Solid87 and Solid92 elements are compatible element types across physics environments.

### Plane35 and Plane2 (2 dimensional elements)

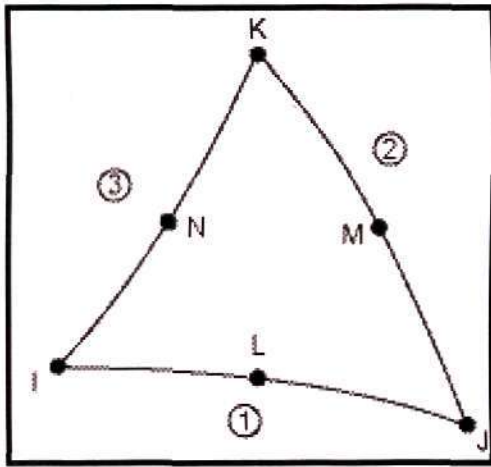


Figure 3.17 Plane35 and Plane2 elements

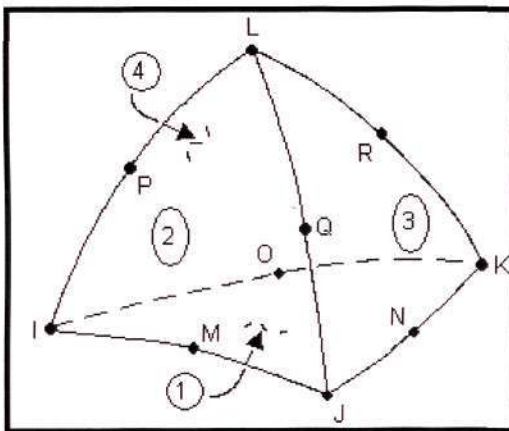
**Plane35** (Figure 3.17) is a 2-D 6-node triangular thermal element with one degree of freedom, temperature, at each node. The triangular shape makes it well suited to model irregular meshes (such as curves). The 6-node thermal element is applicable to a 2-D, steady-state or transient thermal analysis. The output data for this element include nodal temperatures and element data, such as thermal gradients and thermal fluxes. Since the models containing this element are also to be analyzed structurally, the element should be replaced by an equivalent structural element (plane2) for the coupled field analysis.

**Plane2** (Figure 3.17) is a 2-D 6-node triangular structural element. It is defined by six nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element has a quadratic displacement behavior and is well suited to model



irregular meshes (such as curves). It can be used as a plane element (plane stress or plane strain) or as an axisymmetric element. The output data include nodal displacements and element data, such as directional stresses and principal stresses.

### **Solid87 and Solid92 (3 dimensional elements)**



**Figure 3.18 Solid87 and Solid92 elements**

**Solid87** (Figure 3.18) is a 3-D 10-node tetrahedral thermal solid element with one degree of freedom, temperature, at each node. Its shape makes it well suited to model irregular meshes (such as bends and curves). It is applicable to a 3-D, steady-state or transient thermal analysis. Since the models containing this element are also to be analyzed structurally, the element should be replaced by an equivalent structural element (solid92) for the coupled field analysis.

**Solid92** (Figure 3.18) is a 3-D 10-node tetrahedral structural solid element. It is defined by ten nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. The element has a quadratic displacement behavior and is well suited to model irregular meshes (such as bends and curves). This element may be used to analyze large-deflection, large-strain, plasticity, and creep problems.





### ▪ Determination of mesh size and density

Auto-FEA is designed to eliminate human intervention with the processing as much as possible. In the Auto-FEA program, the mesh sizes were not fixed in absolute numbers. This would not be viable given that it is a parametric program where the user can enter and experiment with different values.

Instead, different possibilities and details on mesh density and sizing were factored into the program in this way:

- For each sub-program, it extracts the smallest feature (wall thickness or diameter for example). The smallest feature is the most critical part (high stresses expected) of the model which requires much finer mesh density.
- The program calculates the size of this feature from the parameters which the user has entered.
- The size of the feature is divided by a pre-determined ratio to give the mesh size, which ANSYS will use to mesh the model. The pre-determined ratio has been proven and obtained from convergence tests.
- The mesh area/volume around the critical feature is refined with even smaller mesh sizes.

This method will ensure that the mesh is sufficiently fine in the critical region where high stresses are expected.

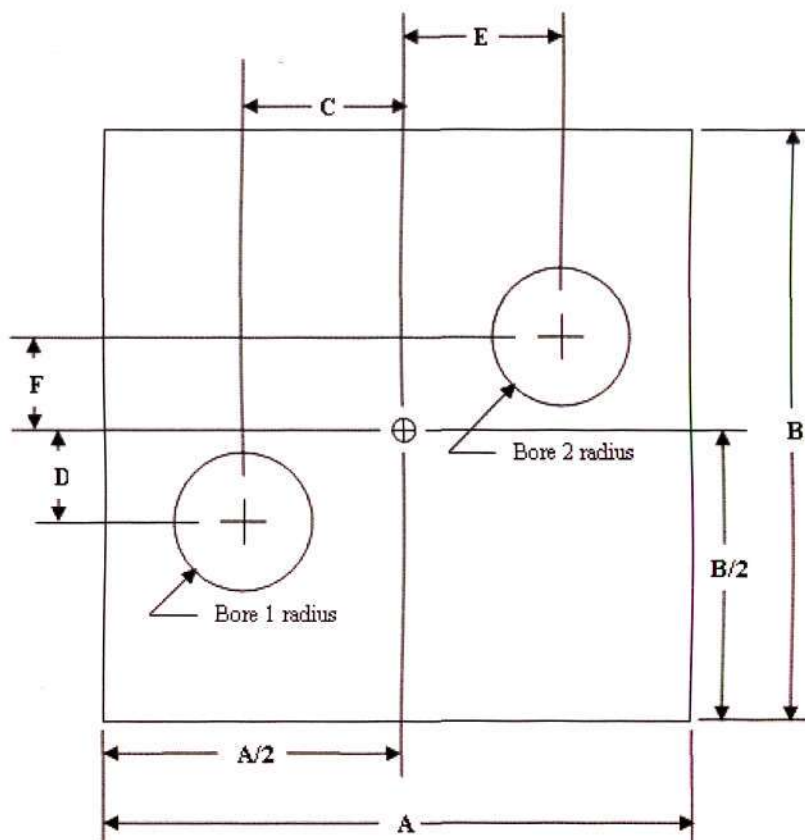


### 3.6 ANSYS details of sub-programs

This section explains the ANSYS model used in each sub-program and its analysis features.

#### ▪ Parallel Bores

##### Solid Modeling



**Figure 3.19** Reference model for parallel bores

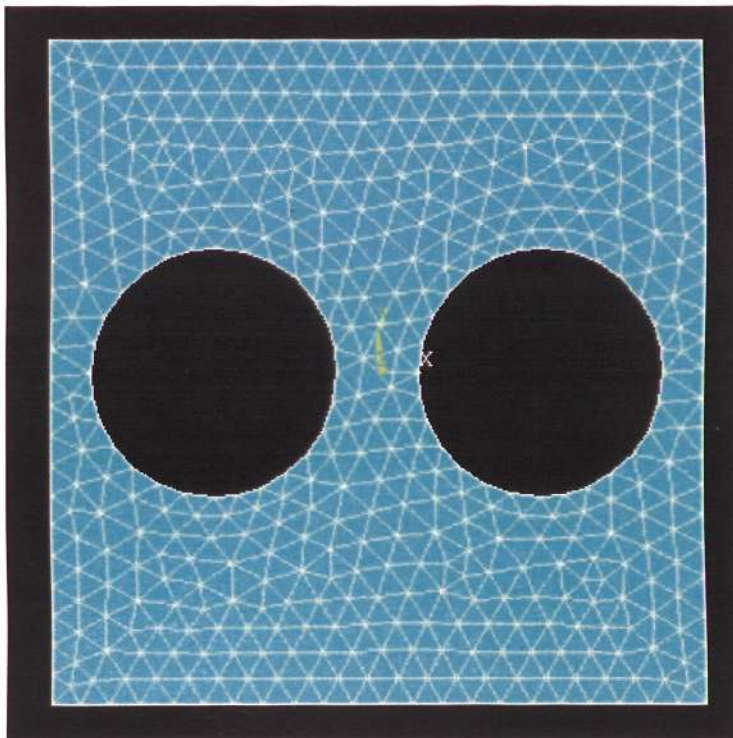
This model simulates a block valve with through bores. The default model has two bores although one bore can be specified by setting the radius of one of the bores to zero.



Block valves come in varying lengths and can be customized according to customer requirements. The length of the block can be more than twice its width.

### **Meshing**

Plane35 and Plane2 (2 dimensional) elements are used to mesh the model, for the thermal and structural analysis respectively.



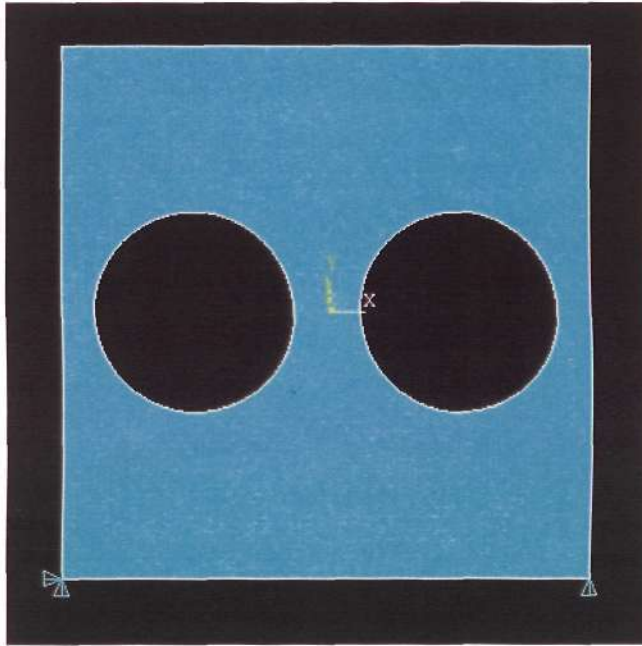
**Figure 3.20 Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). Finer meshing, higher mesh density, is deployed in the region of high stresses to simulate the stress gradients correctly.





### **Boundary conditions and constraints**



**Figure 3.21 Boundary conditions on model**

One corner at the base of the model is constrained along the UX and UY directions. The other corner at the base is constrained along the UY direction. Rigid body motion is prevented.

### **Assumptions**

- The plane strain approach is used to solve and predict the stress distribution of the model. Since the cross-sections are uniform, the block can be visualized as having many infinitely thin planes cut through the length of the block valve, with each plane a plane-strain problem. The strain along the length of the block is zero, thus each plane can be analyzed as a plane-strain problem.
- A linear FEA solution is assumed with an elastic material response for the model.



- The effect of bolts or drill holes on the exterior of the block valves on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.

Although the block valve is free to expand on all sides, stress analysis results are still affected by temperature.

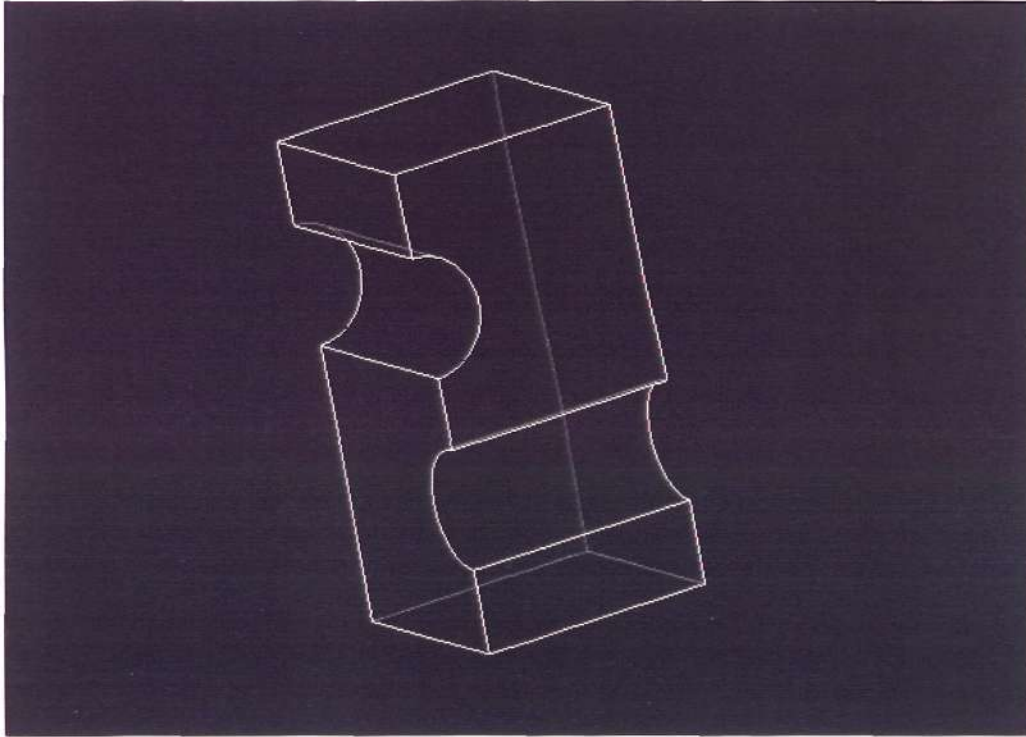
When the block valve is used in an actual field location, it is subjected to external environmental temperatures (can be more than 40°C in desert conditions or less than 0°C in cold climates). Oil, which passes through the bores and cavities, will be at much higher temperatures. There is a constant temperature gradient across the thicknesses of the blocks into the bores/cavities. The temperature gradient results in thermal strains and thermal stresses. There is no temperature equilibrium between the internal and external features of the block as oil is constantly passing through and external climate temperature remains constant.

The above explanation is also applied to the same assumption (the block valve is free to expand on all sides, meaning there are no external support constraints on the block.) mentioned in the other sub-programs in this chapter.



### ▪ Perpendicular Out of plane Bores

#### Solid Modeling



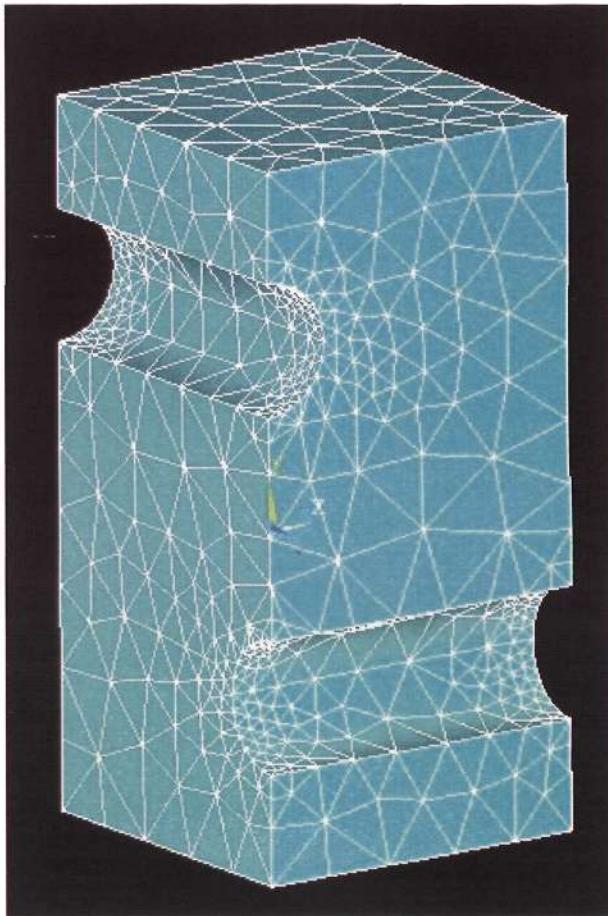
**Figure 3.22 Visual Reference for Perpendicular Out of Plane Bores**

This model simulates a block valve with two perpendicular out of plane bores. Since there are two planes of symmetry, a quarter model of the block valve was used with specified symmetrical boundary conditions on the cut faces.

#### Meshing

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.



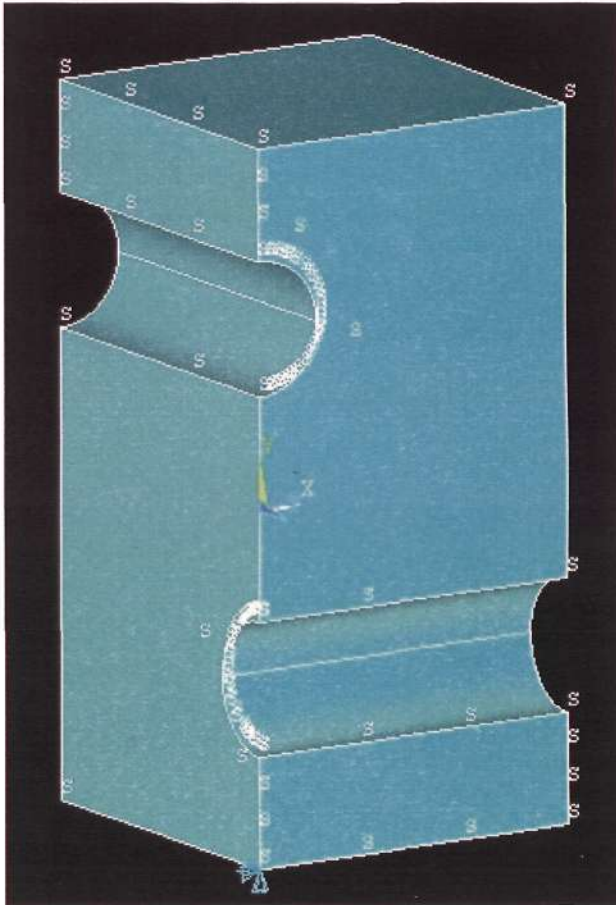


**Figure 3.23** Meshing of model

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). Finer meshing, higher mesh density, is deployed in the region of high stresses to simulate the stress gradients correctly.



### **Boundary conditions and constraints**



**Figure 3.24 Boundary conditions on model**

A single point constraint is used and constrained in all degrees of freedom. Rigid body motion is prevented. Surfaces defined with 'S' on its perimeter lie on planes of symmetry with symmetrical boundary conditions applied.

### **Assumptions**

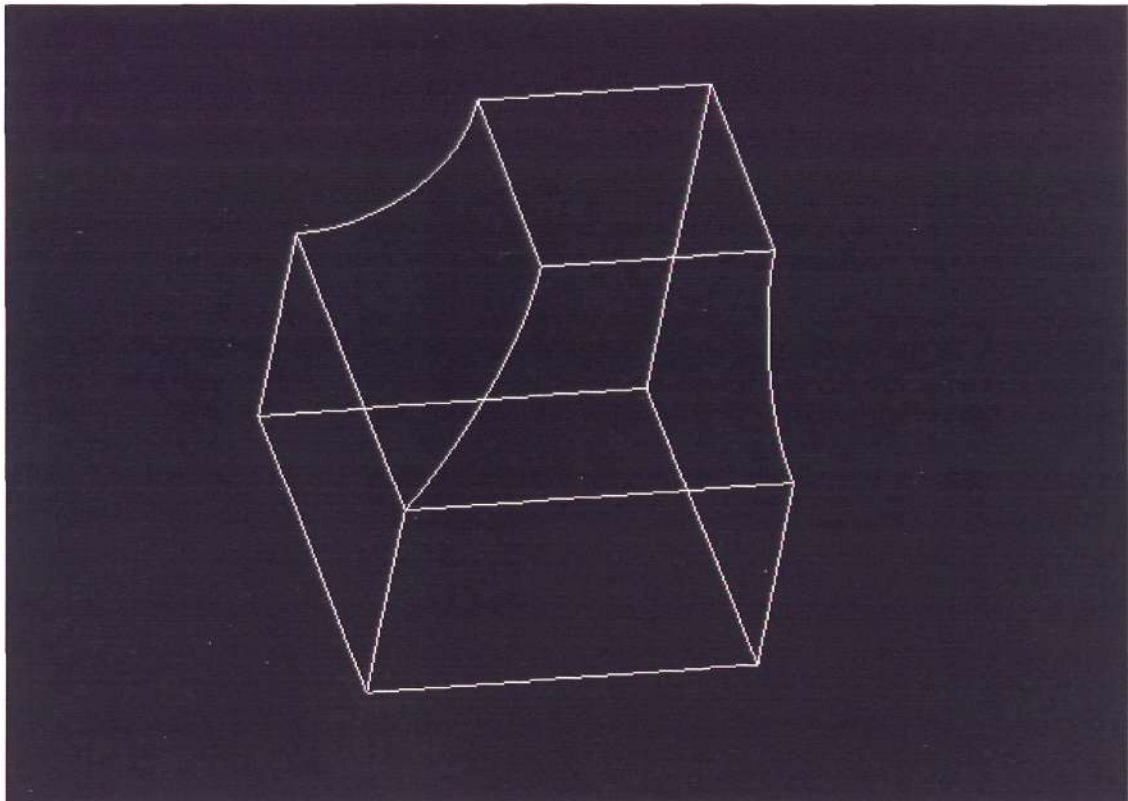
- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.



- The effect of bolts or drill holes on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.

#### ▪ Perpendicular In plane Bores

#### Solid Modeling



**Figure 3.25 Reference model for Perpendicular Out of Plane Bores**

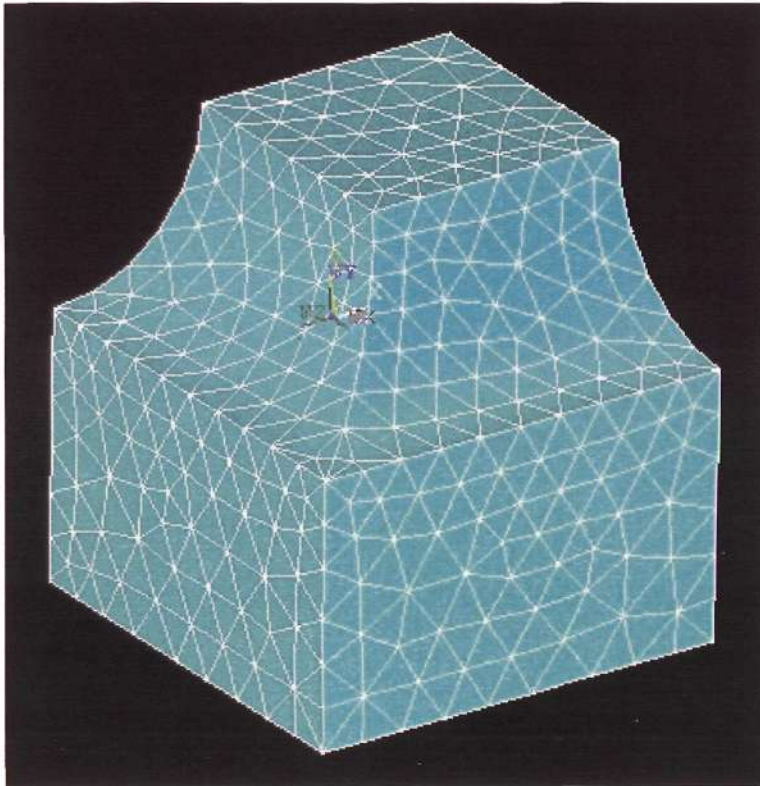
This model simulates a block valve with two perpendicular in plane bores. Since there are three planes of symmetry, one-eighth of the block valve was modeled with specified symmetrical boundary conditions on the cut faces.





## **Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.

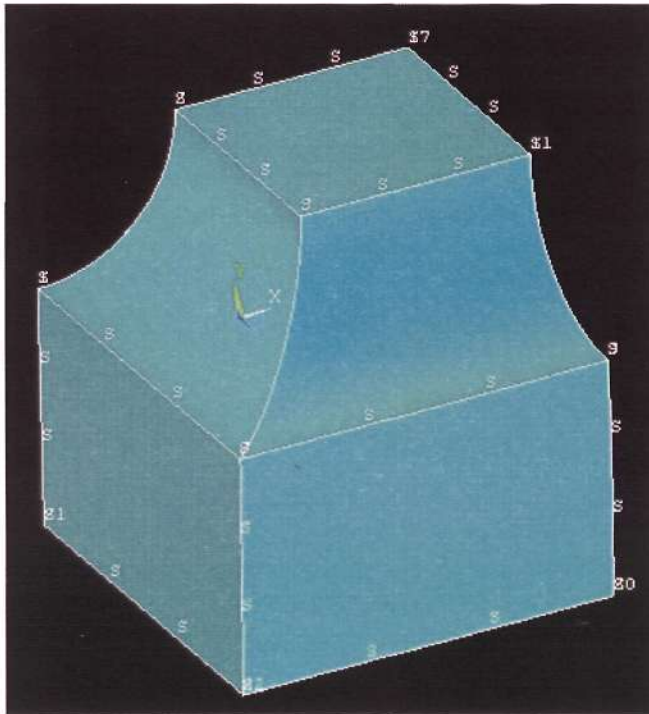


**Figure 3.26    Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). Finer meshing, higher mesh density, is deployed in the region of high stresses to simulate the stress gradients correctly.



### **Boundary conditions and constraints**



**Figure 3.27 Boundary conditions on model**

Rigid body motion is prevented. Surfaces defined with 'S' on its perimeter lie on planes of symmetry with symmetrical boundary conditions applied.

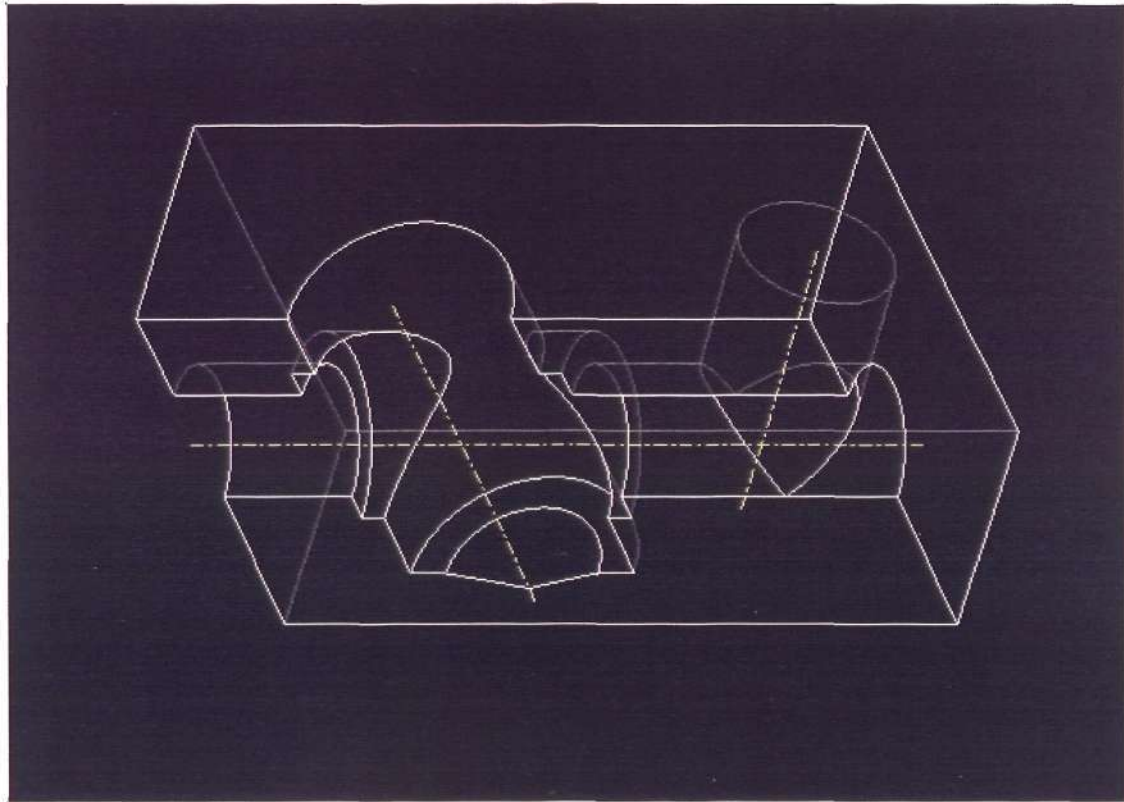
### **Assumptions**

- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.
- The effect of bolts or drill holes on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.



▪ **Valve Cavity and Perpendicular intersecting bores (through)**

**Solid Modeling**



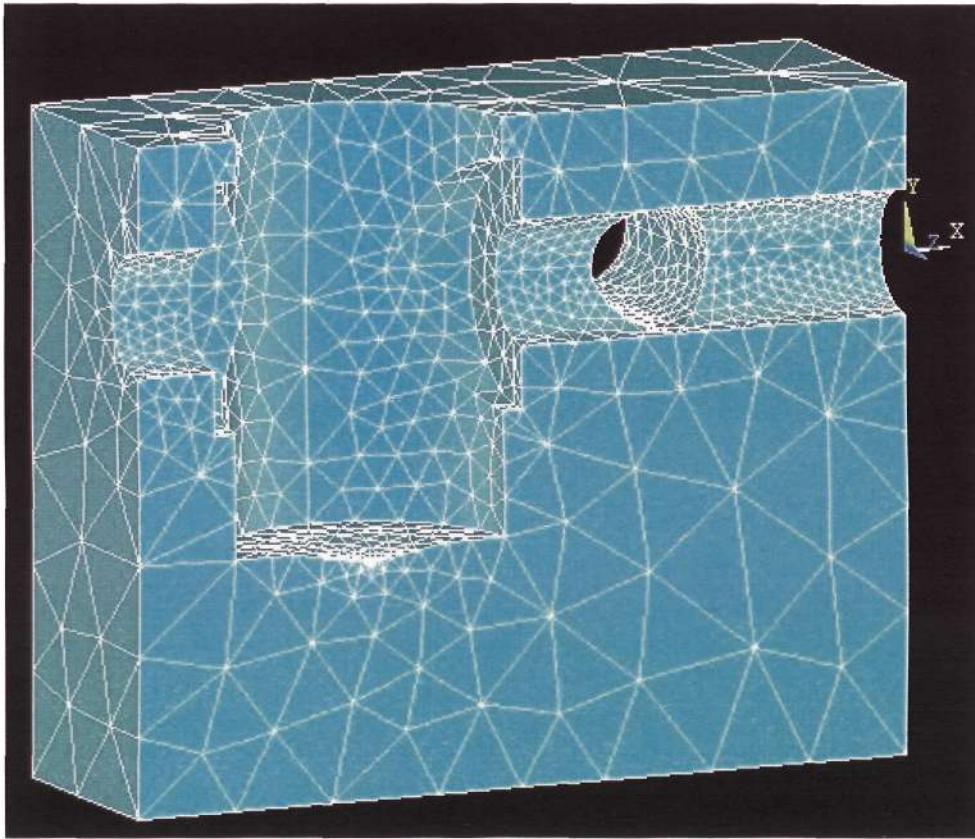
**Figure 3.28 Reference model for Valve Cavity & Perpendicular Intersecting Bores (Through)**

This model simulates a block valve with a valve cavity and perpendicular intersecting through bores. Since there is one plane of symmetry, half of the block valve was modeled with specified symmetrical boundary conditions on the cut face.

**Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.





**Figure 3.29** Meshing of model

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). Finer meshing, higher mesh density, is deployed in the region of high stresses to simulate the stress gradients correctly.

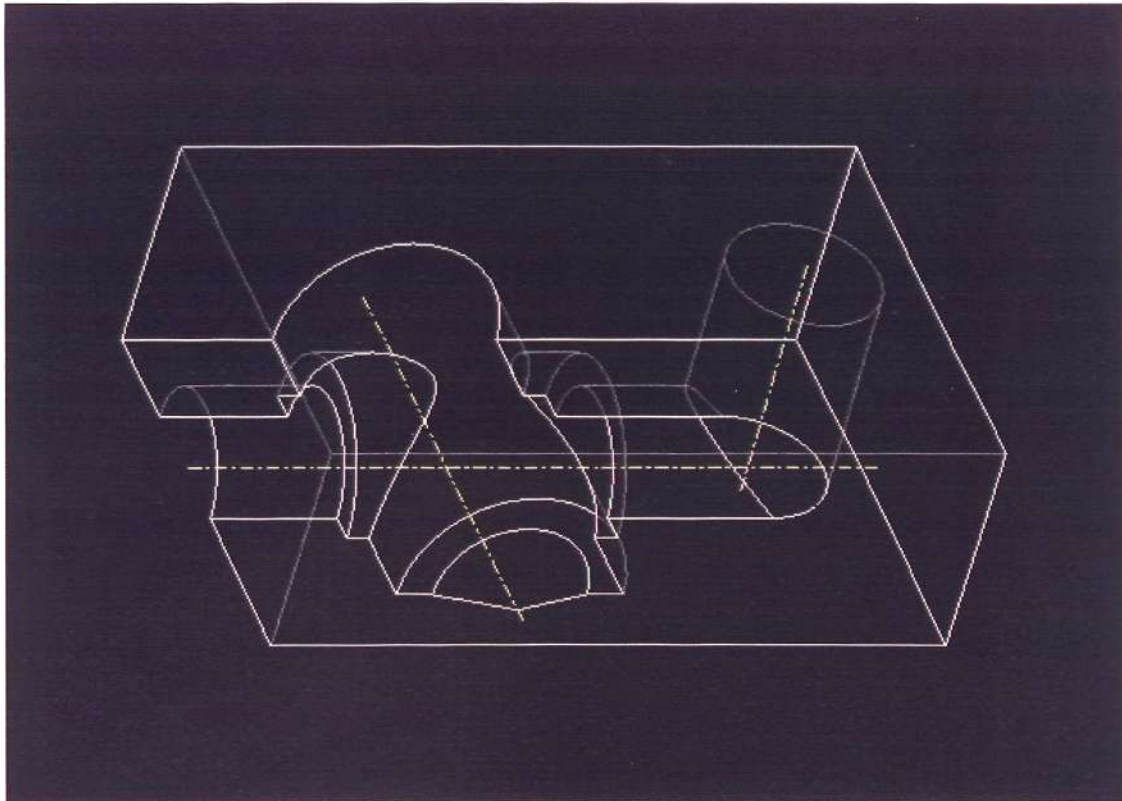




- The effect of bolts or drill holes on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.

▪ **Valve Cavity and Perpendicular intersecting bores (non-through)**

**Solid Modeling**



**Figure 3.31 Reference model for Valve Cavity & Perpendicular Intersecting Bores (Non-through)**

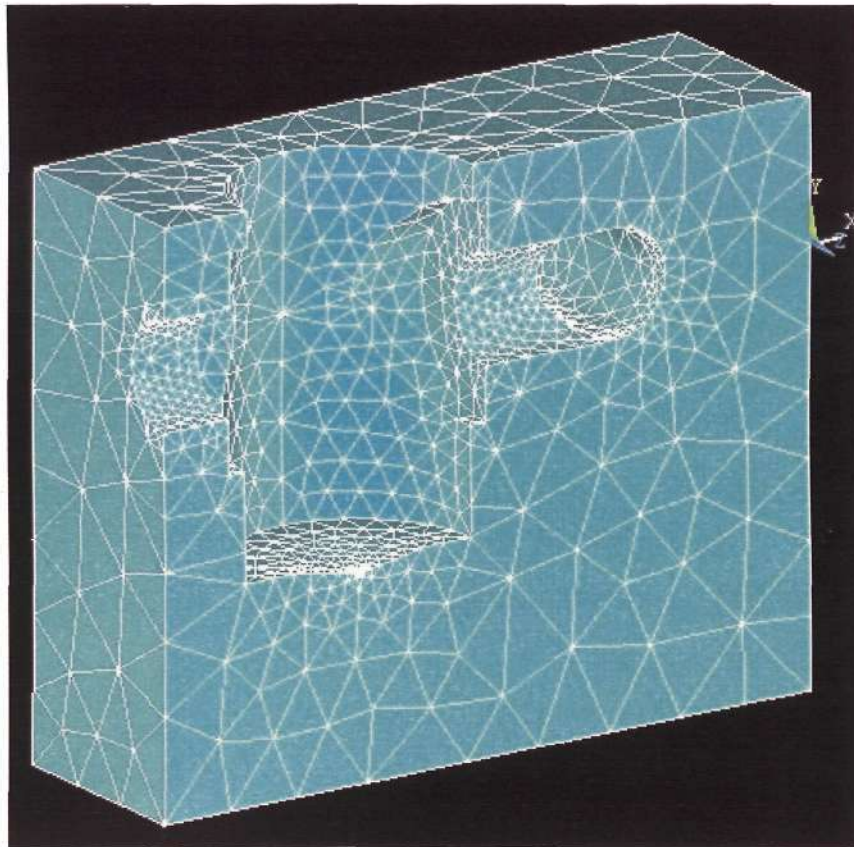




This model simulates a block valve with a valve cavity and perpendicular intersecting bores. Since there is one plane of symmetry, half of the block valve was modeled with specified symmetrical boundary conditions on the cut face.

### **Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.

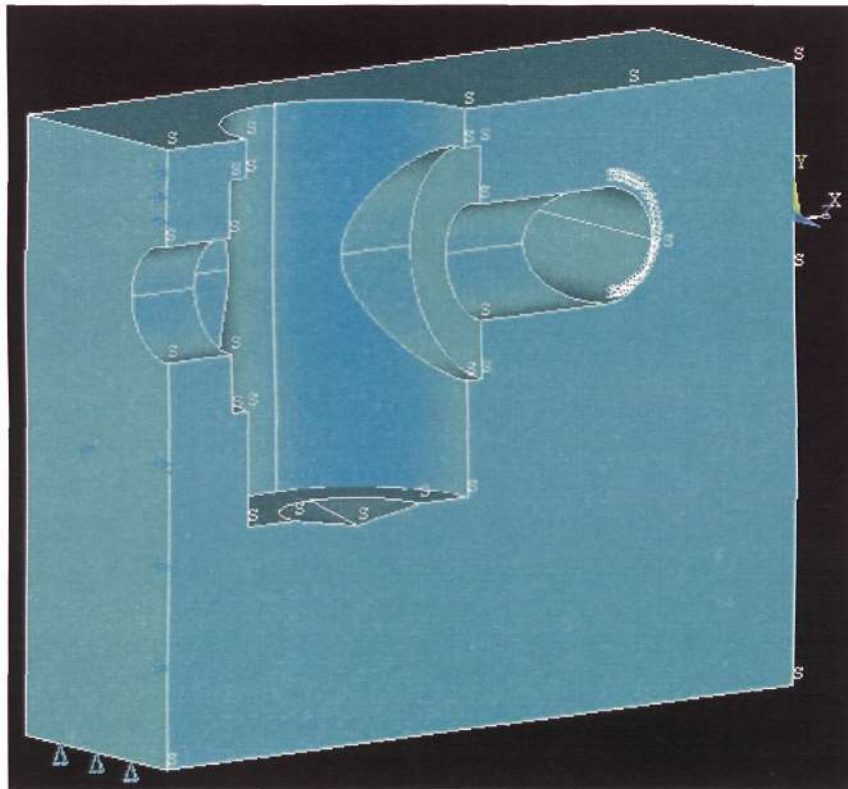


**Figure 3.32 Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). Finer meshing, higher mesh density, is deployed in the region of high stresses to simulate the stress gradients correctly.



### **Boundary conditions and constraints**



**Figure 3.33 Boundary conditions on model**

One area constraint and 2 line constraints are used. The area on one end of the block is constrained along the UX direction. The line at the base of this area is constrained along the UY direction and the adjacent line on the same area is constrained along the UZ direction. Rigid body motion is prevented. Surfaces defined with 'S' on its perimeter lie on planes of symmetry with symmetrical boundary conditions applied.

### **Assumptions**

- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.

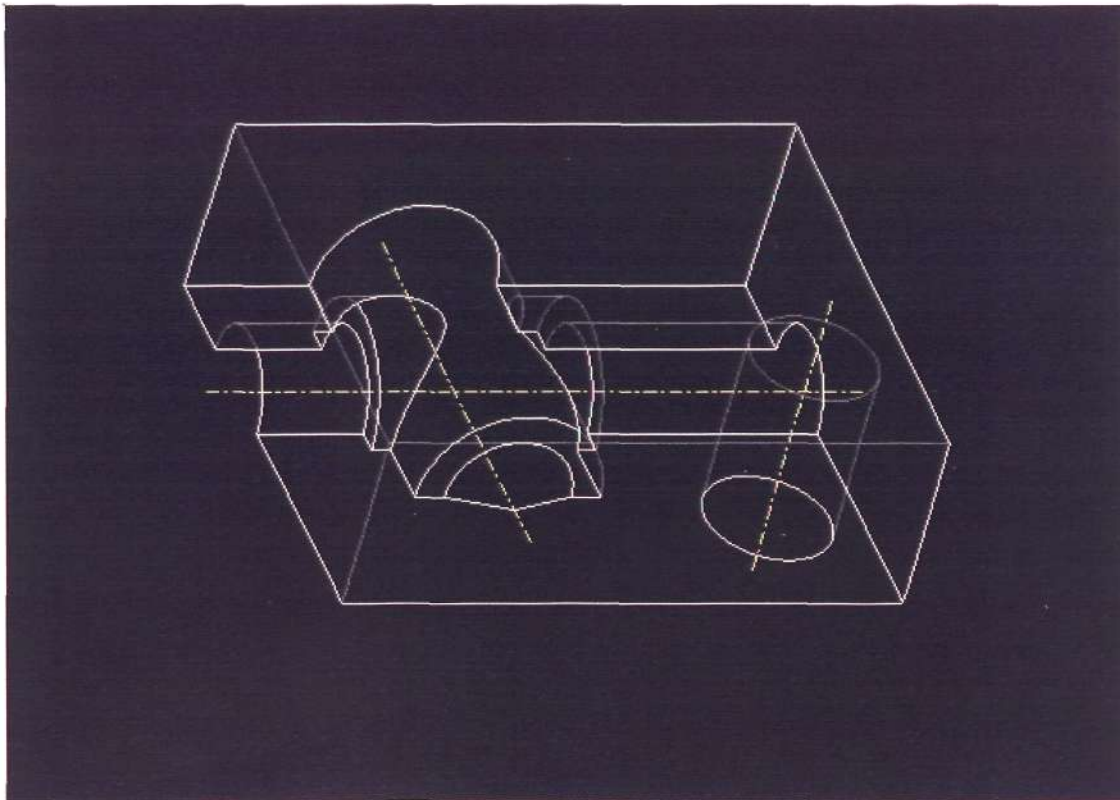




- The effect of bolts or drill holes on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.

#### ▪ Valve Cavity & Offset Perpendicular intersecting bores

#### Solid Modeling



**Figure 3.34** Reference model for Valve Cavity & Offset Perpendicular Intersecting Bores

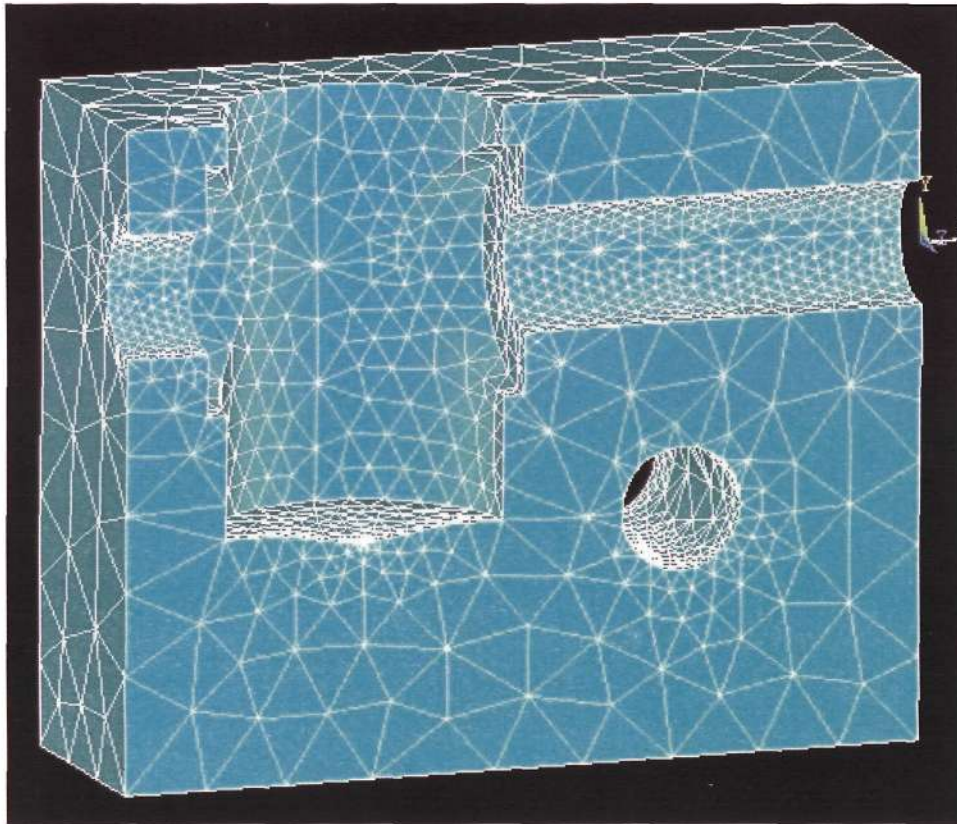
This model simulates a block valve with a valve cavity and offset perpendicular intersecting bores. Since there is one plane of symmetry, half of the block valve was modeled with specified symmetrical boundary conditions on the cut face.





### **Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.

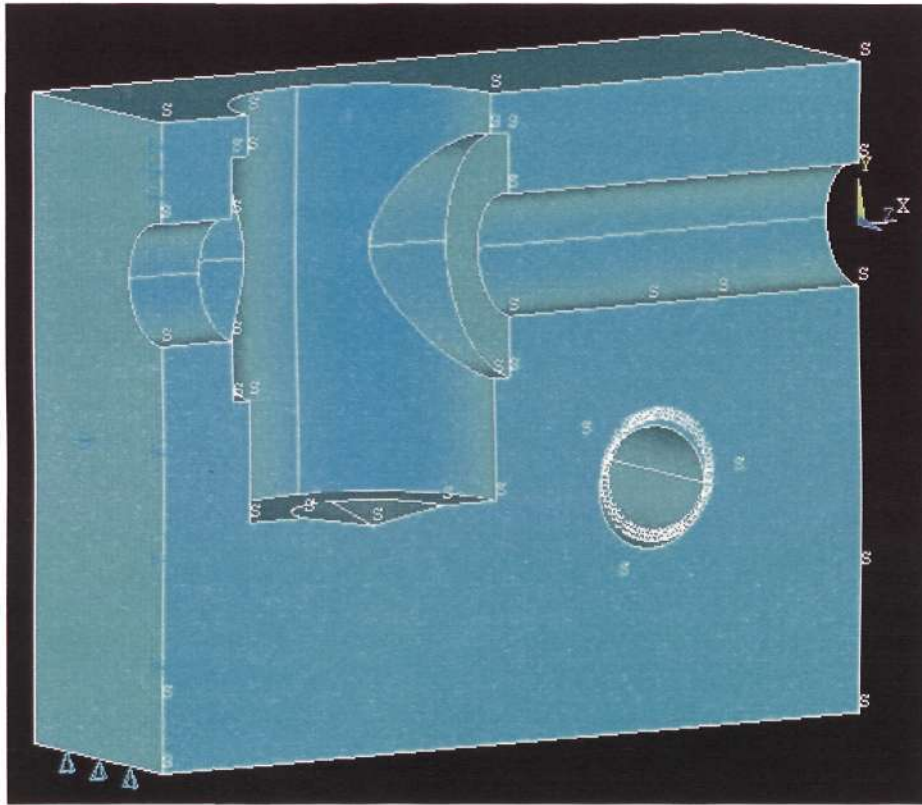


**Figure 3.35    Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). In regions of expected high stresses, the mesh density is high enough to correctly simulate the stress gradients.



### **Boundary conditions and constraints**



**Figure 3.36 Boundary conditions on model**

One area constraint and 2 line constraints are used. The area on one end of the block is constrained along the UX direction. The line at the base of this area is constrained along the UY direction and the adjacent line on the same area is constrained along the UZ direction. Rigid body motion is prevented. Surfaces defined with 'S' on its perimeter lie on planes of symmetry with symmetrical boundary conditions applied.

### **Assumptions**

- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.

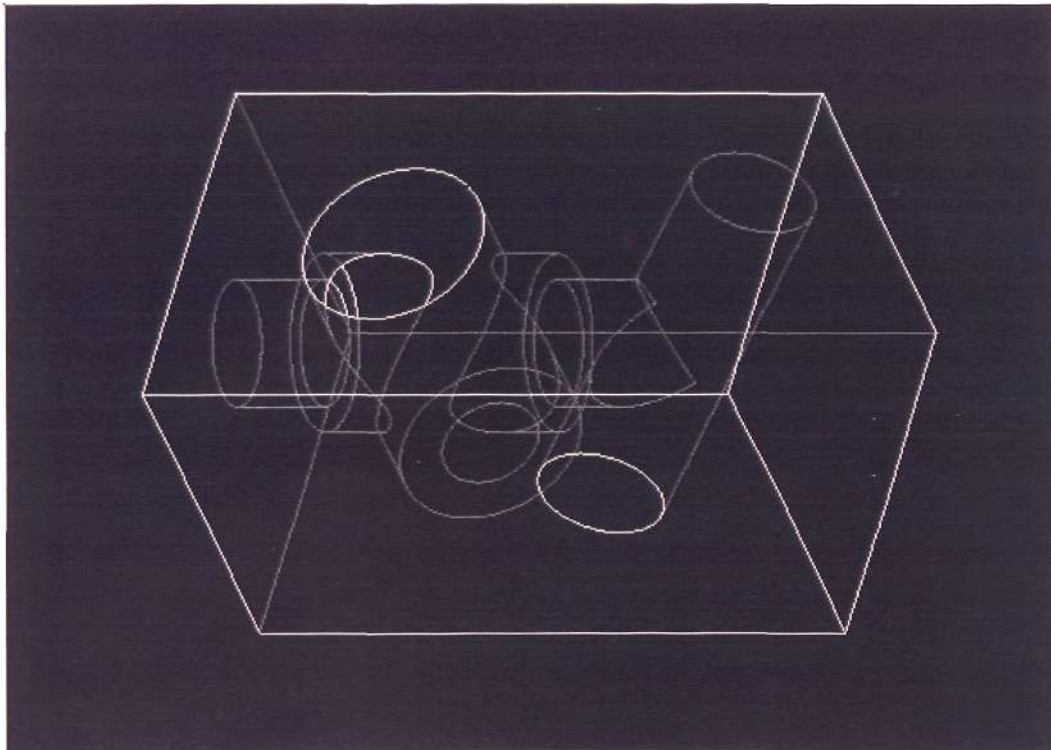




- The effect of bolts or drill holes on the exterior of the block valves on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.

▪ **Valve Cavity & Angled intersecting bores**

**Solid Modeling**



**Figure 3.37 Reference model for Valve Cavity & Angled Intersecting Bores**

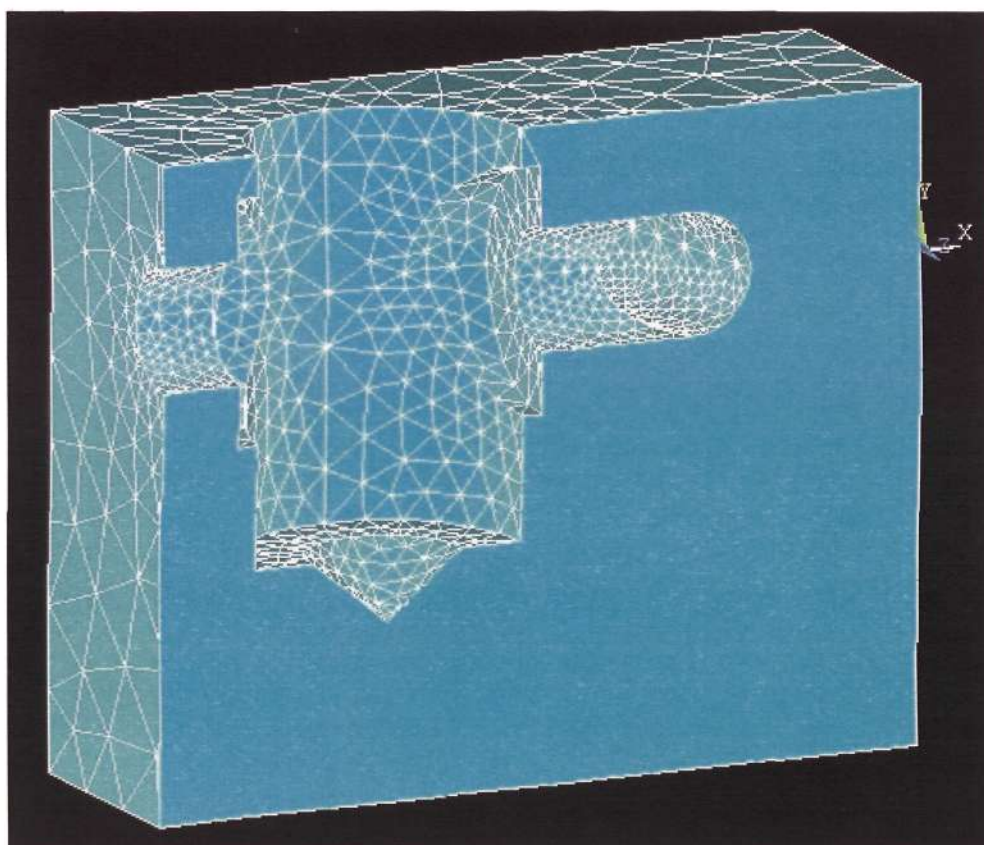
This model simulates a block valve with a valve cavity and angled intersecting bores. The bore that intersects the main valve bore is in the same plane, but the angle between the bores is not  $90^\circ$ . Since there are no planes of symmetry, the whole block valve must be modeled.





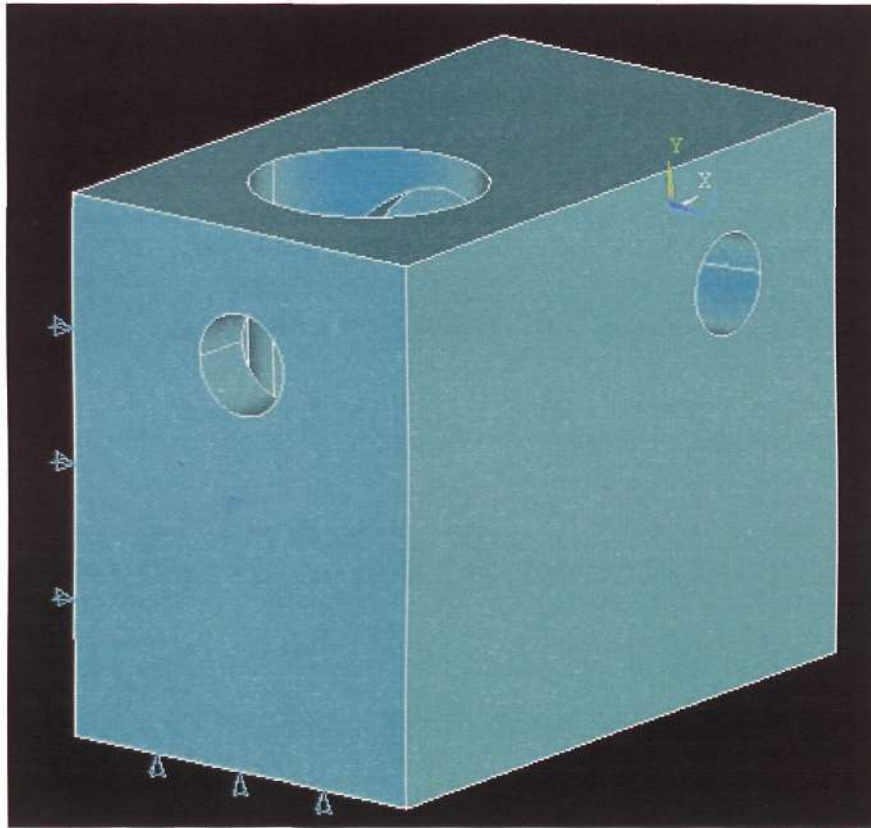
## **Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.



**Figure 3.38 Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, distance from each other, etc). In regions of expected high stresses, the mesh density is high enough to correctly simulate the stress gradients.

**Boundary conditions and constraints****Figure 3.39 Boundary conditions on model**

One area constraint and 2 line constraints are used. The area on one end of the block is constrained along the UX direction. The line at the base of this area is constrained along the UY direction and the adjacent line on the same area is constrained along the UZ direction. Rigid body motion is prevented.

**Assumptions**

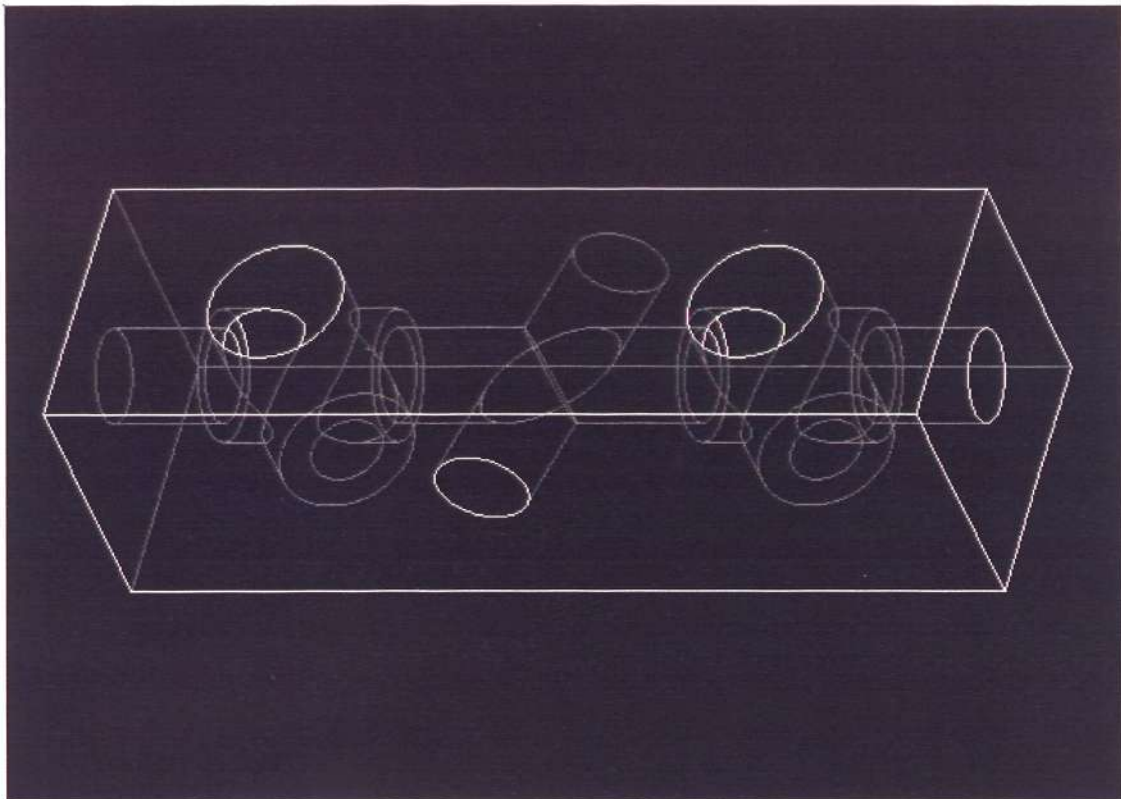
- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.
- The effect of bolts or drill holes on the stress distribution is ignored.



- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.

#### ▪ Valve Cavities & Angled intersecting bores

#### Solid Modeling



**Figure 3.40 Reference model for 2 Valve Cavities & Angled Intersecting Bores**

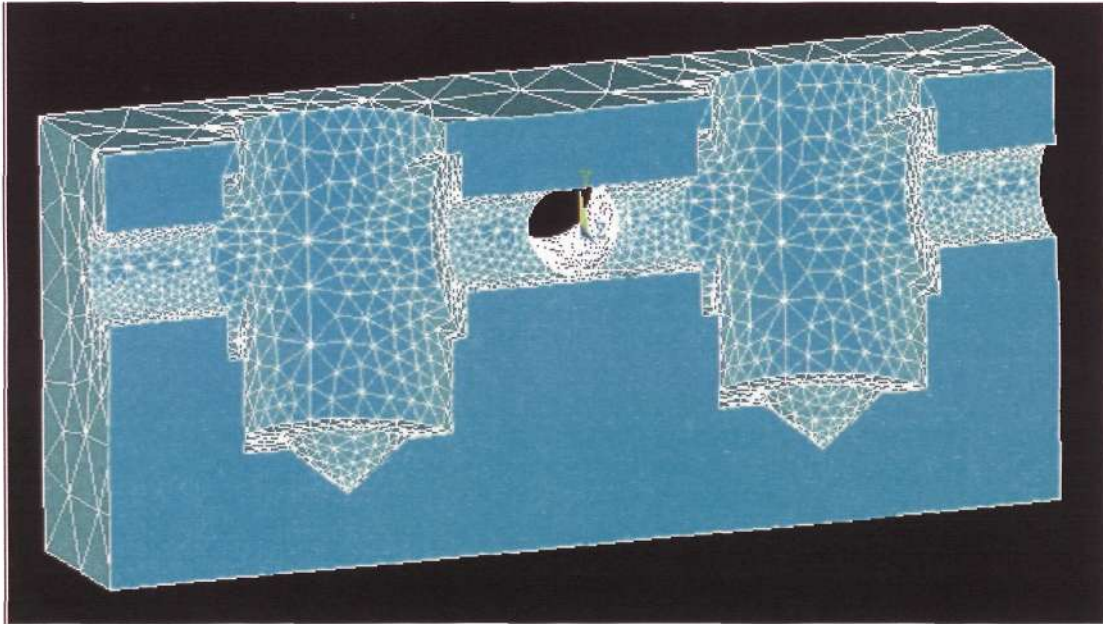
This model simulates a block valve with 2 valve cavities and angled intersecting bores. The bore that intersects the main valve bore is in the same plane, but the angle between the bores is not  $90^\circ$ . Since there are no planes of symmetry, the whole block valve must be modeled.





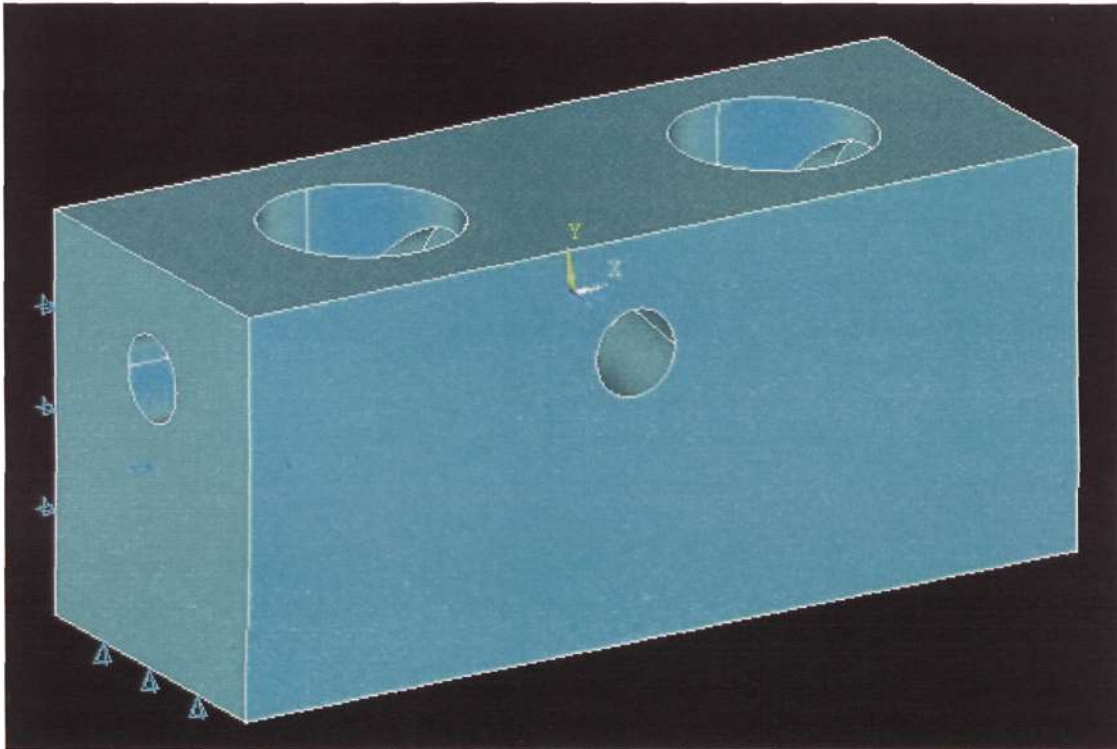
### **Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.



**Figure 3.41    Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, cavity size, distance from each other, etc). In regions of expected high stresses, the mesh density is high enough to correctly simulate the stress gradients.

**Boundary conditions and constraints**

**Figure 3.42 Boundary conditions on model**

One area constraint and 2 line constraints are used. The area on one end of the block is constrained along the UX direction. The line at the base of this area is constrained along the UY direction and the adjacent line on the same area is constrained along the UZ direction. Rigid body motion is prevented.

**Assumptions**

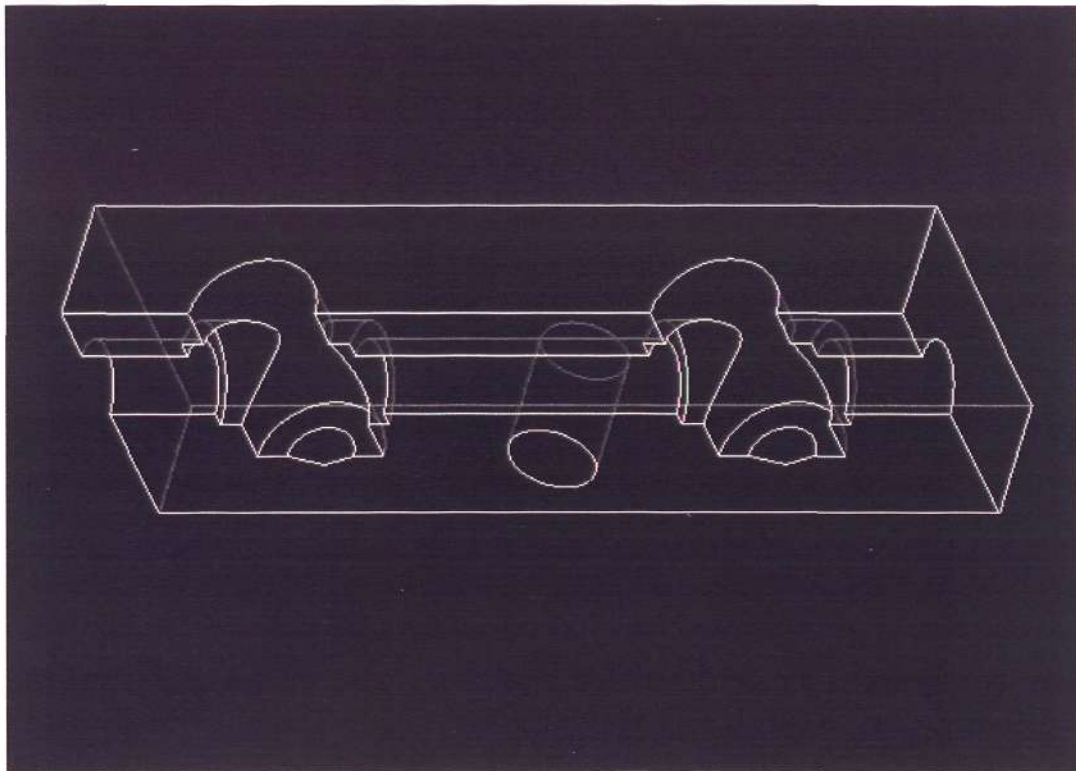
- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.
- The effect of bolts or drill holes on the stress distribution is ignored.



- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.

#### ▪ Valve Cavities & Offset Perpendicular intersecting bores

#### Solid Modeling



**Figure 3.43 Reference model for 2 Valve Cavities & Offset Perpendicular Intersecting Bores**

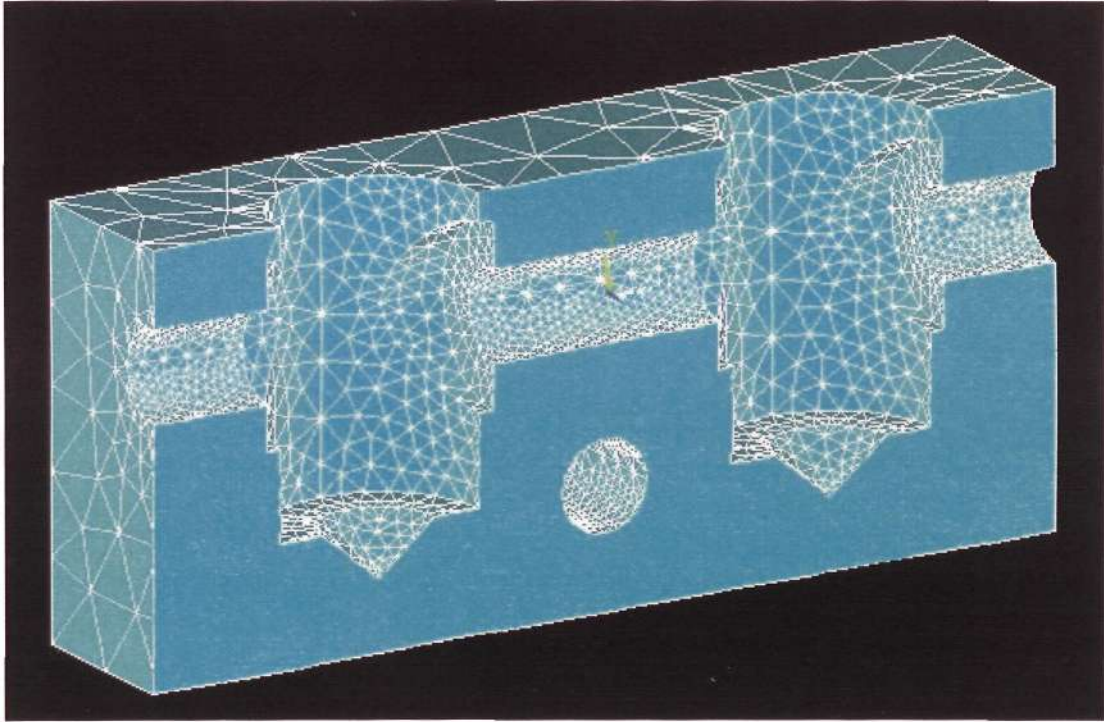
This model simulates a block valve with 2 valve cavities and angled intersecting bores. Since there is 1 plane of symmetry, only half of the block valve will be modeled.





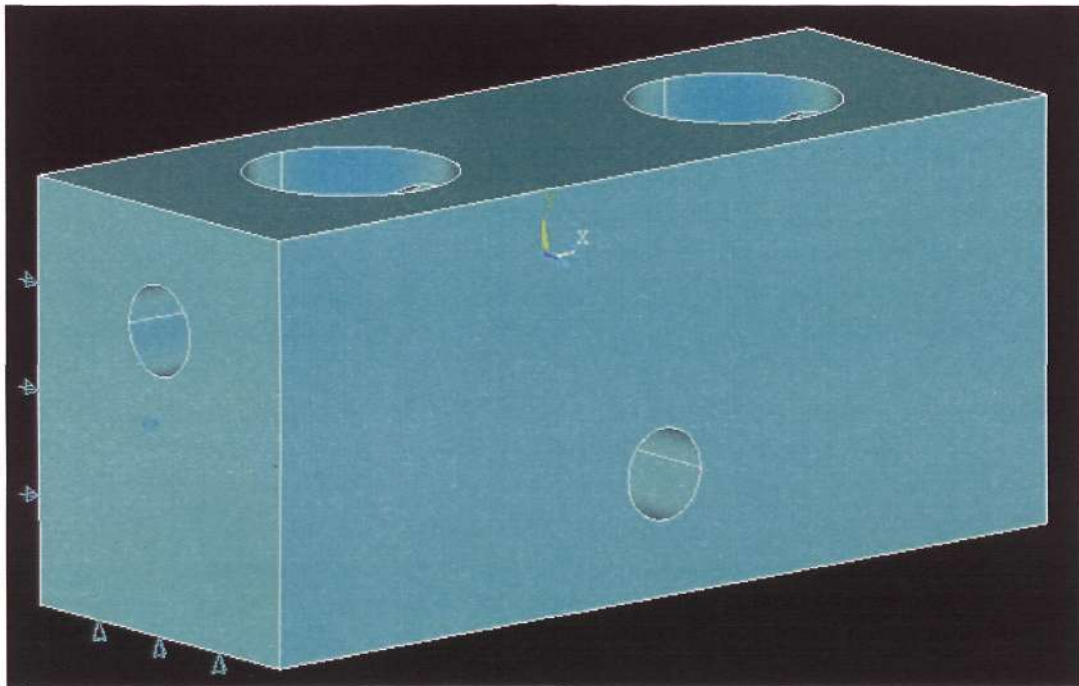
### **Meshing**

Solid87 and Solid92 elements are used to mesh the model, for the thermal and structural analysis respectively.



**Figure 3.44 Meshing of model**

The mesh size and density will change accordingly when the parameters are changed (such as bore size, cavity size, distance from each other, etc). In regions of expected high stresses, the mesh density is high enough to correctly simulate the stress gradients.

**Boundary conditions and constraints**

**Figure 3.45 Boundary conditions on model**

One area constraint and 2 line constraints are used. The area on one end of the block is constrained along the UX direction. The line at the base of this area is constrained along the UY direction and the adjacent line on the same area is constrained along the UZ direction. Rigid body motion is prevented.

**Assumptions**

- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides, meaning there are no external support constraints on the block. It is not under any gravitational force.
- The effect of bolts or drill holes on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.



## 4. ANALYSIS METHODOLOGY

This chapter begins by introducing the 2 models and the critical part of the models to be analyzed. The geometric, material properties, loads and variables to be applied were listed in detail.

### 4.1 Models

The 2 models analyzed in this report were Parallel Bores (Figure 4.1) and Valve Cavity & Perpendicular intersecting bores (non-through) (Figure 4.2).

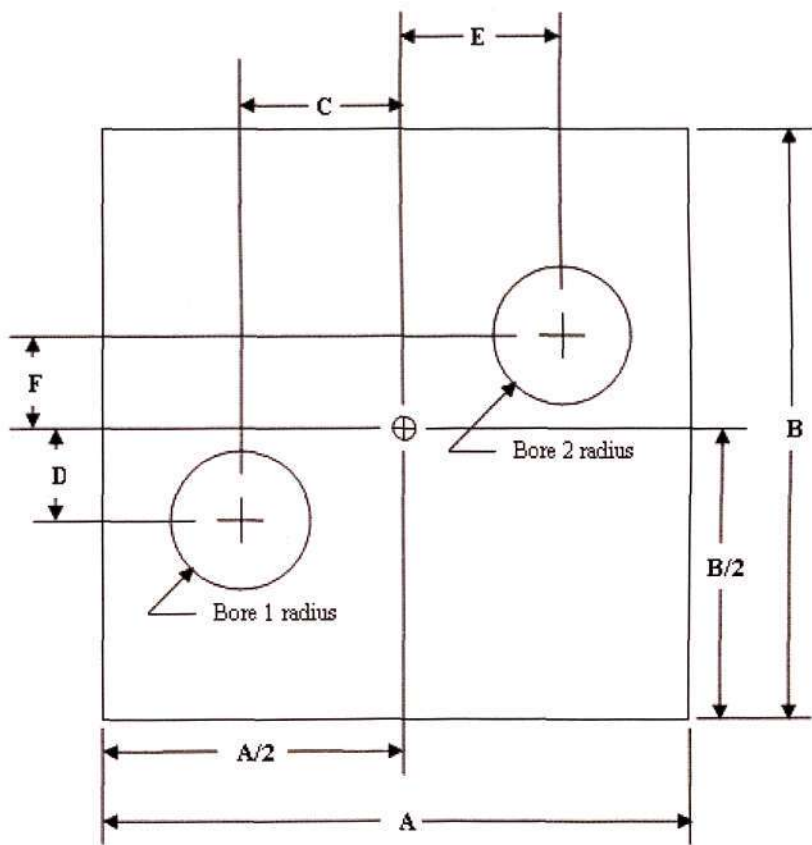
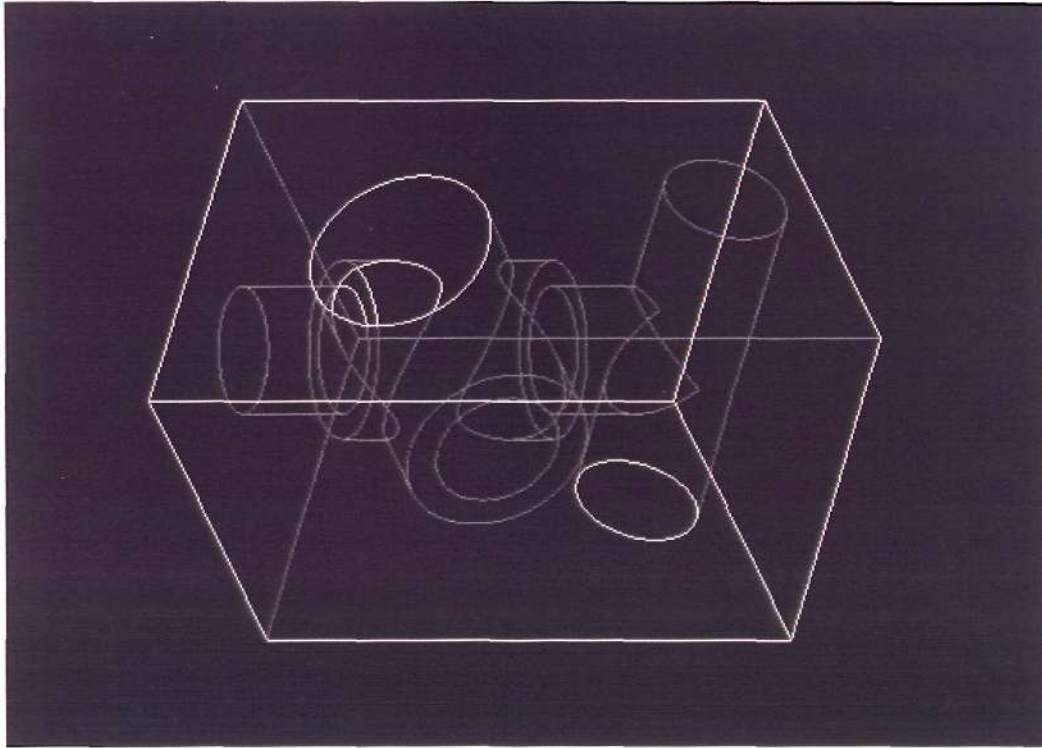


Figure 4.1 Parallel Bores





**Figure 4.2** Valve Cavity & Perpendicular intersecting bores (non-through)

## 4.2 Objectives

---

The objectives of this finite element analysis are to carry out parametric studies on the effect of wall thickness, internal pressure in bores and cavities and internal temperature on the Von Mises stress for the two different models. Figure 4.3 and 4.4 illustrates the wall thickness in the models to be studied respectively.

Both the Von Mises and Maximum shear stress (Tresca) criterion are applicable to ductile materials. Experiments suggest that the Von Mises yield criterion is the one which provides better agreement with observed behaviour than the Tresca yield criterion. Tresca criterion has been shown to be very conservative for ductile materials [6].

ANSI/API 6A also states that Von Mises criterion may be used for design calculations for pressure-containing equipment.

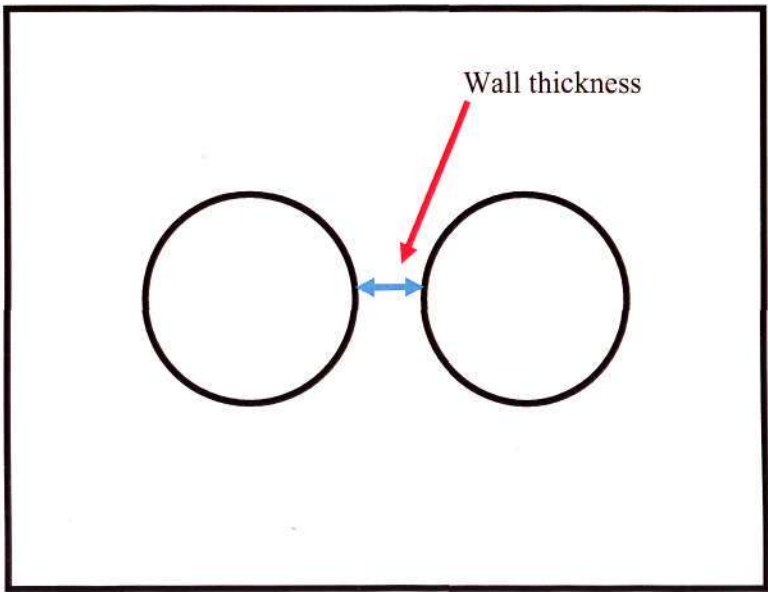


Figure 4.3 Wall thickness for Parallel bores

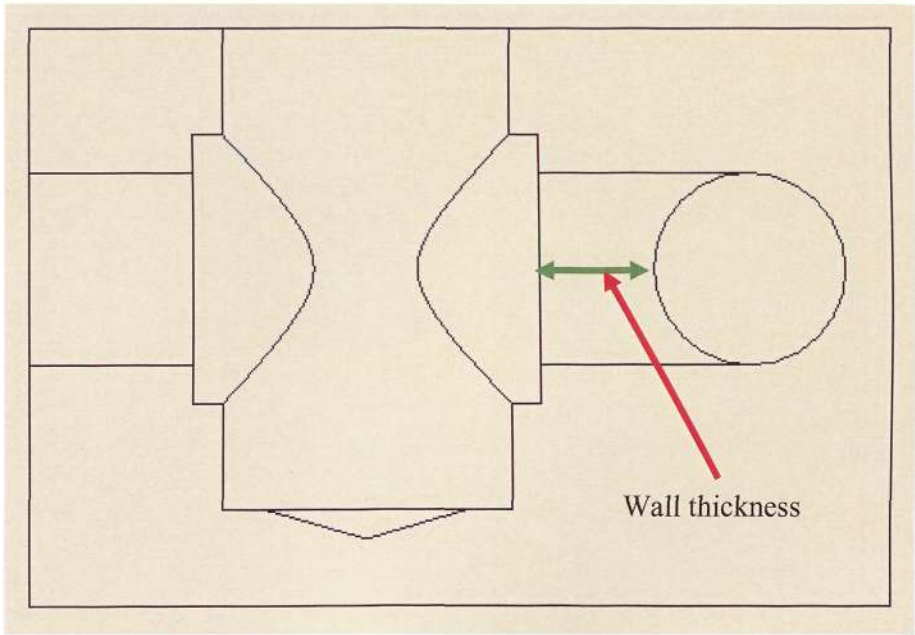


Figure 4.4 Wall thickness for Valve Cavity & Perpendicular intersecting bores (non-through)



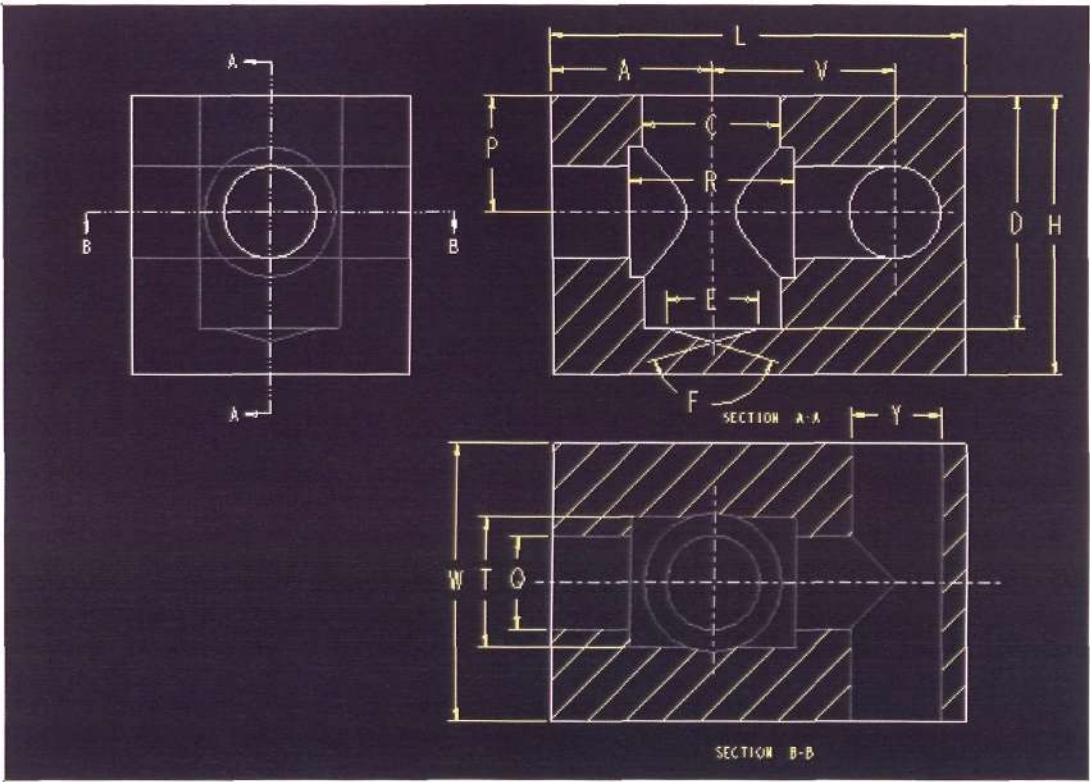
4.3 Properties

Table 4.1 lists the various geometric, material properties and loading conditions used in this parametric study for Parallel Bores (Refer to Figure 4.1 for parameter labels). Table 4.2 lists the various geometric, material properties and loading conditions used in this parametric study for Valve Cavity & Perpendicular intersecting bores (non-through). Variables for the analysis are highlighted in blue.

Parallel Bores		
Geometric Properties	Units	Value
A	m	0.75
B	m	0.75
Radius of bore 1	m	0.065
C	m	0.1
D	m	0
Radius of bore 2	m	0.065
E	m	0.1
F	m	0
Wall thickness	m	0.07
Material Properties		
Young's Modulus	Pa	2E+11
Poisson Ratio		0.3
Thermal conductivity	W/mK	42.6536
Coefficient of thermal expansion	µm/m-K	11.34
Yield Strength	Pa	4.48E+08
Loads		
External pressure	Pa	0
Bore 1 Pressure	Pa	13800000
Bore 2 Pressure	Pa	13800000
Bore 1 Temperature	K	275
Bore 2 Temperature	K	275
External Temperature	K	275
Reference Temperature	K	275

Table 4.1 Properties used for Parallel Bores





**Figure 4.5** Geometric parameter labels for Valve Cavity & Perpendicular intersecting bores (non-through)

The geometric parameter labels for Valve Cavity & Perpendicular intersecting bores (non-through) are indicated in Figure 4.5.



Valve Cavity & Perpendicular intersecting bores		
Geometric Properties	Units	Value
L	m	0.55
W	m	0.4
H	m	0.5
A	m	0.2
C	m	0.1651
D	m	0.40132
E	m	0.048
F	rad	2.7925
P	m	0.2
Q	m	0.12
R	m	0.18
T	m	0.1526
V	m	0.16
Y	m	0.12
wall thickness	m	0.01
Material Properties		
Young's Modulus	Pa	2E+11
Poisson Ratio		0.3
Thermal conductivity	W/mK	42.6536
Coefficient of thermal expansion	µm/m-K	11.34
Yield Strength	Pa	4.48E+08
Loads		
External pressure	Pa	0
Cavity pressure	Pa	20700000
Flowbore pressure	Pa	20700000
Crossbore pressure	Pa	0
Cavity Temperature	K	300
Flowbore Temperature	K	300
Crossbore Temperature	K	275
External Temperature	K	275
Reference Temperature	K	275

Table 4.2      Properties used for Valve Cavity & Perpendicular intersecting bores  
(non-through)



4.4 Variables

Table 4.3 and Table 4.4 lists the variables used in parametric studies for Parallel Bores and Valve Cavity & Perpendicular intersecting bores (non-through) respectively. The values chosen were based on the rated working pressures and temperature classification V for FL and FLS gate valves.

Variables for Parallel Bores	Units	Values				
C	m	0.1	0.09	0.08	0.075	0.0725
E	m	0.1	0.09	0.08	0.075	0.0725
Wall thickness	m	0.07	0.05	0.03	0.02	0.015
Bore 1 Pressure	Pa	13800000	34500000	69000000	138000000	
Bore 2 Pressure	Pa	13800000	34500000	69000000	138000000	
Bore 1 Temperature	K	275	334.5	394		
Bore 2 Temperature	K	275	334.5	394		

Table 4.3 Variables used in parametric study for Parallel Bores

Variables for Valve Cavity & Perpendicular intersecting bores	Units	Values				
V	m	0.16	0.17	0.18	0.2	0.25
wall thickness	m	0.01	0.02	0.03	0.05	0.1
Cavity pressure	Pa	20700000	69000000	103500000		
Flowbore pressure	Pa	20700000	69000000	103500000		
Cavity Temperature	K	300	345	394		
Flowbore Temperature	K	300	345	394		

Table 4.4 Variables used in parametric study for Valve Cavity & Perpendicular intersecting bores (non-through)





## **4.5 Finite Element Analysis**

---

The programs listed in Chapter 3 for the Parallel Bores and Valve Cavity & Perpendicular intersecting bores (non-through) were used for running the finite element analysis. Parameter values from Table 4.1, 4.2, 4.3 and 4.4 were input into the program and graphical plots of the stress contours were obtained. Location and value of maximum von Mises stresses were obtained from the graphical plots. From the results obtained, different graphs were plotted and will be discussed in detail in Chapter 5. Stress data for the plots were entered manually in Excel and plotted.



## 5. RESULTS AND DISCUSSION

This chapter begins by illustrating the contour stress plots and plots of the graphs of Von Mises Stress vs wall thickness for different internal bore and cavity pressures and temperatures. From the graphs plotted, trends were observed and the effect of the parametric variables was discussed.

### 5.1 Parallel Bores

- **Ansys stress contour plot**

A sample contour stress plot is shown in Figure 5.1 and Figure 5.2.

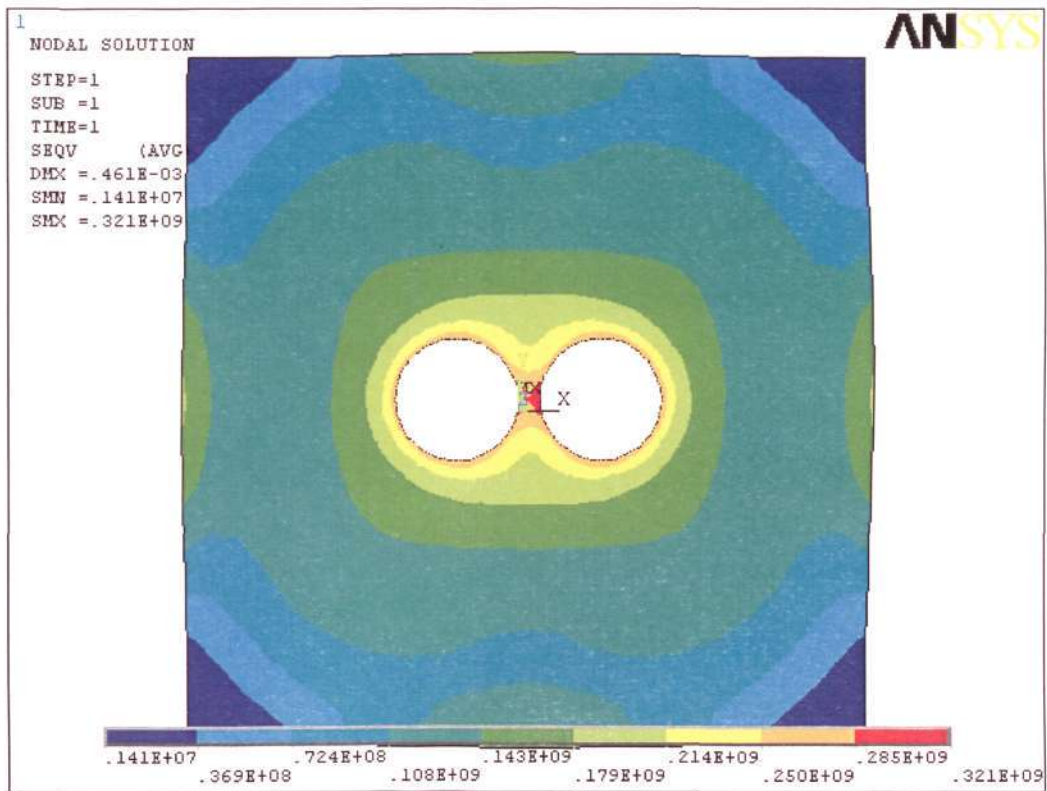


Figure 5.1 Contour stress plot

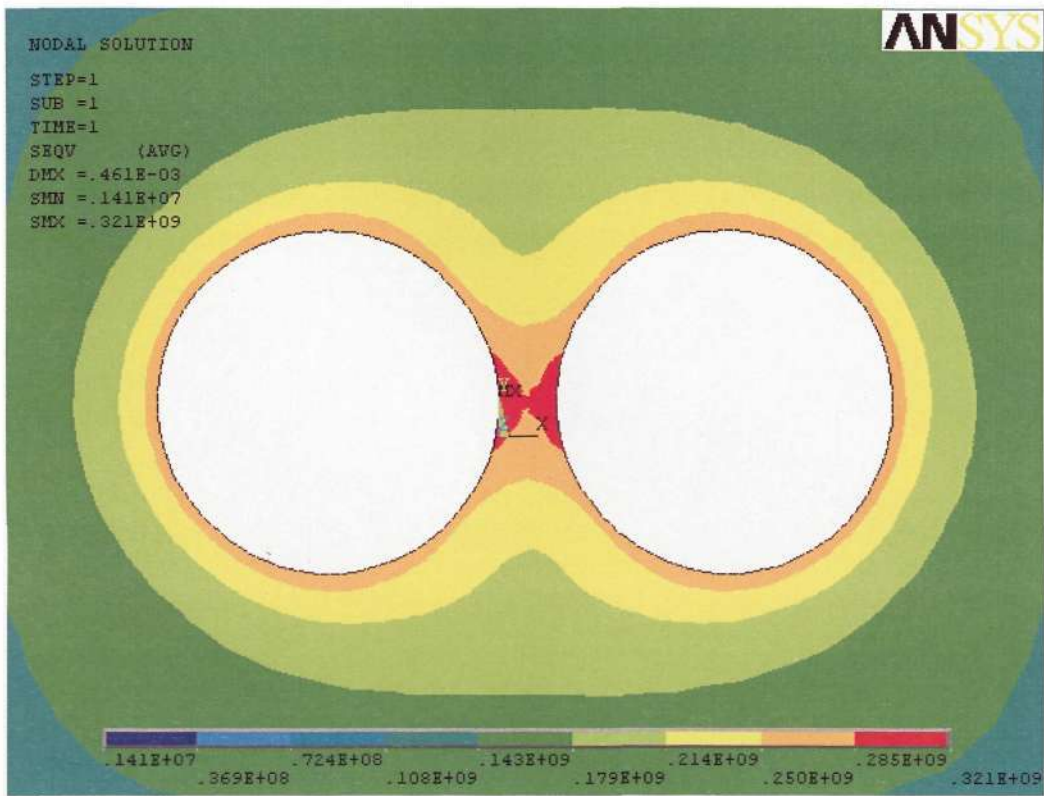
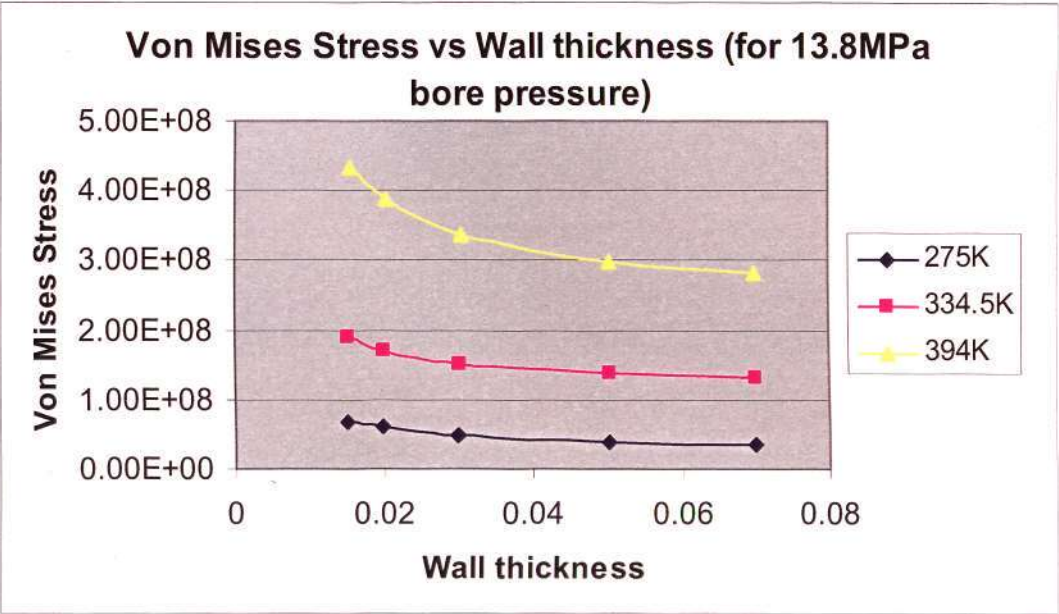


Figure 5.2 Contour stress plot zoom in

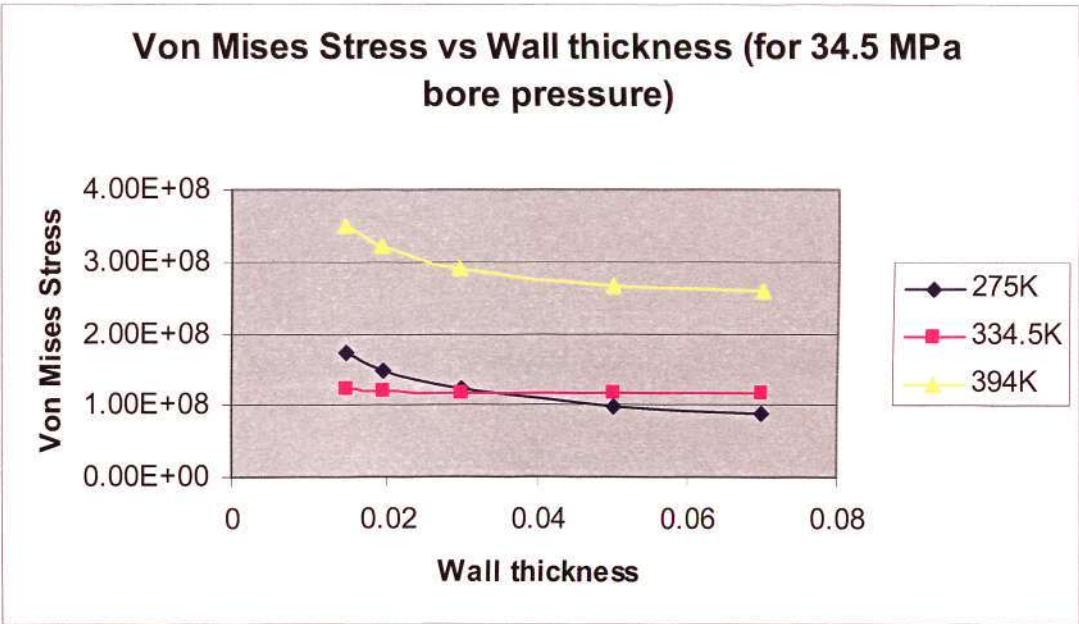




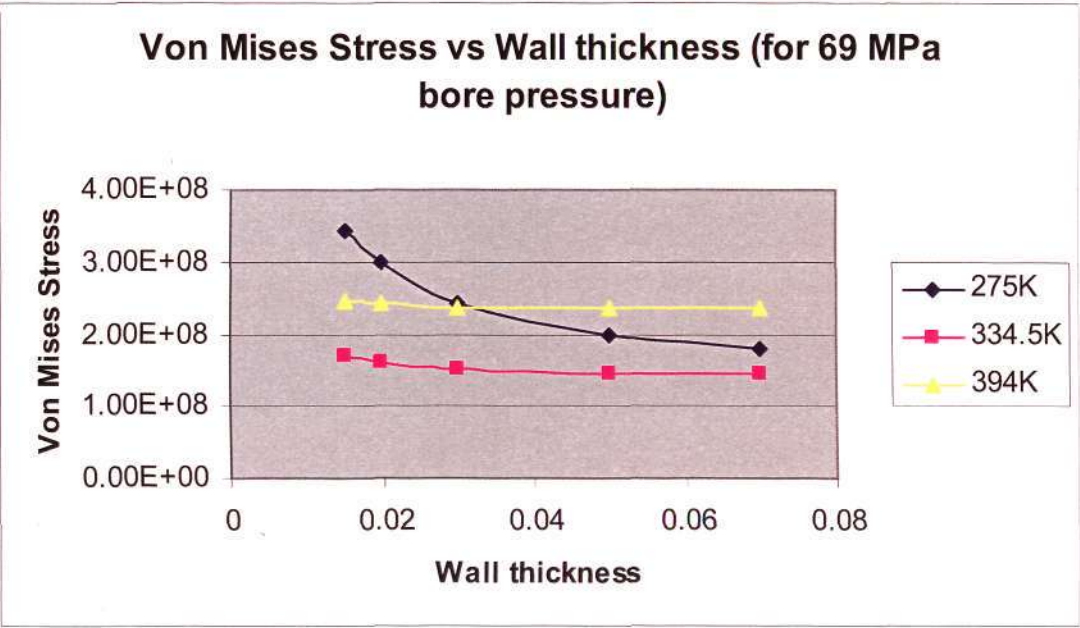
▪ Graphs



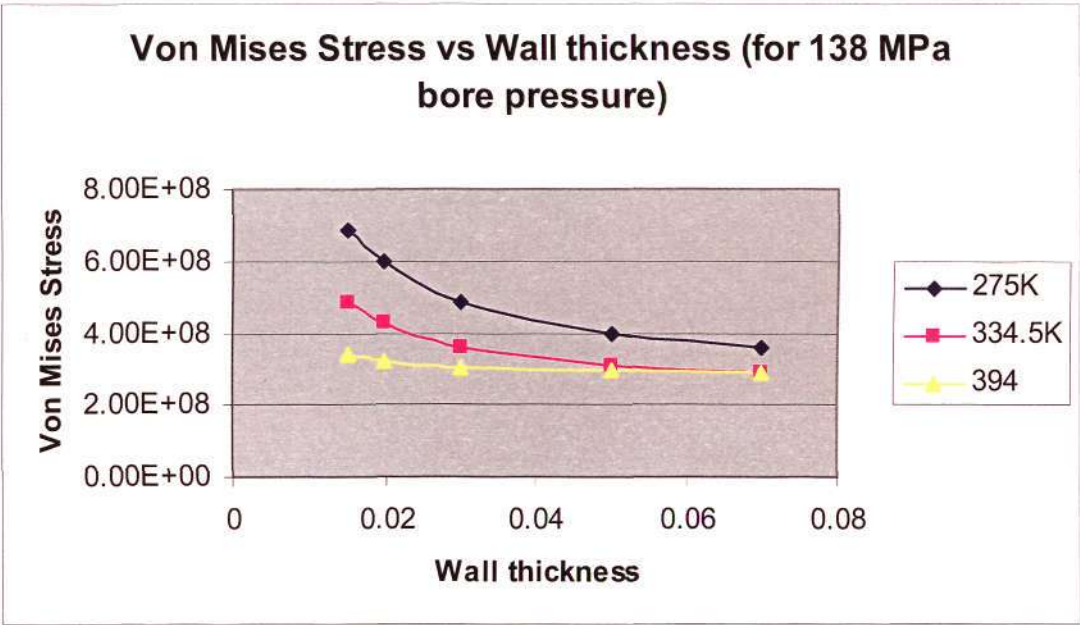
Graph 5.1 Variation of Von Mises Stress vs Wall thickness (for 13.8 MPa bore pressure)



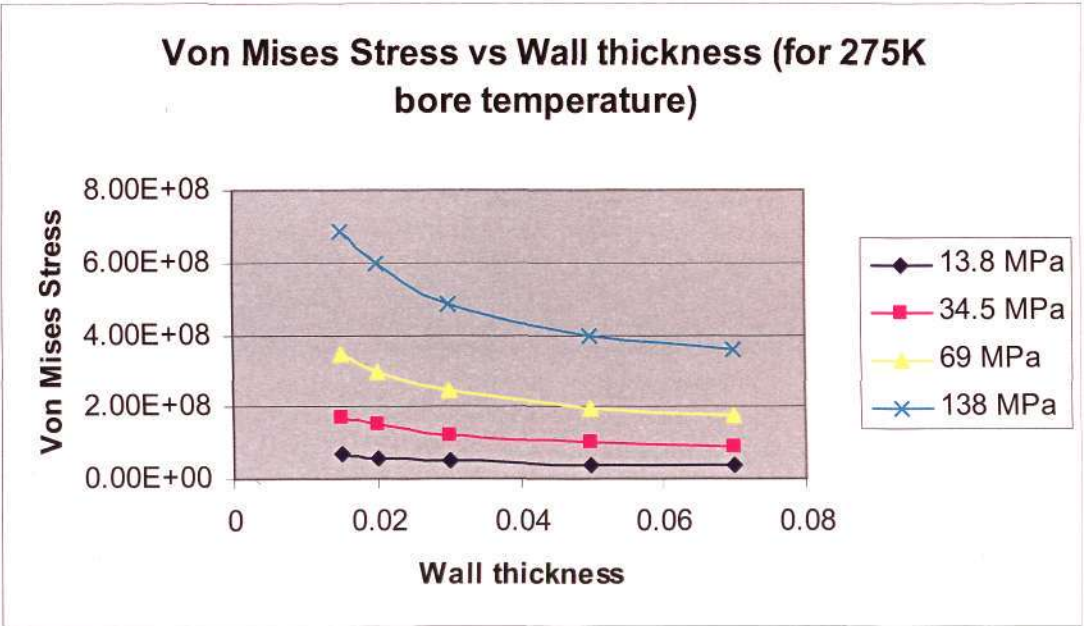
Graph 5.2 Variation of Von Mises Stress vs Wall thickness (for 34.5 MPa bore pressure)



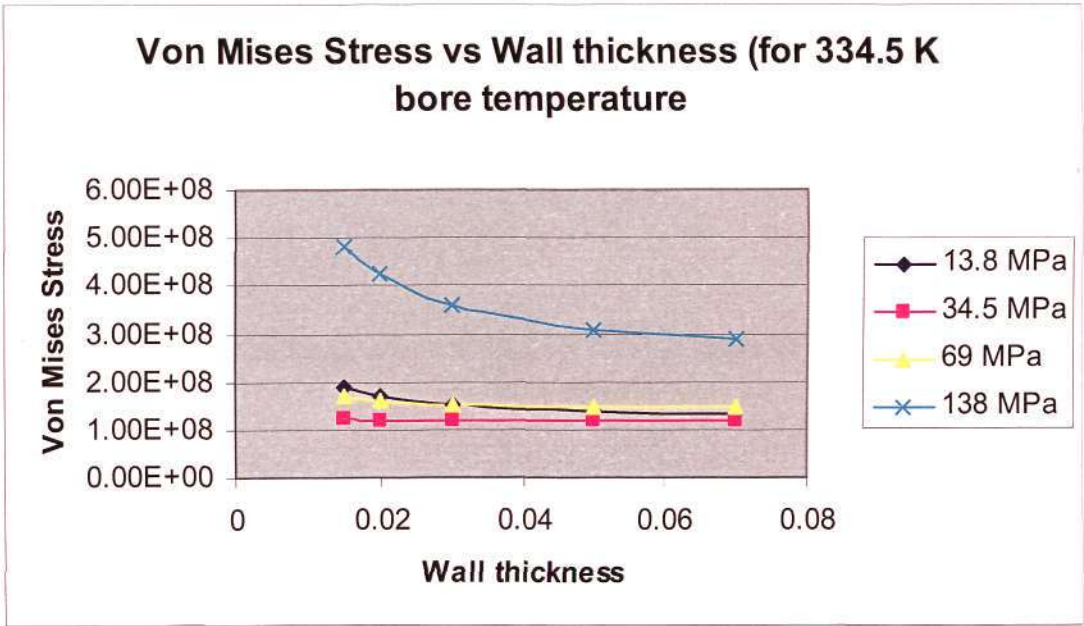
Graph 5.3 Variation of Von Mises Stress vs Wall thickness (for 69 MPa bore pressure)



Graph 5.4 Variation of Von Mises Stress vs Wall thickness (for 138 MPa bore pressure)

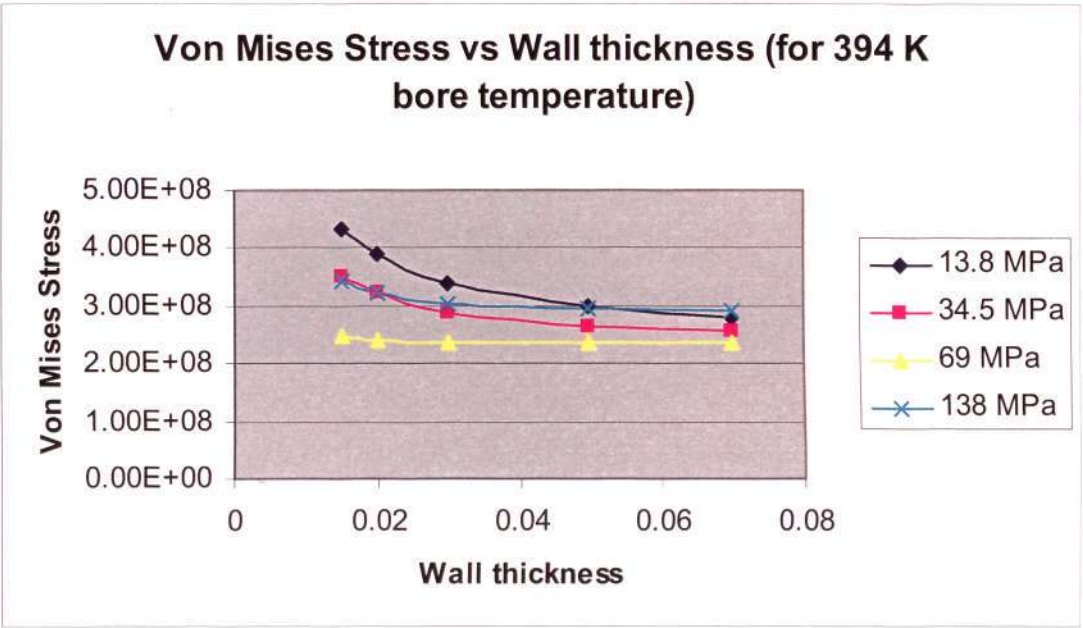


Graph 5.5 Variation of Von Mises Stress vs Wall thickness (for 275K bore temperature)



Graph 5.6 Variation of Von Mises Stress vs Wall thickness (for 334.5K bore temperature)





Graph 5.7 Variation of Von Mises Stress vs Wall thickness (for 394K bore temperature)

▪ Discussion

The three main variables that change are the internal temperatures in the bores, internal pressures and the wall thickness between the two bores.

Effect of internal temperature

The effect of temperature on stress can be investigated by keeping the internal pressures of the two bores and wall thickness fixed. From Graph 5.1, it was observed that for each wall thickness, the Von Mises stress increased when the internal temperatures increased from 275K to 394K. This can be explained by the increase in thermal stresses when the temperature increased, leading to overall higher stresses throughout the model.



From the equation below,

$$\sigma_t = E \epsilon_t$$

$$\epsilon_t = \alpha \Delta T$$

where

$\sigma_t$  = thermal stress

$\epsilon_t$  = thermal strain

$\alpha$  = coefficient of thermal expansion

$$\Delta T = T - T_{\text{ref}}$$

$T$  = current temperature at the point in question

$T_{\text{ref}}$  = reference (strain-free) temperature

The external temperature is the same as the reference temperature in the current analysis. When the internal temperature of the bores increases, the temperature gradient between the bores and the exterior of the block valve increases, giving a higher  $\Delta T$ . This in turn will give rise to a higher thermal strain and thermal stress.

An interesting observation was made for Graph 5.4. For each wall thickness, the Von Mises stress decreased when the internal temperatures increased from 275K to 394K. Based on the above explanation, when the internal temperature increase, the Von Mises stress should increase and this seems contradictory to the observation made in Graph 5.4. Looking at the trend of the curves from Graph 5.5 to Graph 5.7, as the internal temperature increased, curves for lower internal pressures started shifting upwards and curves for higher internal pressures shifting downwards. The Von Mises stresses are increasing for curves with lower internal pressures and decreasing for curves with higher internal pressures.

This can be explained by Figure 5.3 and Figure 5.4.

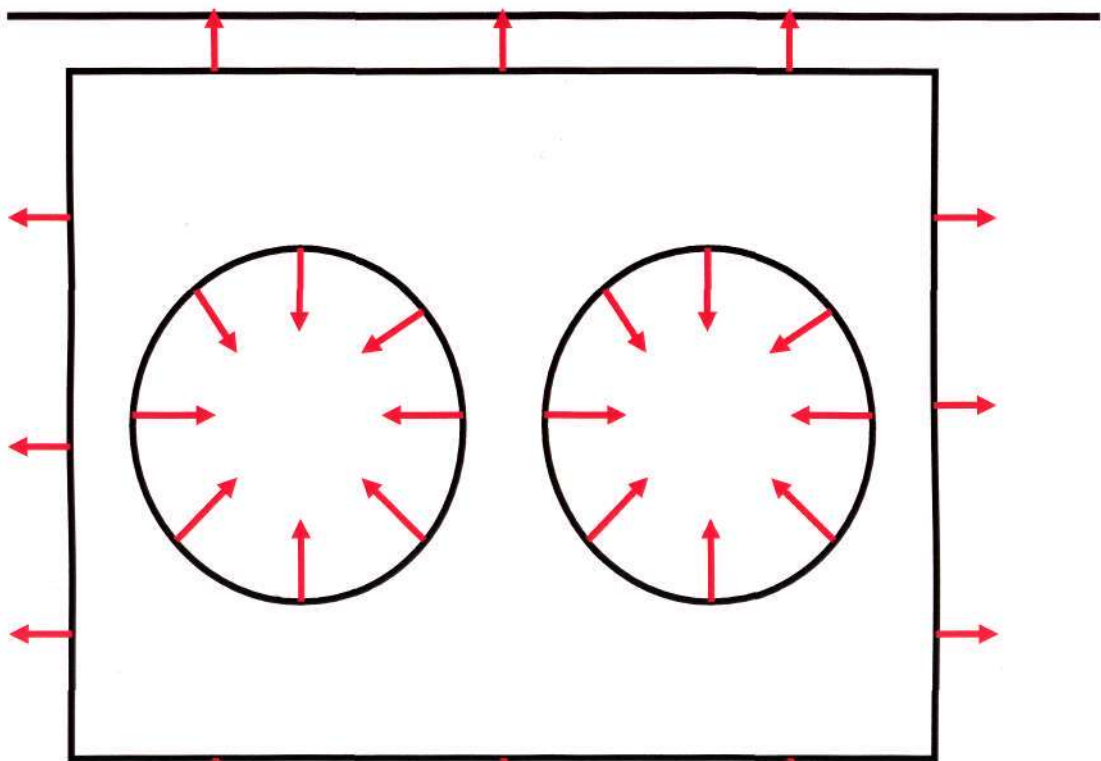


Figure 5.3 Tensile thermal stresses

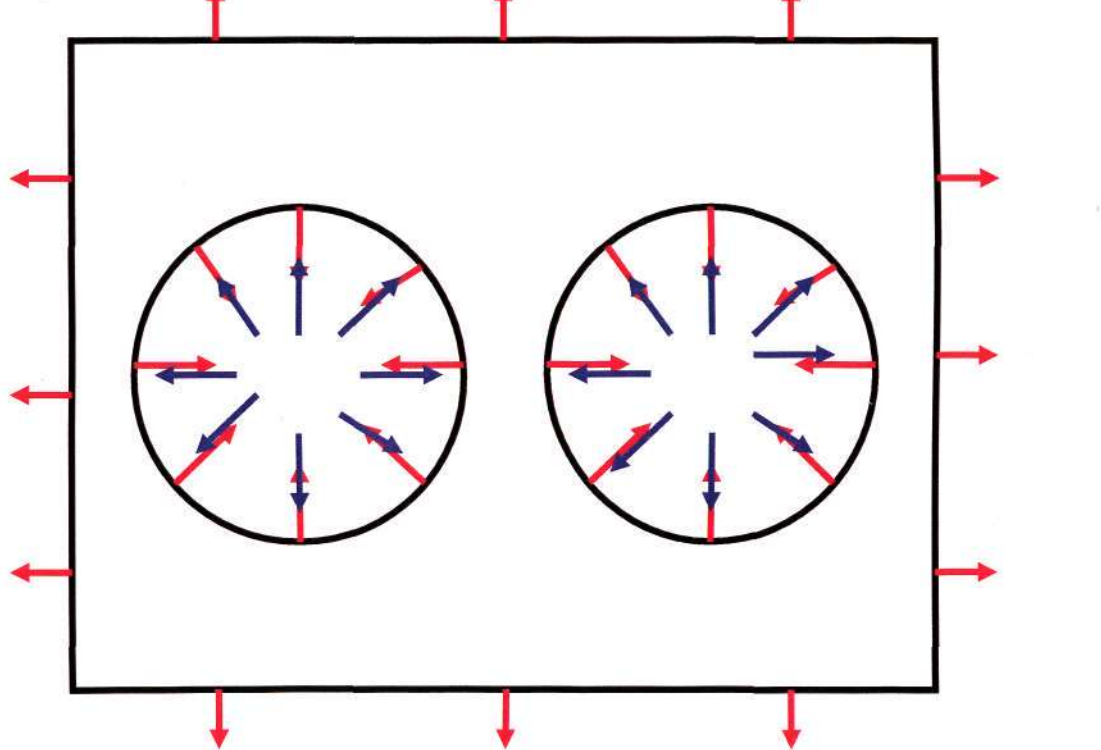


Figure 5.4 Tensile thermal stresses counteracted by compressive stress





Figure 5.3 illustrates the tensile stresses when no internal pressures exist and only thermal stresses are present due to thermal expansion. When internal pressures exist in the bores as shown in Figure 5.4, they exert a compressive stress on the bores, canceling out some of the expansive thermal stresses. Therefore, when the internal pressures increase to say 138 MPa in Graph 5.4, the structural stresses caused by internal pressures canceled out part of the thermal stresses induced by the temperature gradients.

#### Effect of internal pressure

The effect of internal pressure on stress can be investigated by keeping the internal temperatures of the two bores and wall thickness fixed. From Graph 5.5, it was observed that for each wall thickness, the Von Mises stress increased when the internal pressures increased from 13.8 MPa to 138 MPa. Since the internal temperature and external temperature were both at 275K, no thermal strain and stresses exist and the Von Mises stress was predominantly structural stress. When internal pressure increases, a greater compressive force is exerted on the bores, resulting in higher Von Mises stresses.

An observation was made on Graph 5.7 where a lower internal pressure gave higher Von Mises stresses. This can be similarly explained by the fact that some of the structural stresses cancel out the thermal stresses.

#### Effect of wall thickness

The effect of wall thickness between the bores on stress can be investigated by keeping the internal temperatures and pressures of the two bores fixed. From Graphs 5.1 to 5.7, the curves were all sloping downwards. It was observed that as the wall thickness increased, the Von Mises stress decreased, regardless of the internal pressure and temperature.



## 5.2 Valve Cavity & Perpendicular intersecting bores (non-through)

- Ansys stress contour plot

A sample contour stress plot is shown in Figure 5.5 and Figure 5.6.

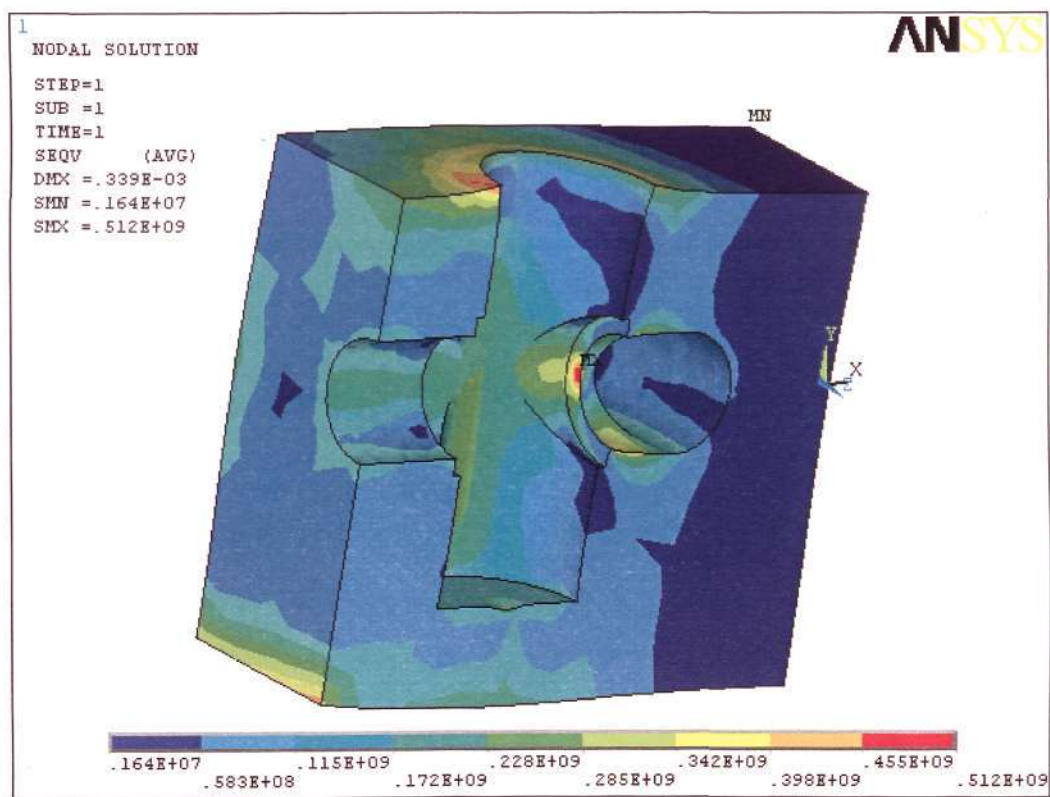


Figure 5.5 Contour stress plot

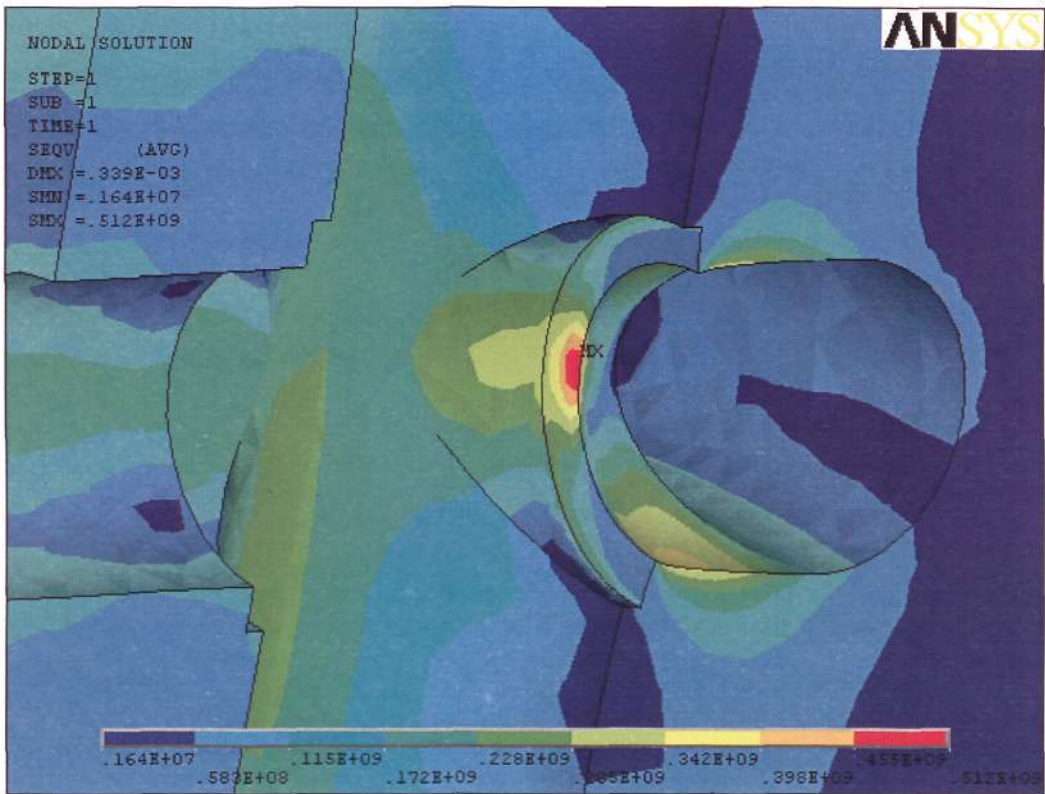
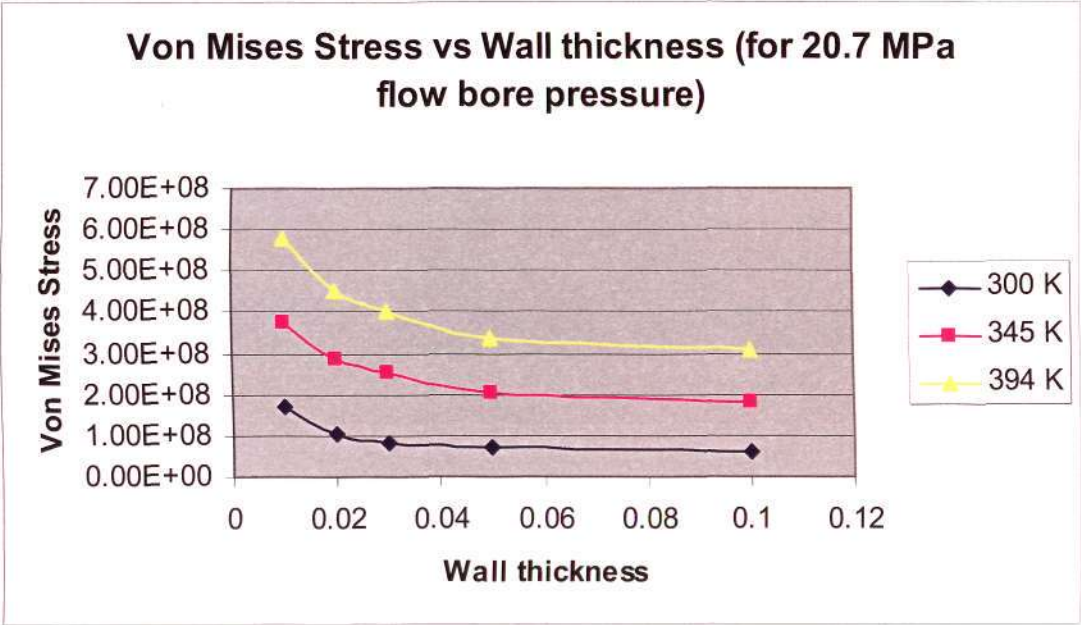


Figure 5.6 Contour stress plot zoom in

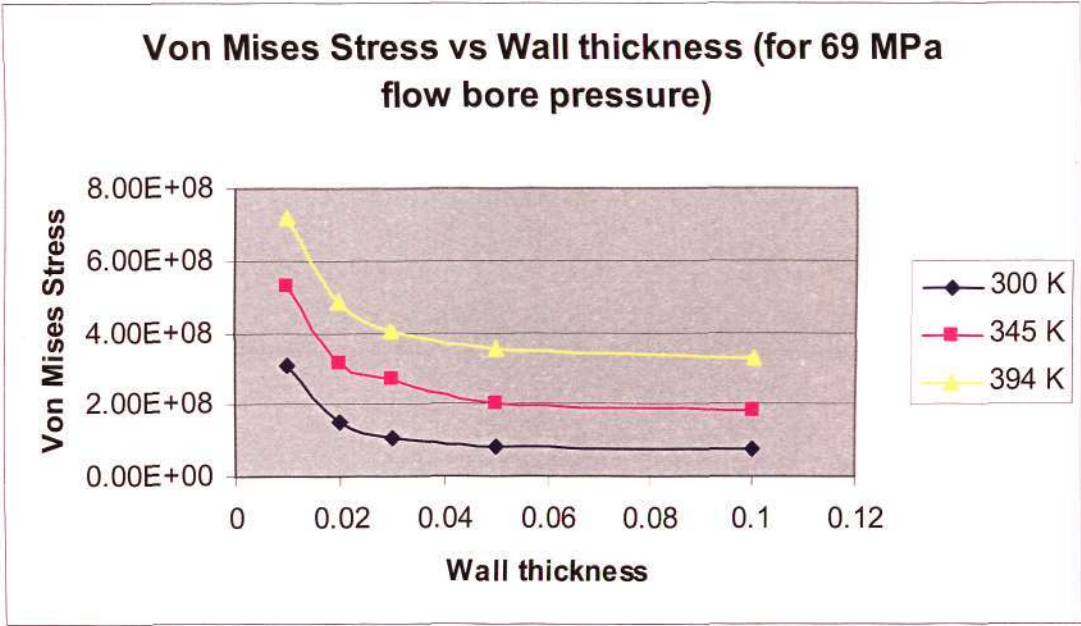




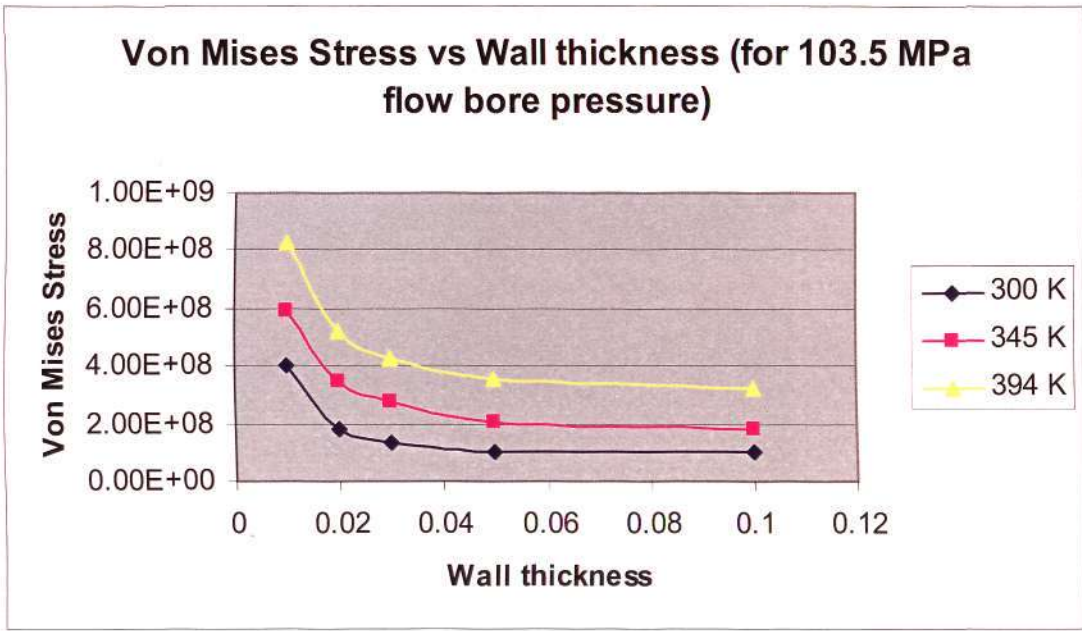
■ Graphs



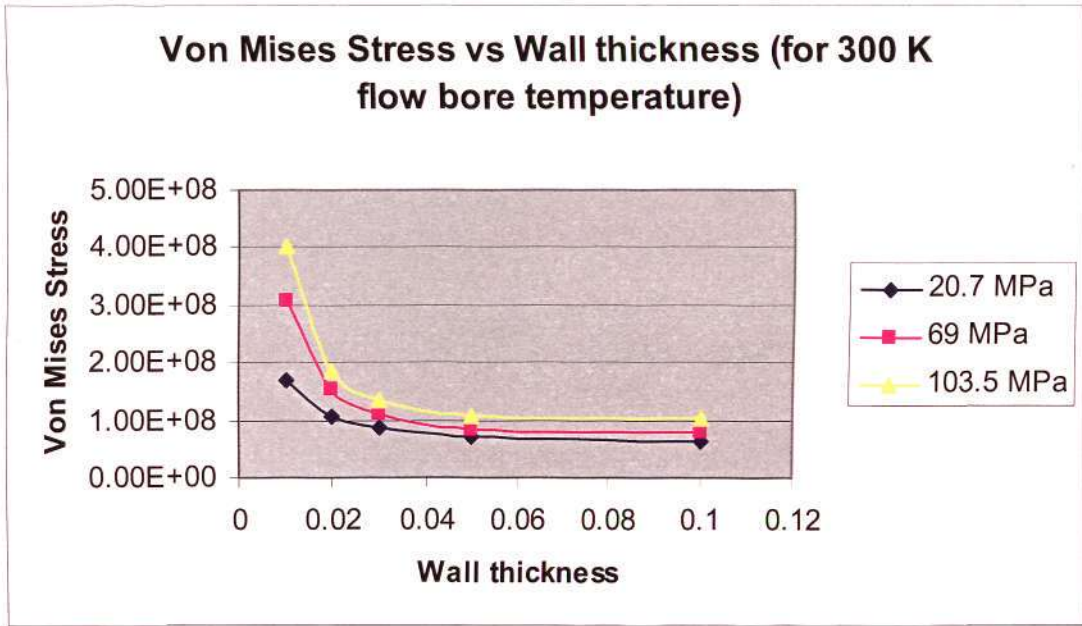
Graph 5.8 Variation of Von Mises Stress vs Wall thickness (for 20.7 MPa flow bore pressure)



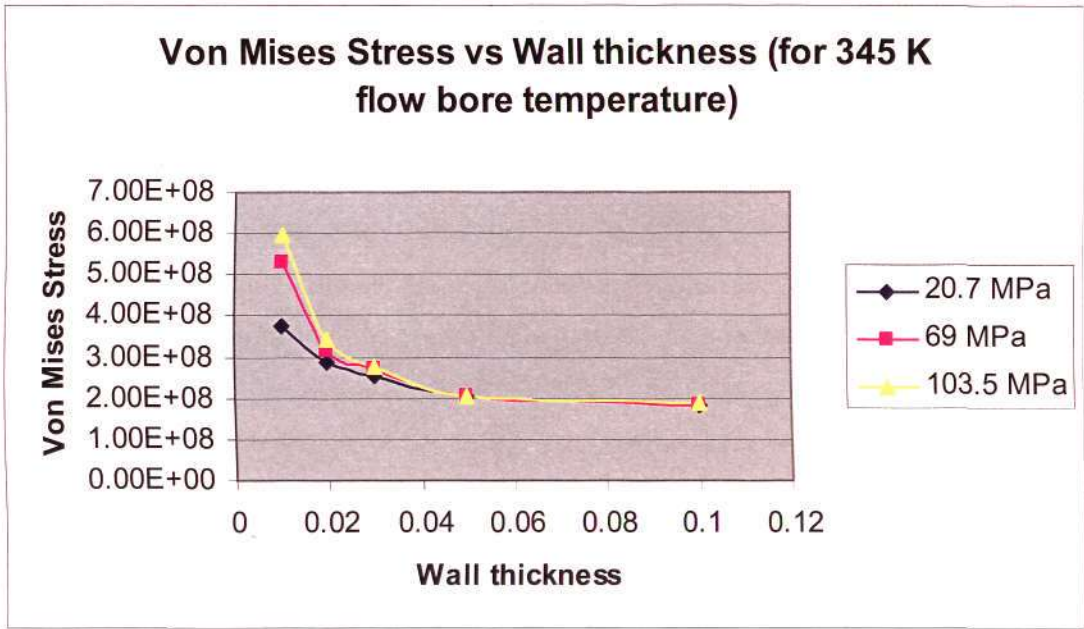
Graph 5.9 Variation of Von Mises Stress vs Wall thickness (for 69 MPa flow bore pressure)



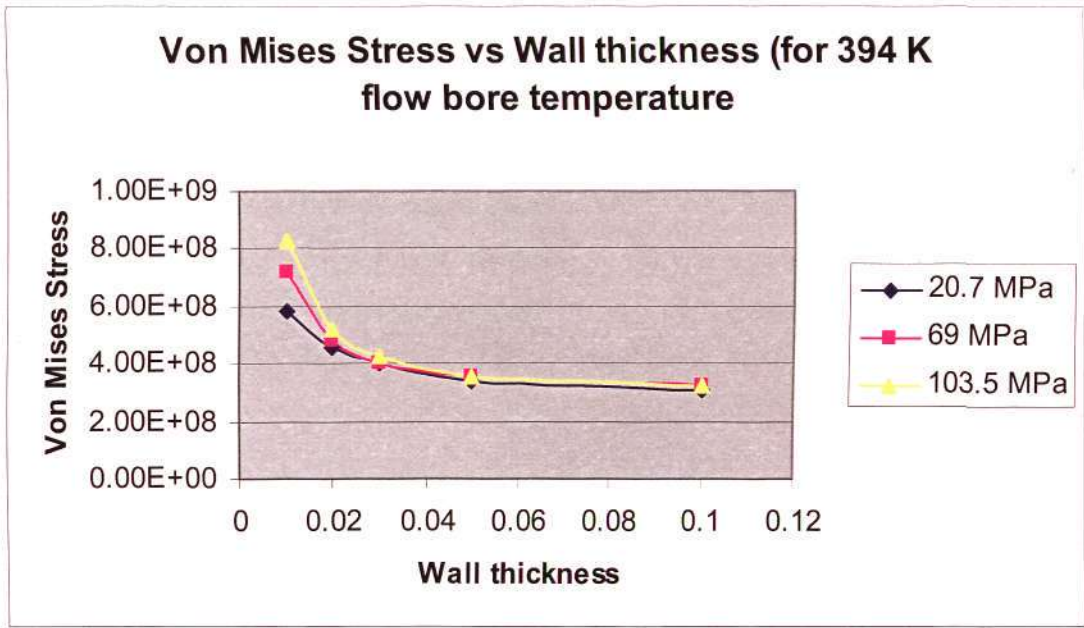
Graph 5.10 Variation of Von Mises Stress vs Wall thickness (for 103.5 MPa flow bore pressure)



Graph 5.11 Variation of Von Mises Stress vs Wall thickness (for 300K flow bore temperature)



Graph 5.12 Variation of Von Mises Stress vs Wall thickness (for 345K flow bore temperature)



Graph 5.13 Variation of Von Mises Stress vs Wall thickness (for 394K flow bore temperature)





## ▪ Discussion

The three main variables that change are the internal temperatures and pressures in the flow bore and cavity and the wall thickness.

### Effect of internal temperature

The effect of temperature on stress can be investigated by keeping the internal pressures of the flow bore and cavity and wall thickness fixed. From Graphs 5.8, 5.9 and 5.10, it was observed that for each wall thickness, the Von Mises stress increased when the internal temperatures increased from 300K to 394K. This can be explained by the increase in thermal stresses when the temperature increased, leading to overall higher stresses throughout the block valve. When the internal temperature of the flow bore and cavity increases, the temperature gradient between the interior and the exterior of the block valve increases, giving a higher temperature difference. This in turn will give rise to a higher thermal strain and thermal stress.

### Effect of internal pressure

The effect of internal pressure on stress can be investigated by keeping the internal temperatures of the flow bore and cavity and wall thickness fixed. From Graphs 5.11, 5.12 and 5.13, it was observed that for each wall thickness, the Von Mises stress increased when the internal pressures increased from 20.7 MPa to 103.5 MPa. When internal pressure increases, a greater compressive force is exerted on the flow bore and seat, resulting in higher Von Mises stresses.

### Effect of wall thickness

The effect of wall thickness between the bores on stress can be investigated by keeping the internal temperatures and pressures of the two bores fixed. From Graphs 5.8 to 5.13, the curves were all sloping downwards. It was observed that as the wall thickness increased, the Von Mises stress decreased, regardless of the internal pressure and temperature.



6. Fracture Mechanics approach to analyze As-clad valves

Cooper Cameron valves have a fillet weld looking as-clad geometry (refer to Fig 6.1), the result of welding overlays, where the cavity bore meets the bottom of the cavity. These bead welds are then sent for grinding processes to smooth the surface and remove the surface defects. Such processes take up time and added costs, leading to an increased lead time and cost quoted to customers. Also, if the grinding process is not controlled tightly, possibility of scrapped castings is high and these will impact the overall production.

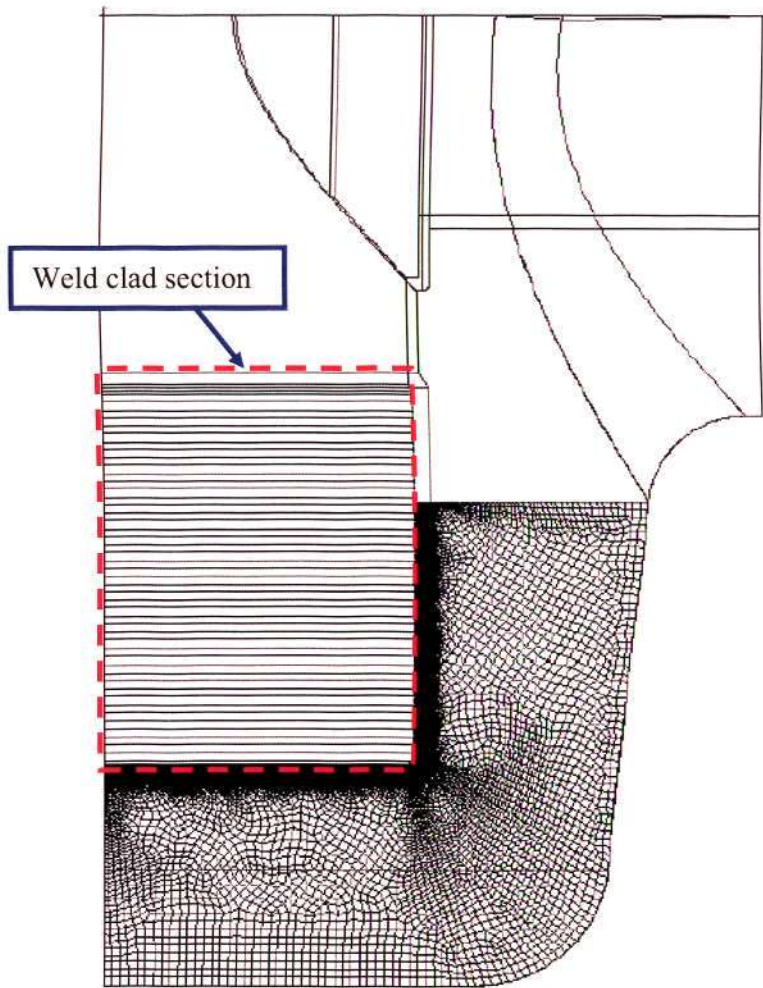


Figure 6.1 Symmetrical view of As-clad valve





The objective of this chapter is to determine the effect of welds in the As-Clad valves using fracture mechanics approach. If the presence of the welds does not initiate any crack propagation through the wall thickness of the valve, no grinding process is required and valuable manufacturing cost and lead time is reduced.

The interfaces between the welds were simulated as cracks to analyze their behavior in terms of crack propagation. Individual displacements and deformed shapes on the cracks were observed. Stress intensity factors for Mode I, Mode II and Mode III fracture were obtained from the ANSYS outputs and compared with the fracture toughness of the material. If stress intensity factors are lower than the fracture toughness of the material, crack propagation is not initiated.

A combined thermal and structural analysis was carried out on all the models. Individual thermal stress analysis or structural analysis was also performed. The sequentially-coupled physics analysis approach using physics environments is used to solve the thermal structural analyses in ANSYS.

The As-clad valve used will be 6-3/8" 10Ksi FLS Gate Valve, made of Inconel 625 material.

In total, 4 different scenarios of loading conditions were investigated and details of the analysis are present here:

#### Effect on cracks

- Scenario 1 – Thermal expansion
- Scenario 2 – structural stresses & no thermal stresses
- Scenario 3 – Both internal pressure and temperature higher than external pressure and temperature
- Scenario 4 – external temperature higher than internal temperature and internal pressure higher than the external.





### 6.1 Simulation of welds

The study of the effect of weld beads uses the axisymmetric finite element model. The actual geometry of the weld beads shown in Figure 6.2 (a) is also simplified to Figure 6.2 (b). This facilitates the modeling process and the end results are not compromised. The detail of the crack tip is shown in Figure 6.3.

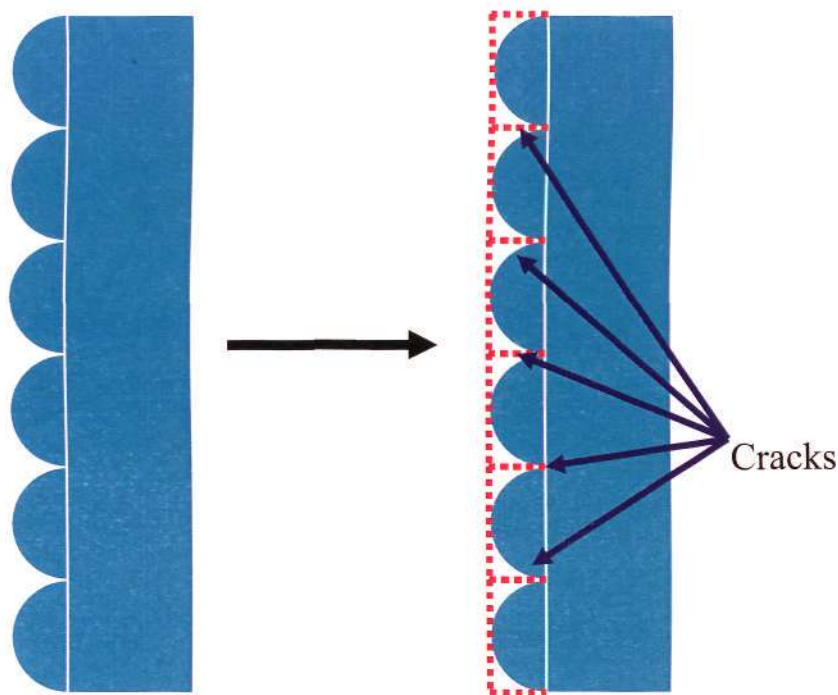


Figure 6.2 (a) Actual geometry

Figure 6.2 (b) Simplified geometry

Figure 6.2 shows a simplified model of the wall thickness with the weld beads clad uniformly along the interior of the cavity.

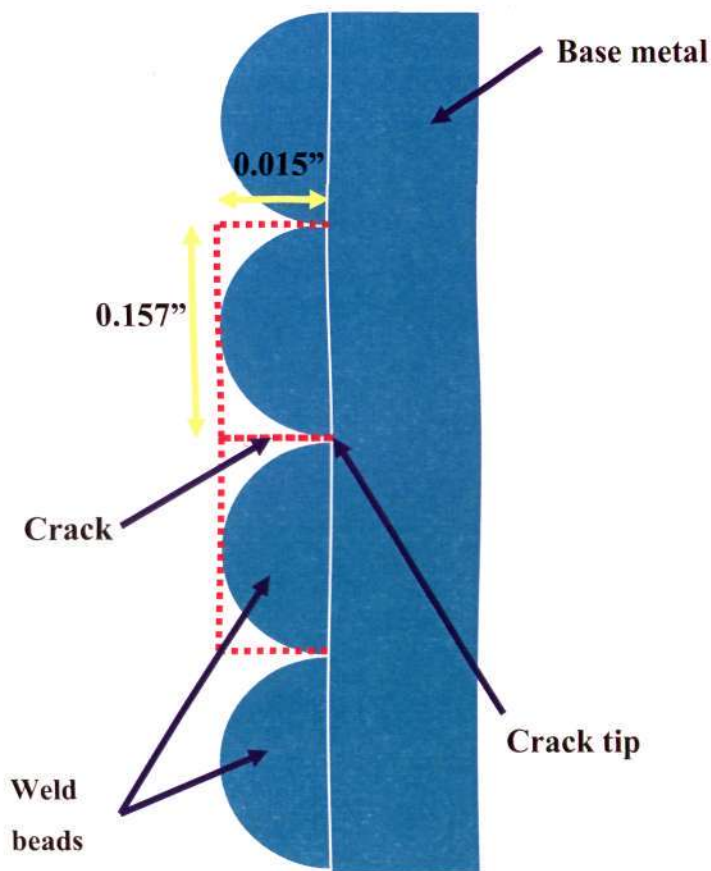


Figure 6.3 Crack simulation

The crack faces can be easily simulated as 2 straight parallel lines instead of the actual 'semicircular' profiles of the weld bead. The areas enclosed by the red dotted lines are added to achieve this. In the event of the cracks initiating (opening up), these 'added' areas do not take any loading against each other. The weld metal is laid and the crack tip is initiating between the weld passes. From the weld bead geometry, the weld beads are about 0.157" wide and 0.015" from peak to valley of the bead.

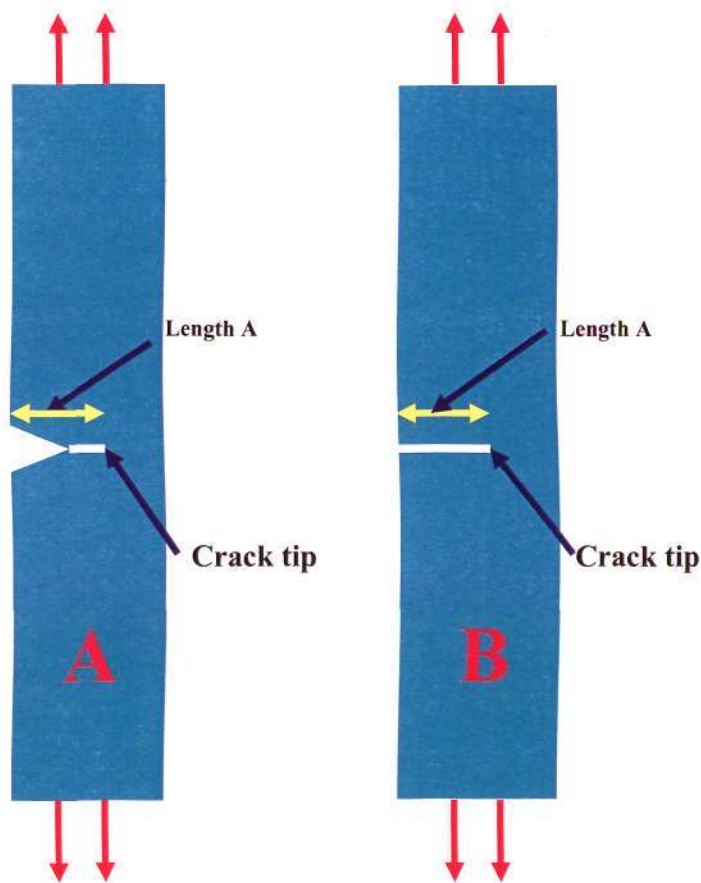


Figure 6.4 Comparison of the actual with simplified model

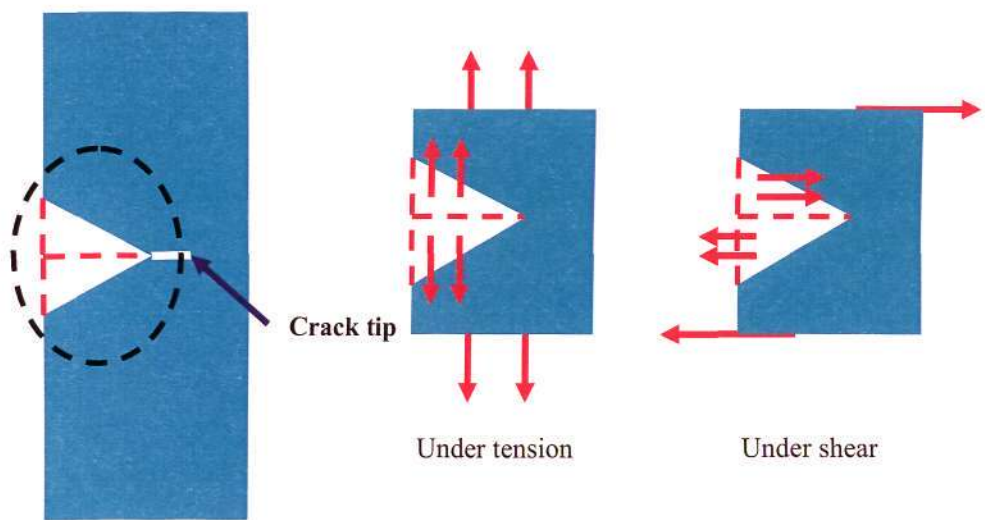


Figure 6.5 Effects of tension and shearing





From both models above, we can see that the added areas do not exert any force or pressure on each other.

To prove that the assumption of the simplified welds is valid, a simple analysis was carried out using ANSYS. Two models, A and B were created as shown in Figure 6.4 with details shown in Figure 6.5. Model A has a notch coupled with a shorter crack length profile. Model B has no notch with a longer crack length. However, the 'effective' crack length- Length A, was the same for both models.

All other dimensions of both models were kept constant and tension was applied on both ends of the model.

Results show that

$K_I$ , stress intensity factors for A =  $K_I$ , stress intensity factors for B

Since stress intensity factor (mode I) for both models are approximately the same, the assumption is proven correct.



## 6.2 Experimental determination of critical fracture toughness

The critical fracture toughness,  $K_{IC}$ , of Inconel 625 is used as criteria to determine crack propagation for the As-clad valve. To obtain this value,  $J_{IC}$ , J integral toughness tests were first performed on standard 3 point bend specimens according to ASTM E1820. The experimentally determined  $J_{IC}$  is then correlated back to obtain  $K_{IC}$ .

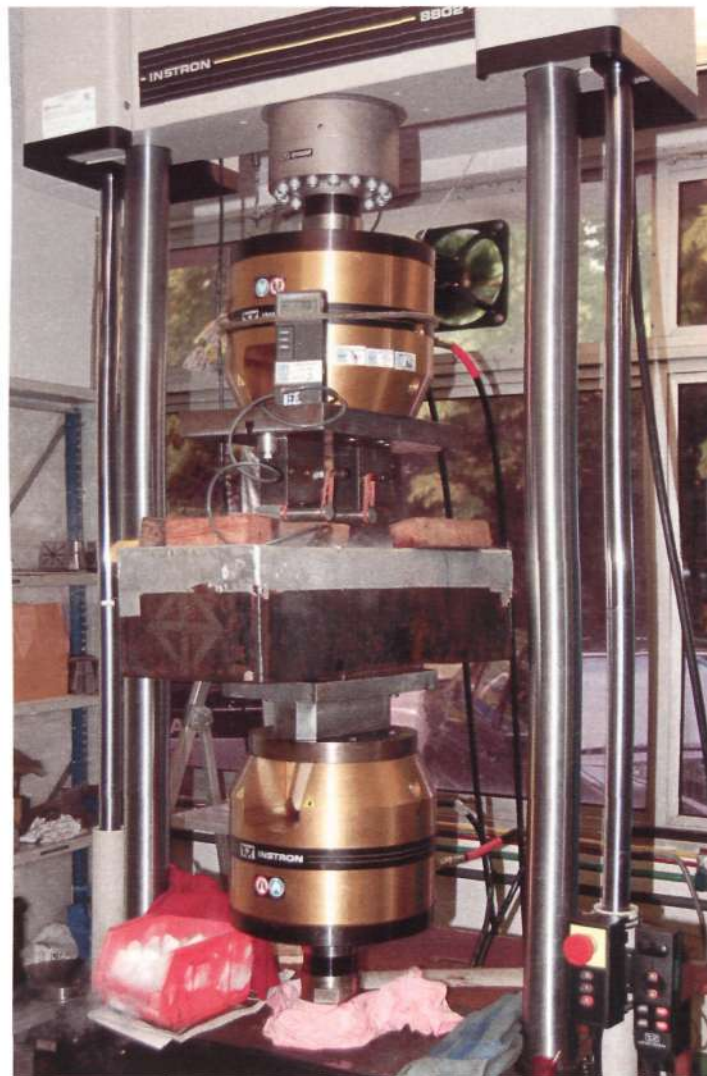
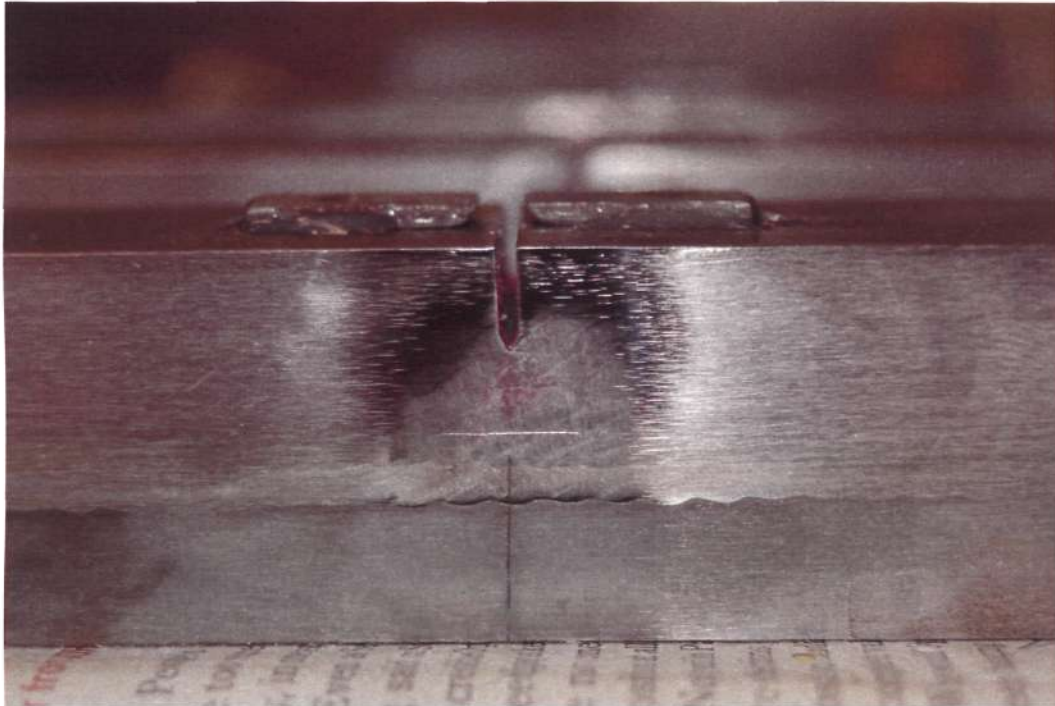


Figure 6.6 Experimental setup



Figure 6.6 shows the setup to carry out the experiment to determine  $J_{IC}$  and Figure 6.7 illustrates the notch in the testing specimen.



**Figure 6.7 Specimen used in  $J_{IC}$  testing**

*Note: Due to confidentiality reasons and money invested by Cooper Cameron, the vendor employed to carry out this experiment and details of other data cannot be revealed in this report.*

*The experimentally determined value of  $K_{IC}$  for Inconel 625 is  $376 \text{ MPam}^{1/2}$*





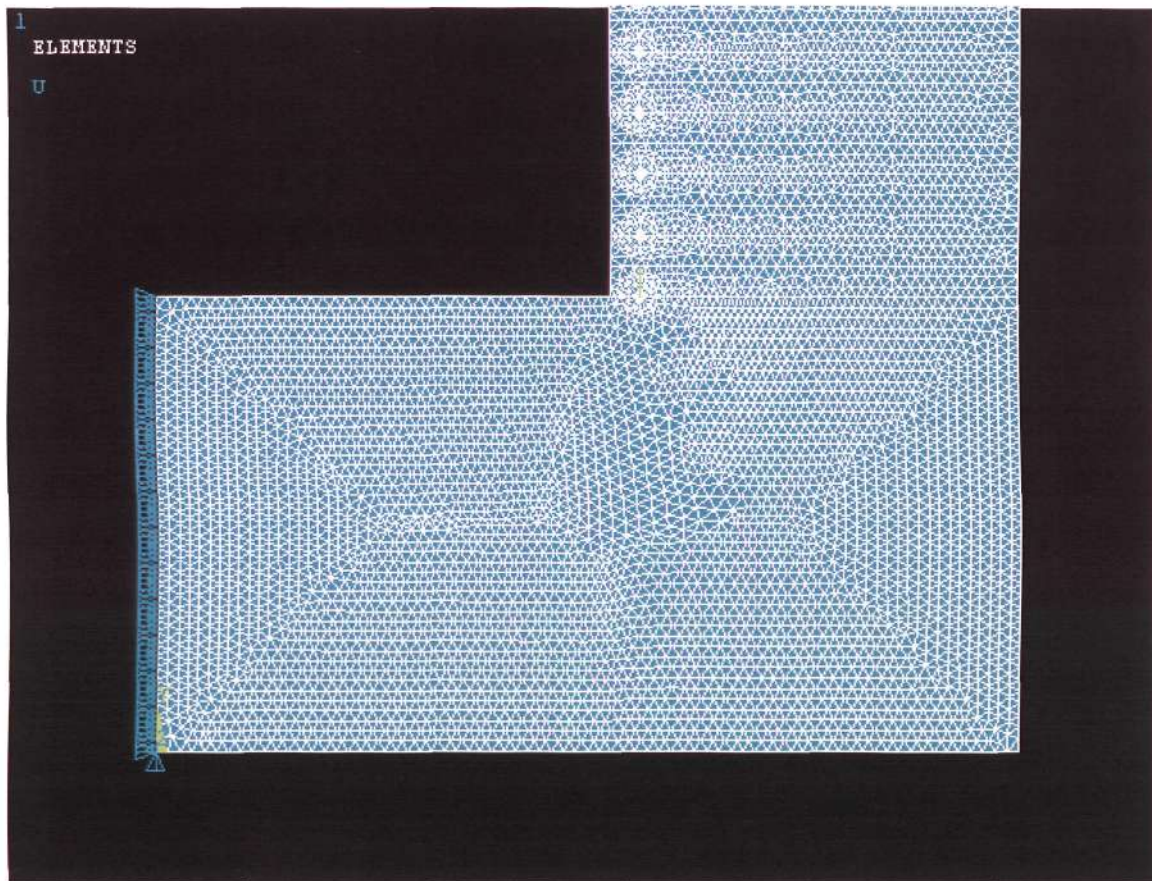
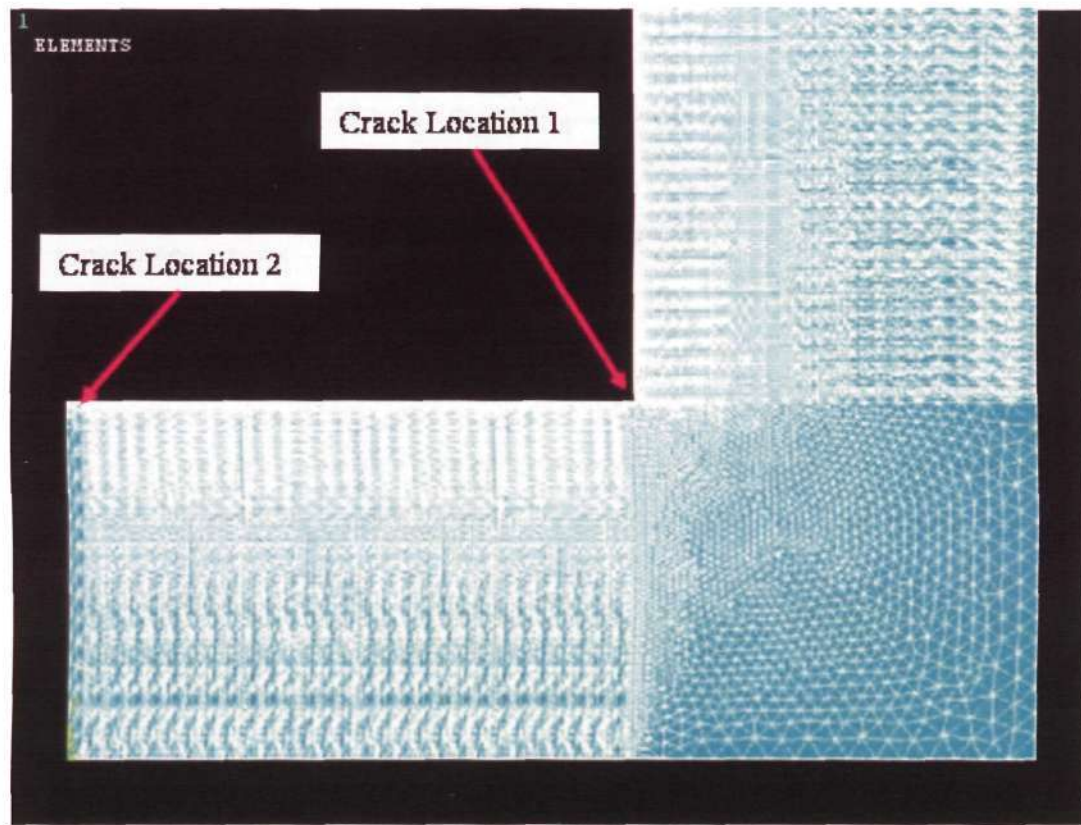


Figure 6.9 Boundary constraints due to axisymmetry

### Assumptions

- The effect of bores are not considered and thus not modeled.
- A linear FEA solution is assumed with an elastic material response for the model.
- The block valve is free to expand on all sides and not under any gravitational force.
- The effect of bolts or drill holes on the stress distribution is ignored.
- The material is assumed to be homogeneous.
- The coefficient of thermal expansion is assumed to be constant throughout.





**Figure 6.10 Crack locations**

For each scenario, 2 crack locations with possible critical SIF were analysed. One is at the corner of the base of cavity and the other is in the middle of the cavity base.

## 6.4 Fracture mechanics FEA model results & discussion

### Scenario 1

This scenario illustrates the effect of thermal stresses on crack behaviour. Internal pressure in the cavity and external pressure were kept constant.





Input parameters

Parameters	Units	Value
Young Modulus	Pa (Psi)	= 2.075 E+11 (3.00958 E+07)
Poisson Ratio		= 0.3
Thermal conductivity	W/m K (BTU-in/hr-ft²-°F)	= 12.5 (87)
Coefficient of Thermal expansion	µm/m-K (µin/in-°F)	= 13.1 (7.3)
External Pressure	Pa (Psi)	= 0
Pressure in cavity	Pa (Psi)	= 0
Cavity temperature	K (°F)	= 394 (250)
External temperature	K (°F)	= 227.6 (-50)
Reference temperature	K (°F)	= 227.6 (-50)

Table 6.1      Input parameters for Scenario 1

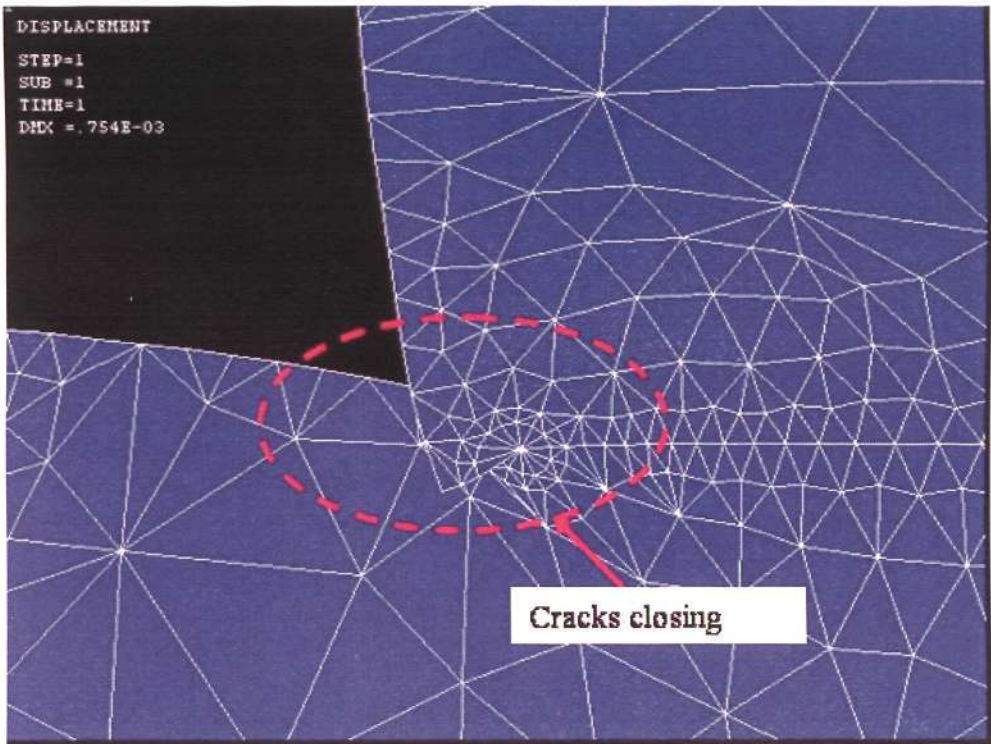


Figure 6.11      Scenario 1 location 1



Stress intensity factors

Stress intensity factor	Value
Mode I	= 37.220 (MPam <sup>1/2</sup> )
Mode II	= 9.5012 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.2      Stress intensity values for Scenario 1 location 1

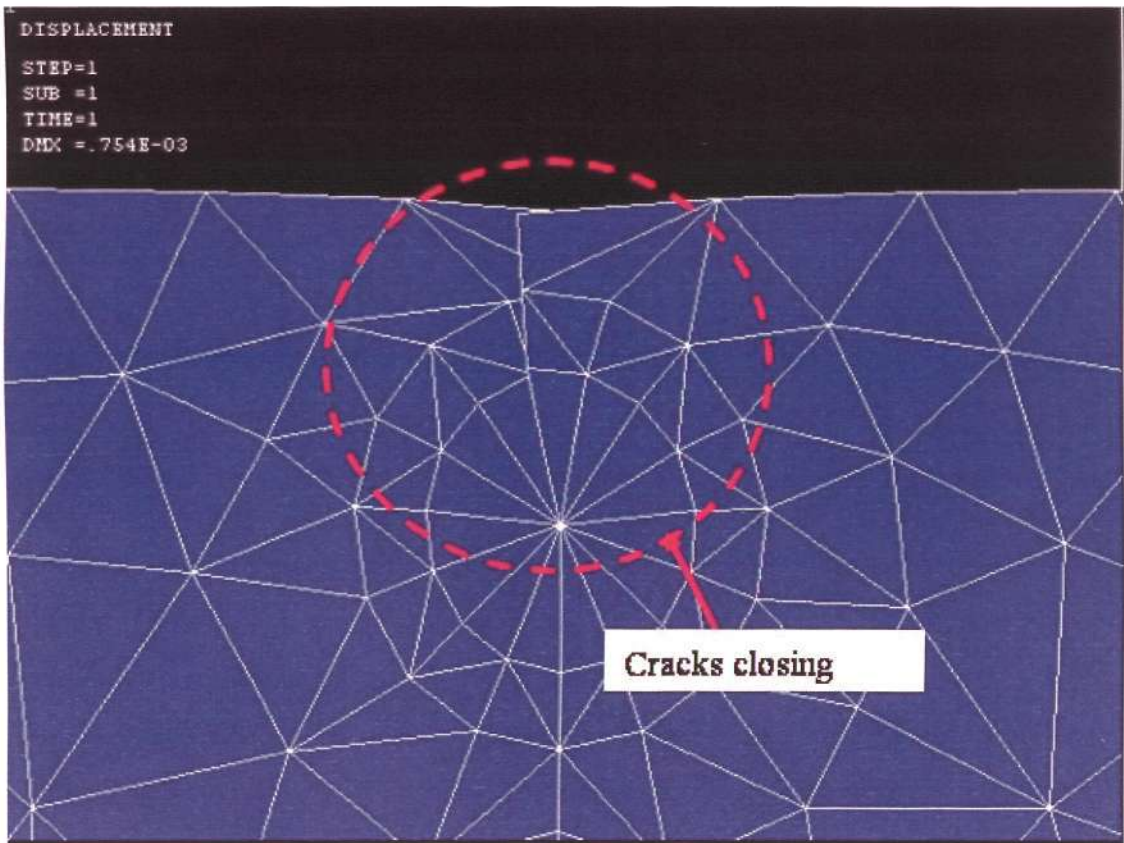


Figure 6.12      Scenario 1 location 2



Stress intensity factors

Stress intensity factor	Value
Mode I	= 11.710 (MPam <sup>1/2</sup> )
Mode II	= 0.12648 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.3      Stress intensity values for Scenario 1 location 2

Although the deformed shape shows the cracks closing up (not to scale), positive stress intensity factors were obtained. This can be explained by the following 3 equations (SIF) for the 3 modes of fracture.

$$K_I = \sqrt{2\pi} \frac{G}{1 + \kappa} \frac{|\Delta v|}{\sqrt{r}}$$

$$K_{II} = \sqrt{2\pi} \frac{G}{1 + \kappa} \frac{|\Delta u|}{\sqrt{r}}$$

$$K_{III} = \sqrt{2\pi} \frac{G}{1 + \kappa} \frac{|\Delta w|}{\sqrt{r}}$$

u, v, w = displacements in a local Cartesian coordinate system

Since there is a modulus on the change in u, v and w displacements, with a negative displacement along the v coordinates, the stress intensity factors displayed will still be positive.

Therefore, the scenario illustrates the importance of checking the displacements and deformed shapes around the crack. If it is evident that the cracks do not close, the stress intensity factors will then be used to compare with the fracture toughness to determine crack initiation.





## Scenario 2

This scenario illustrates the effect of structural stresses on crack behaviour with the absence of thermal stresses. Internal temperature in the cavity and external temperature were kept constant.

The same input parameters were used in Scenario 1 except that the pressure in the cavity is not zero but 69000000 Pa (10000 Psi) and the cavity temperature is equal to the external temperature at 227.6 K (-50°F).

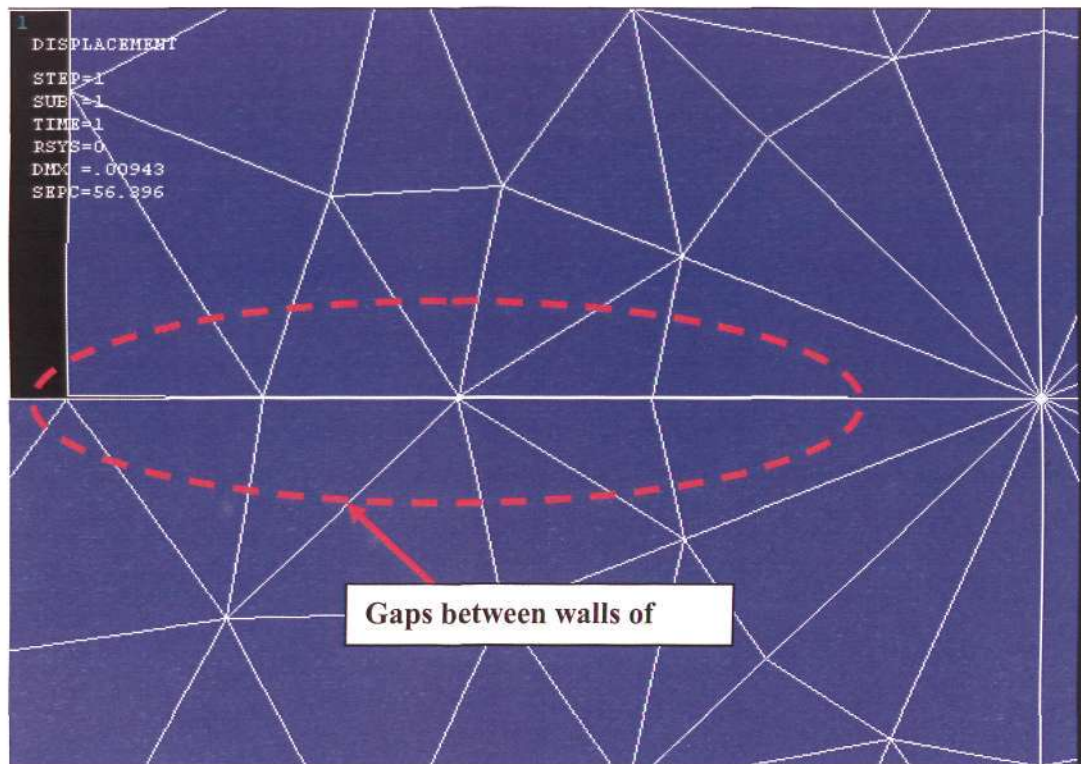


Figure 6.13 Scenario 2 location 1



Stress intensity factors

Stress intensity factor	Value
Mode I	= 2.0448 (MPam <sup>1/2</sup> )
Mode II	= 0.53653 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.4      Stress intensity values for Scenario 2 location 1

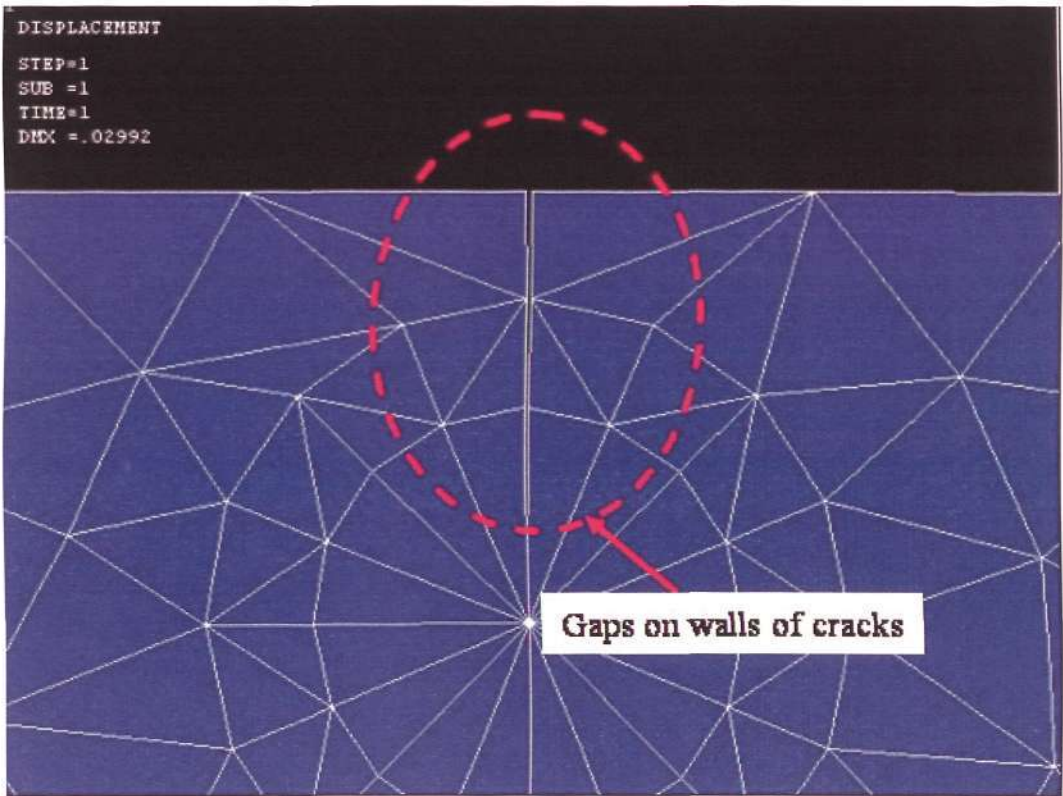


Figure 6.14      Scenario 2 location 2

Stress intensity factors

Stress intensity factor	Value
Mode I	= 21.599 (MPam <sup>1/2</sup> )
Mode II	= 0.26565 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.5      Stress intensity values for Scenario 2 location 2





### Comment

The plot of the deformed shape and displacements shows that there is a slight gap between the walls of the crack (not closing up). In this case, the stress intensity factors were valid for comparison with the fracture toughness.

### Scenario 3

This scenario illustrates the effect of thermal and structural stresses on crack behaviour. Both internal pressure and temperature are higher than the external pressure and temperature

The same input parameters were used in Scenario 2 except that the cavity temperature is changed to 394K (250°F).

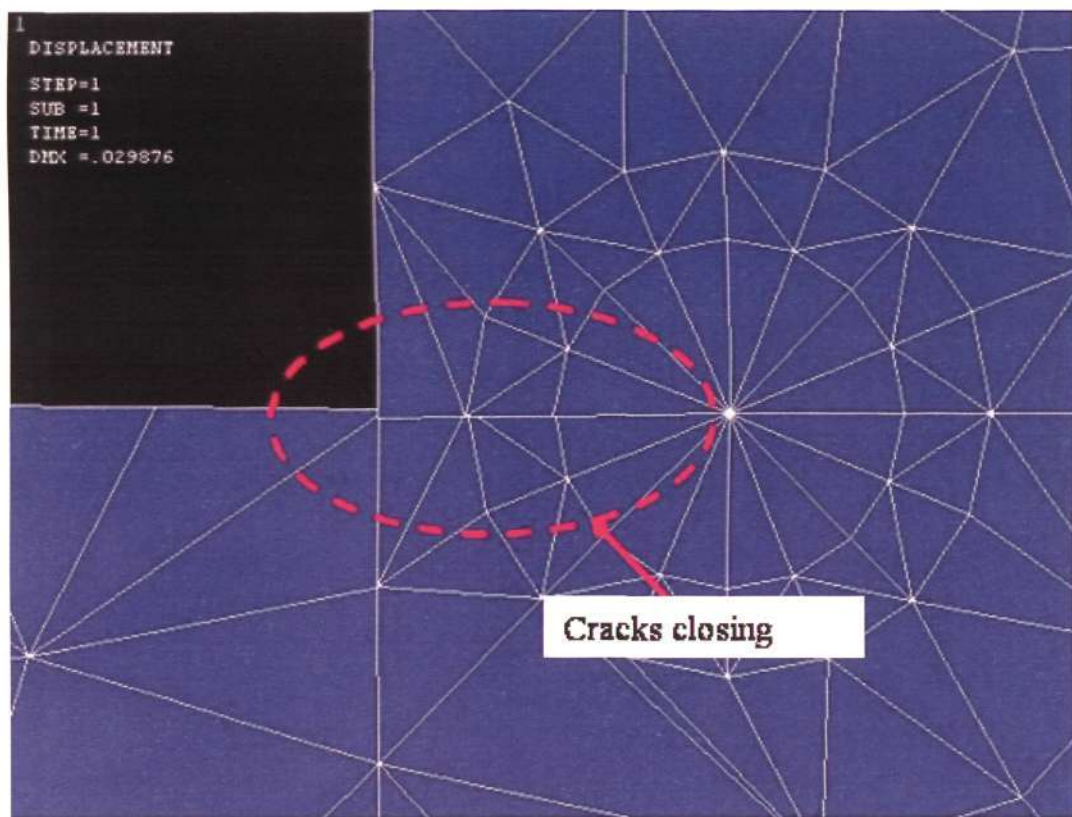


Figure 6.15 Scenario 3 location 1





Stress intensity factors

Stress intensity factor	Value
Mode I	= 35.176 (MPam <sup>1/2</sup> )
Mode II	= 8.9646 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.6      Stress intensity values for Scenario 3 location 1

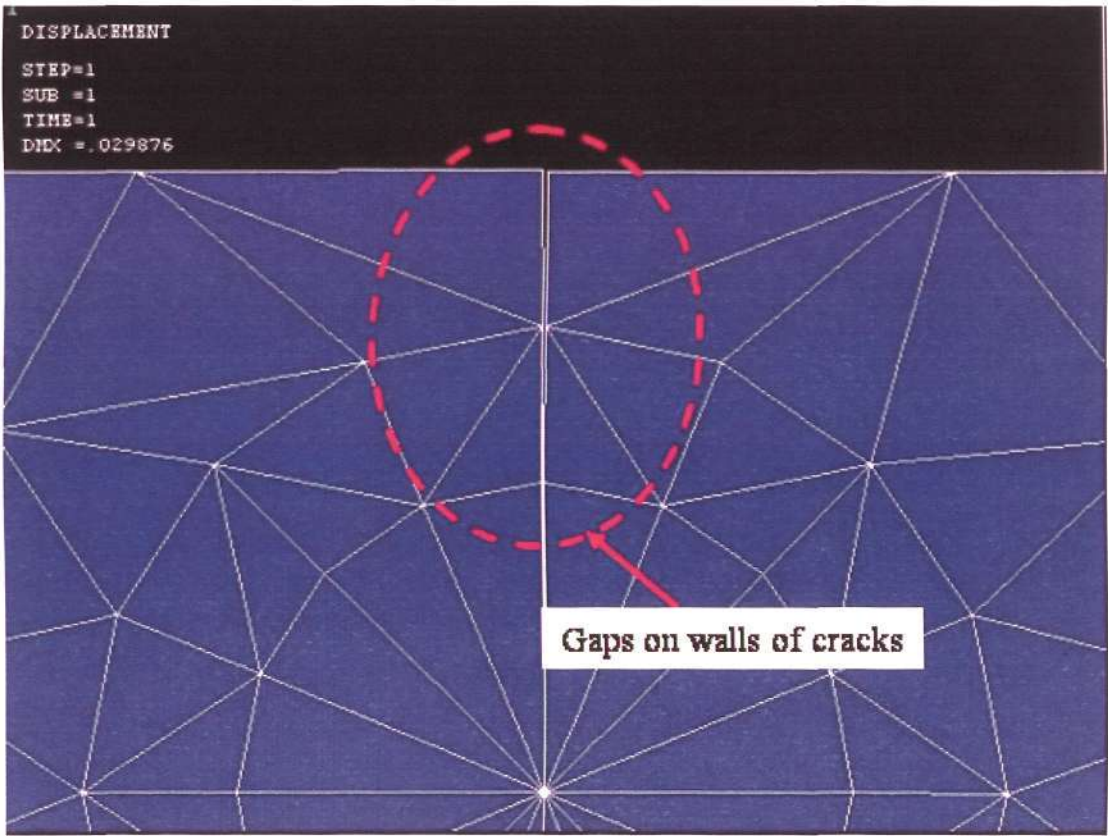


Figure 6.16      Scenario 3 location 2

Stress intensity factors

Stress intensity factor	Value
Mode I	= 9.8899 (MPam <sup>1/2</sup> )
Mode II	= 0.13917 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.7      Stress intensity values for Scenario 3 location 2

**Comment**

Since the plot of the deformed shape and displacements shows that the crack at location 2 was opening up, the stress intensity factor here can be used for comparison with the fracture toughness. Crack propagation is not initiated at location 1 due to the gap closing.

**Scenario 4**

This scenario illustrates the effect of thermal and structural stresses on crack behaviour. However, external temperature is higher than the internal temperature and internal pressure is higher than the external. This case may be evident in temperate climates such as deserts where the external temperature is high. When there is a leakage of gas in the pipelines on one side of the valve, it decreases the temperature within the pipe and subsequently the valves rapidly.

The same input parameters were used in Scenario 2 except that the external temperature and reference temperature are both changed to 323.16 K (122°F).

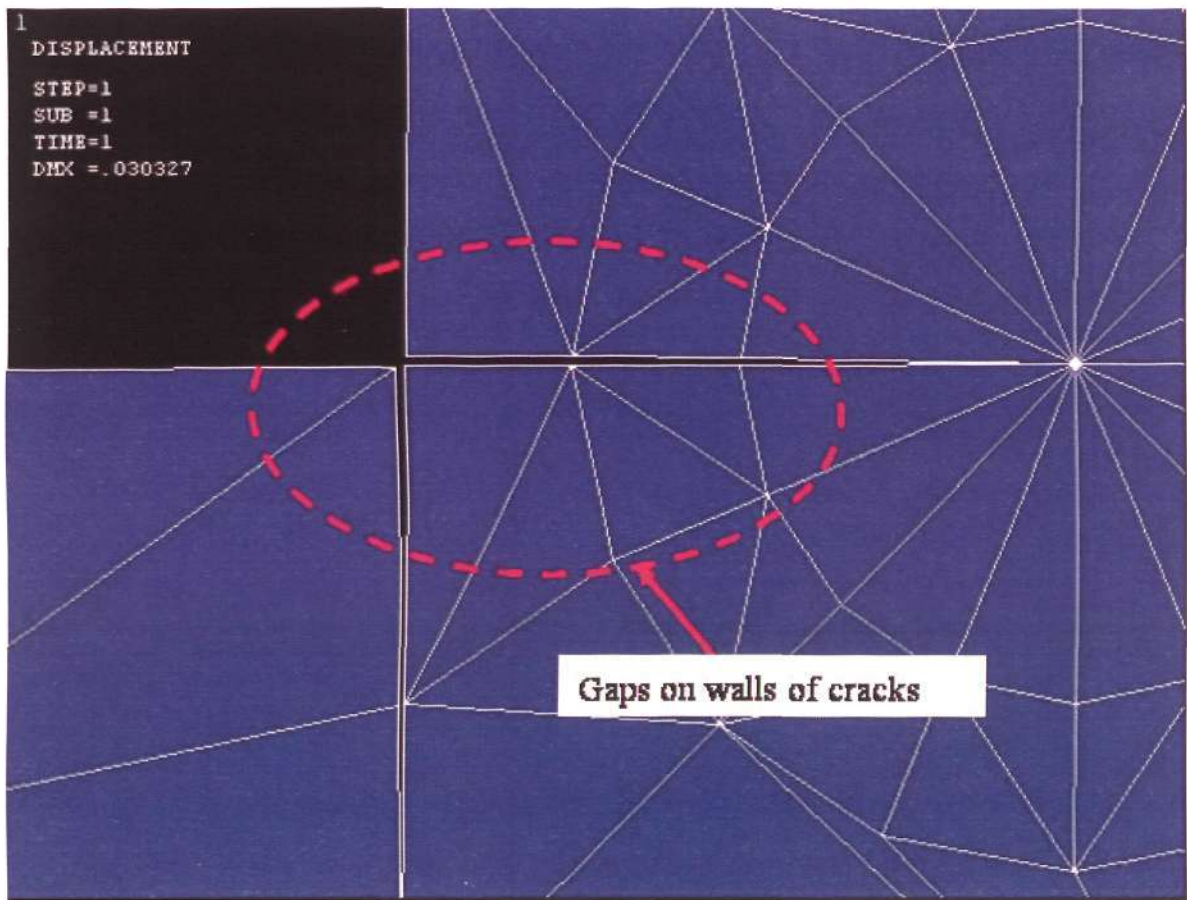


Figure 6.17 Scenario 4 location 1

Stress intensity factors

Stress intensity factor	Value
Mode I	= 23.420 (MPam <sup>1/2</sup> )
Mode II	= 5.9928 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.8 Stress intensity values for Scenario 4 location 1



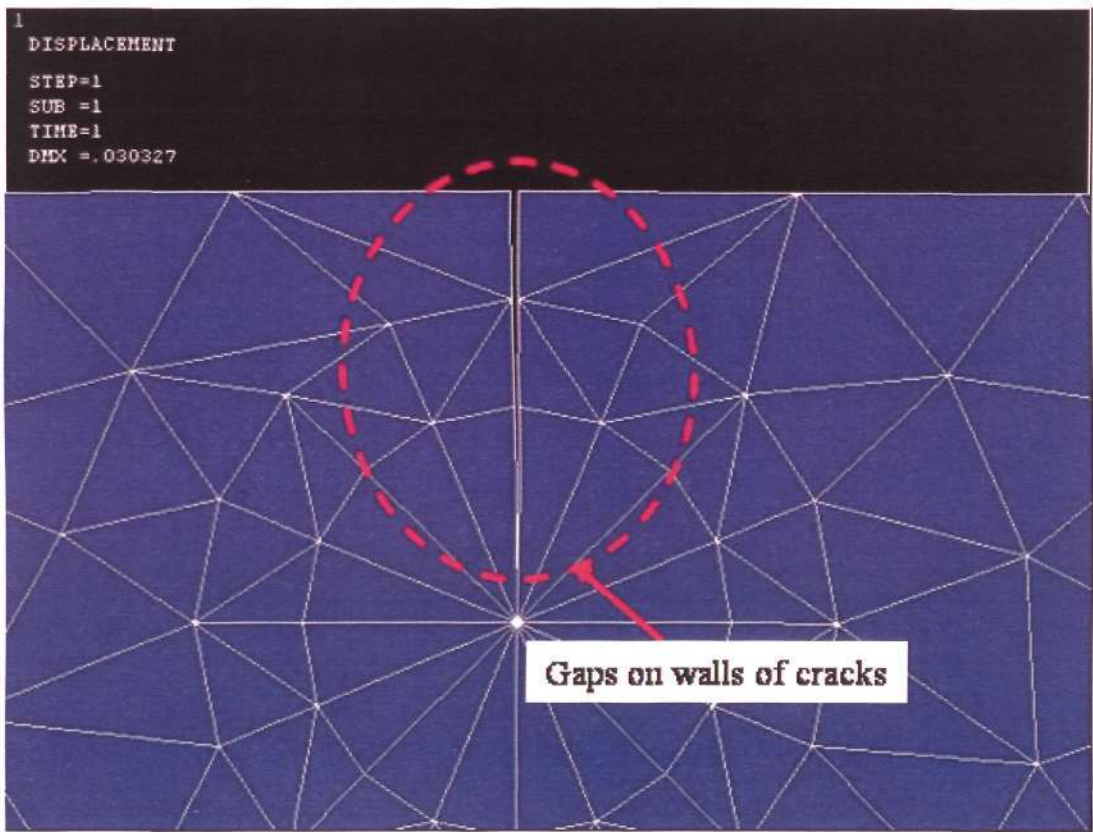


Figure 6.18 Scenario 4 location 2

Stress intensity factors

Stress intensity factor	Value
Mode I	= 28.324 (MPam <sup>1/2</sup> )
Mode II	= 0.33828 (MPam <sup>1/2</sup> )
Mode III	= 0

Table 6.9 Stress intensity values for Scenario 4 location 2



Comment

Since the plot of the deformed shape and displacements shows that the cracks did not close up, the stress intensity factors will be used for comparison with the fracture toughness.

Summary

Case	SIF ( $K_I$ ) (MPam <sup>1/2</sup> )	SIF ( $K_{II}$ ) (MPam <sup>1/2</sup> )	SIF ( $K_{III}$ ) (MPam <sup>1/2</sup> )	Crack close up?	SIF ( $K_I$ ) < $K_{IC}$ (376 MPam <sup>1/2</sup> )?
Scenario 1 Location 1	37.220	9.5012	0	Yes	Yes
Scenario 1 Location 2	11.710	0.12648	0	Yes	Yes
Scenario 2 Location 1	2.0448	0.53653	0	No	Yes
Scenario 2 Location 2	21.599	0.26565	0	No	Yes
Scenario 3 Location 1	35.176	8.9646	0	Yes	Yes
Scenario 3 Location 2	9.8899	0.13917	0	No	Yes
Scenario 4 Location 1	23.420	5.9928	0	No	Yes
Scenario 4 Location 2	28.324	0.33828	0	No	Yes

Table 6.10      Summary of results from different scenarios & locations

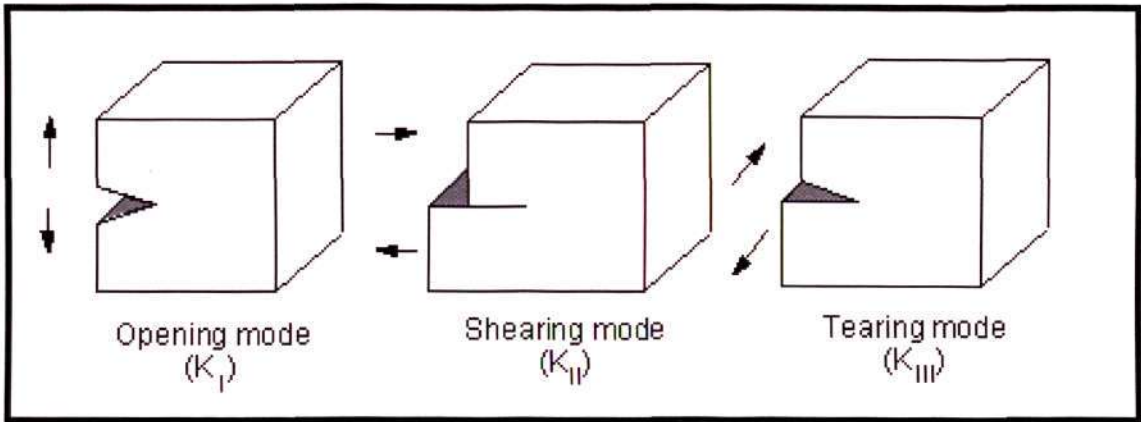


Figure 6.19 Different modes of fracture

Table 6.10 summarizes the stress intensity factors (SIF) obtained at the 2 locations from the 4 scenarios and Figure 6.19 shows the different modes of fracture.

In all the 8 cases, the SIF ( $K_I$ ) for Mode I, tearing, is higher than the SIF ( $K_{II}$ ) for the shearing mode. Comparing SIF ( $K_I$ ) against the critical fracture toughness of Inconel 625, all 8 cases show that  $K_I$  is lower and thus crack propagation is not initiated.

From this, we can conclude that the machining process to remove the weld beads on the As-clad valves is not required.





---

## 7. CONCLUSION

---

This thesis report in fulfillment for Master of Engineering details the design of a new parametric program to evaluate code compliance and as a design tool for block valves. It allows a user to carry out finite element analysis without knowing the basics of FEM or any FEA software. The methodology was followed closely from the training, specifications, up to finite element analysis results and discussions. A literature review was carried out to acquire all relevant information useful to the project. This information was referenced constantly throughout the project. A training stint for the author was conducted at the production facilities to understand the operation, functions and mechanisms of gate and block valves. Regular meetings were held with the R&D manager and engineers from Cooper Cameron to constantly make reviews and improvements to the programs.

The parametric program was called Auto-FEA and includes user friendly options such as menus and access to databases. The nine different sub-programs under it are

- Parallel Bores
- Perpendicular Out of plane Bores
- Perpendicular In plane Bores
- Valve Cavity and Perpendicular intersecting bores (through)
- Valve Cavity and Perpendicular intersecting bores (non-through)
- Valve Cavity & Offset Perpendicular intersecting bores
- Valve Cavity & Angled intersecting bores
- 2 Valve Cavities & Angled intersecting bores
- 2 Valve Cavities & Offset Perpendicular intersecting bores



with each catering to a certain configuration of block valves. These programs have been successfully introduced to the Product Design Department of Cooper Cameron for their usage and review.

Two block valves, Parallel Bores and the Valve Cavity and Perpendicular intersecting bore, were analyzed and parametric studies carried out to study the effects of temperature, pressure and wall thickness on the stresses in the block valve. Generally, when the internal temperatures and pressures of the bores increase, the Von Mises stress would increase. However, there are exceptions to this trend when the compressive stresses on the bores caused by internal pressure cancel out part of the thermal stresses. A consistent observation was made that when wall thickness between bores or between a seat and a bore decreases, the Von Mises stress increase.

Fracture mechanics approach through finite element analysis was used to investigate the possibility of crack propagation between the weld beads on the As-clad valves. Results show that the interface between the weld beads do not propagate as cracks into the wall thickness of the valves. Machining process to remove the protrusions of these weld bead clads are not necessary and this will reduce the manufacturing cost and time greatly.

In summary, the project has achieved its preliminary goal to design a parametric software to carry out finite element analysis and its minor objective to evaluate the possibility of cracking on As-clad valves.



---

## REFERENCES

- [1] ANSI/API Specification 6A: 2001 ,Specification for Wellhead and Christmas Tree Equipment , ISO 10423:2003, (Modified) Petroleum and natural gas industries — Drilling and production equipment —Wellhead and Christmas tree equipment
  
- [2] ASME Boiler & Pressure Vessel Code: 1998, VIII, Division 1, Rules for Construction of Pressure Vessels
  
- [3] ASME Boiler & Pressure Vessel Code: 1998, VIII, Division 2, Alternative rules for Construction of Pressure Vessels
  
- [4] ANSYS 8.0. 2004. User's Manual, Analysis Systems, Inc., Houston, Pa
  
- [5] Cooper Cameron Corporation. Cameron Division.1997.Manually Actuated Cameron FL and FLS Gate Valves, Operation & Maintenance Manual
  
- [6]. PP Benham, RJ Crawford & CG Armstrong. Mechanics of Engineering Materials, 2<sup>nd</sup> edition. Prentice Hall. 1996.