INVESTIGATION ON STRESS CONCENTRATION FACTOR OF PIPING BRANCH JUNCTIONS SUBJECTED TO INTERNAL PRESSURE

By

ALI AZMAN BIN OMAR

FINAL PROJECT REPORT

Submitted to the Mechanical Engineering Programme
in Partial Fulfillment of the Requirements
for the Degree
Bachelor of Engineering (Hons)
(Mechanical Engineering)

Universiti Teknologi Petronas
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

© Copyright 2008
by
Ali Azman bin Omar, 2008

CERTIFICATION OF APPROVAL

INVESTIGATION ON STRESS CONCENTRATION FACTOR OF PIPING BRANCH JUNCTIONS SUBJECTED TO INTERNAL PRESSURE

by

Ali Azman bin Omar

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

Approved:

____

Dr. Khairul Fuad
Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK

June 2008

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

Ali Azman bin Omar

ABSTRACT

Pipe intersections are a regular feature of piping systems and are often the subject of multiple loads, usually acting simultaneously and at irregular intervals. For some time it has been understood that when material is removed from a run pipe in order to accommodate, for example, a branch pipe, the opening in that vessel promotes increased stresses around the edge of the hole. The objective of this project is to investigate the stress concentration factor of piping branch junction resulting from internal pressure using ANSYS. Three different angle of pipe that is 45^o, 60^o and 90^o with different ratios of pipe diameter are being studied. The three-dimensional plane stress analysis will be conducted and the von Mises stress is taken into account to determine the stress concentration factor and to analyze the stress distribution at the adjacent pipe hole where the maximum stress are being observed. Reports would be done after each work has been done. In this report, there are some discussions made base on result obtain from the simulation of ANSYS. To be concluded, the effect of different angle and dimension ratio of pipe on the stress concentration factor of the piping branch junction would give a good analysis to enhance the current application of the branch system.

ACKNOWLEDGEMENTS

Firstly the author would like to praise Allah the Almighty for giving him the strength in completing this Final Year Project as part of fulfillment for the study in Universiti Teknologi Petronas.

A special thank and appreciation to the author's supervisor, Dr. Khairul Fuad for the guidance and supervision through out the whole project development process. Without his support and advice, it would be difficult for the author to complete this project successfully. Working under his supervision had been an interesting experience for the author.

The author would also like to express his warmest and greatest gratitude for all the parties that have contributed whether directly or indirectly towards the success of this project. Special thanks to friends and family for their supports.

Lastly, a special dedication to Universiti Teknologi Petronas (UTP), especially the Mechanical Engineering Department that have contributed so much in the development of essential skills in order to achieve excellence in theoretical and practical works.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

ANSYS- Analysis System

ASME- American Society of Mechanical Engineers

AUTOCAD- Auto Computer Aided Design

DBL EH- Double extra heavy

DIN- Deutsches Institut für Normung eV (German Institute for Standardization; similar to US ANSI)

d/D- Ratio of branch pipe diameter (*d)* to header pipe diameter (*D*)

EH-Extra heavy

FEA- Finite Element Method

JIS- Japanese Industrial Standards

kpsi- Kilo Pounds per Square Inch

NB- Nominal bore

NPS- Nominal pipe size

psi- Pounds per Square Inch

SCH- Schedule

STD- Standard

3-D- Three Dimension

CHAPTER 1
INTRODUCTION

1.1 Background of Study

Pipeline fittings are common engineering components used in various technological applications, among others in oil platforms and in nuclear power generating plants. Their integrity assessment is a key point in safe guarding the pressurized piping system. Until now, there is very little data for cracked branch junctions available as literature. Pipe fittings are used when the other connection is needed to be form from the main pipe. The use of pipe flange connections is standardized in the codes of JIS, ASME, DIN and so on. However, these codes are almost entirely dependent on experience, and subsequently some problems concerning pipe flange connections have been encountered.

For some time it has been understood that when material is removed from a pressure vessel or run pipe in order to accommodate, for example, a branch pipe, the opening in that vessel promotes increased stresses around the edge of the hole. In an effort to compensate for the inherent weakness at the hole, piping engineers may choose to fit a reinforced branch outlet or nozzle that is designed in such a way as to provide material compensation in the area immediately surrounding the hole in the run pipe.

Pipe intersections are a regular feature of piping and pressure vessel systems and are often the subject of multiple loads, usually acting simultaneously and at irregular intervals. Due to the nature and complexity of the loading, the subject has received a significant amount of study from designers and stress analysts, certainly over the last 50 years, in an effort to resolve some of the difficulties of stressing pressure structures. Many pressure vessel and piping codes have adopted their own approach to the stress analysis of piping branch junctions.

1.2 Problem Statement

Pipe fittings experience a significance effect of stress concentration due to internal fluid pressure. The stress concentration is defined as the ratio of the maximum stress that seems in the pipe fitting to the Von Mises stress without branch. The maximum Von Mises stress depends on the angle of pipe branch.

This project would be emphasizing the study of stress concentration factor at piping branch junctions. Simulation of the pipe fitting would be done using ANSYS.

1.3 Objectives and Scope of Study

1.3.1 Objectives of the project:

- To analyze and investigates the stress concentration factor of piping branch junction resulting from internal pressure using ANSYS.
- To study the result and effect of different angle of junction in order to recommend an improved design of the pipe fitting.

1.3.2 Scope of Study:

Initially, the characterizations and specifications of the various angle of pipe fitting is done. These specifications are needed in order to declare standards size of pipe fitting. The specifications included the properties of the pipe fitting such as material used, size of the pipe fitting and the specific angle of the fitting. Analysis would be made after the simulation done using computer software, ANSYS.

CHAPTER 2

LITERATURE REVIEW AND THEORY

2.1 Literature Review

There are some study has been made on regarding topic that can be made as review of information. Some managed to be a good reference whiles other not. Below is the literature review that has been referred.

1. **Effective stress factors for reinforced butt-welded branch outlets subjected to internal pressure or external moment loads by J. P. Finlay , G. Rothwell, R. English and R. K. Montgomery**

   This paper presents finite element data for 92 reinforced butt-welded branch outlet piping junctions designed according to the ASME B31.3 process piping code, for the purpose of investigating their effectiveness in the light of data for un-reinforced fabricated tee junctions. The data suggest that the reinforcement provided under the ASME B31.3 design is effective for the internal pressure load case and all external bending moment loads with the exception of branch out-of-plane bending for thin-walled assemblies.

2. **Thermoelastic Stress Analysis of Composite Piping Component**

   N. Sathon and J.M. Barton have made a study to find measurements of the full-field stress distribution in a component. It is necessary to identify the potential failure location. Various non-contact approaches are available such as photoelasticity, shearography, optical holographic, etc. These have proved effective for design purposes, particularly for the relatively rapid developed of prototypes in terms of design and new materials.

2.2 Theory of Application

2.2.1 Structural Analysis

Structural analysis is the computation of deflections, and internal forces or stresses within structures, either for design or for performance evaluation of existing structure. Structural analysis needs input data such as the applied forces or environmental effects, the structure's geometry and support conditions, and the material's properties. Output quantities may include support reactions, stresses and displacements. In design, the calculated stresses are compared to the allowable stresses for the materials used while the calculated displacements are compared to various standards for serviceability.

There are two broad classes of analysis that are classical methods and matrix methods. Classical methods provide answers by means of analytical formulation, applicable mostly for simple structural models, while matrix methods are computer-oriented, applicable to structures of arbitrary size and complexity. Both approaches however are based on the same three fundamental relations that are equilibrium, constitutive and compatibility. The solutions are approximate when any of these relations are only approximately satisfied. Classical methods are available for individual members such as beams, columns, shafts and for the entire structures such as trusses, frames, plates and shells. They are still used for small structures and for preliminary design of large structures. Matrix methods model a structure as an assembly of elements or components with various forms of connection between them. Early application of matrix methods were for articulated frameworks with truss, beam and column elements; later and more advanced matrix methods, usually referred to as finite element model an entire structure with one, two and three dimensional elements and can be used for articulated as well as continuous structures such as a pipe, plates and shells.

2.2.2 Finite Element Analysis (FEA)

Finite element analysis (FEA) or finite element method (FEM) is a numerical technique for solution of boundary-value problems. It was first developed for use in structure analysis. In its application, the object or system is represented by a geometrically similar model consisting of multiple, linked, simplified representations of discrete regions, i.e., finite elements.

Equations of equilibrium, in conjunction with applicable physical considerations such as compatibility and constitutive relations are applied to each element, and a system of simultaneous equations is constructed. The system of equations is solved for unknown values using the techniques of linear algebra or nonlinear numerical schemes, as appropriate. While being an appropriate method, the accuracy of the FEA method can be improved by refining the mesh using more elements and nodes.

2.2.3 Concept of Stress

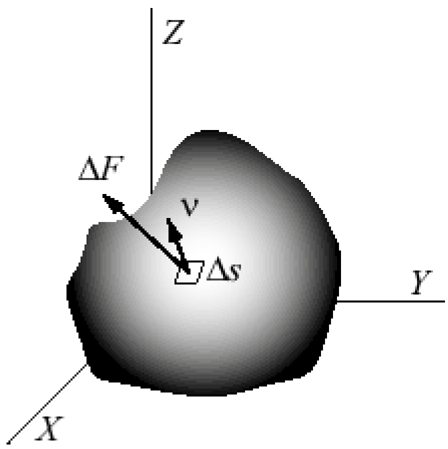


Figure 2 Representation of a 3D model

First, we look at the external traction T that represents the force per unit area acting at a given location on the body's surface of **Figure 1**. Traction T is a *bound vector*, which means T cannot slide along its line of action or translate to another location and keep the same meaning. The stress field is the distribution of internal "tractions" that balance a given set of external tractions and body forces.

In other words, a traction vector cannot be fully described unless both the force and the surface where the force acts on have been specified. Given both *dF* and *ds,* the traction T can be defined as

$$\mathbf{T} = \lim_{\Delta s \to 0} \frac{\Delta \mathbf{F}}{\Delta s} = \frac{d\mathbf{F}}{ds}$$

The internal traction within a solid, or stress, can be defined in a similar manner. Suppose an arbitrary slice is made across the solid shown in the above figure, leading to the free body diagram shown at **Figure 2**. Surface tractions would appear on the exposed surface, similar in form to the external tractions applied to the body's exterior surface. The stress at point *P* can be defined using the same equation as was used for T.

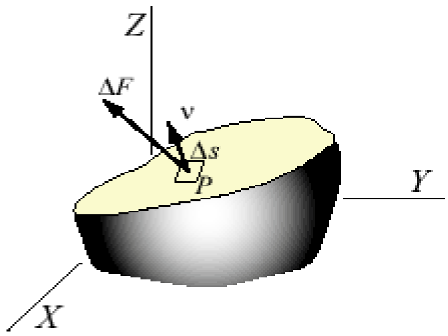


Figure 2 Free body

Surface tractions, or stresses acting on an internal datum plane, are typically decomposed into three mutually orthogonal components. One component is normal to the surface and represents *direct stress*. The other two components are tangential to the surface and represent *shear stresses*.

Direct stresses tend to change the volume of the material (e.g. hydrostatic pressure) and are resisted by the body's bulk modulus (which depends on the Young's modulus and Poisson ratio). *Shear stresses* tend to deform the material without changing its volume, and are resisted by the body's shear modulus.

Defining a set of internal datum planes aligned with a Cartesian coordinate system allows the stress state at an internal point *P* to be described relative to *x*, *y*, and *z* coordinate directions.

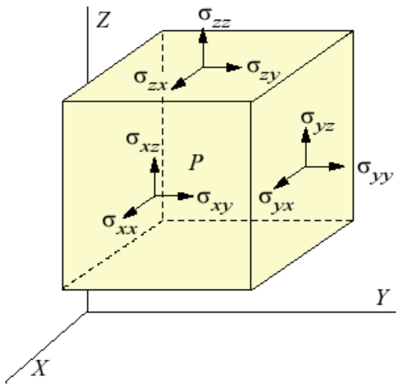


Figure 3 Stress components on a free body

For example, the stress state at point *P* can be represented by an *infinitesimal* cube with three stress components on each of its six sides (one direct and two shear components).

Since each point in the body is under static equilibrium (no net force in the absence of any body forces), only nine stress components from three planes are needed to describe the stress state at a point *P*.

Referring to **Figure 3**, these nine components can be organized into the matrix:

$$\begin{bmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{bmatrix}$$

here shear stresses across the diagonal are identical (i.e. $s_{xy} = s_{yx}$, $s_{yz} = s_{zy}$, and $s_{zx} = s_{xz}$) as a result of static equilibrium (no net moment). This grouping of the nine stress components is known as the *stress tensor* (or stress matrix).

2.2.4 Average Normal Stress

In order for the bar to undergo the uniform deformation, it is necessary that force P, is applied along the central axis of the cross section, and the material must be homogeneous and isotropic. Homogeneous material has the same physical and mechanical properties throughout its volume, and isotropic material has the same properties in all directions. Many engineering materials may be approximated as being both homogeneous and isotropic as assumed here. Steel, for example contains thousand of randomly oriented crystals in each cubic millimeter of its volume, and since most problems involving this material have a physical size that is much larger than a single crystals, the above assumption regarding its material composition is quite realistic.

2.2.5 Stress Strain Relationship

Stress always being discussed with the existing of strain. The stress-strain relationship is very important in structure design because the stress and strain can be known during the design. The ignorance of stress-strain relationship can lead to the failure of the design. The stress-strain relationship can be described from the equation below.

$$\varepsilon_x = +\frac{\sigma_x}{E} - \frac{\nu\sigma_y}{E} - \frac{\nu\sigma_z}{E} \tag{2.1}$$

$$\varepsilon_y = -\frac{\nu\sigma_x}{E} + \frac{\sigma_y}{E} - \frac{\nu\sigma_z}{E} \tag{2.2}$$

$$\varepsilon_z = +\frac{\nu\sigma_x}{E} - \frac{\nu\sigma_y}{E} - \frac{\sigma_z}{E} \tag{2.3}$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G} \qquad \gamma_{yz} = \frac{\tau_{yz}}{G} \qquad \gamma_{zx} = \frac{\tau_{zx}}{G} \tag{2.4}$$

2.2.6 Poisson's Ratio

When a deformable body is subjected to an axial tensile force, not only it does elongate but it also contracts laterally. Compressive force acting on a body causes it to contract in the direction of the force and yet its sides expend laterally. Within the elastic range the ratio of this strain is constant, since the deformations are proportional. This constant is referred to Poisson's ratio, υ, and it has a numerical value that is unique for particular material that is both homogeneous and isotropic as the equation below.

$$\upsilon = -\frac{\varepsilon_{lat}}{\varepsilon_{long}} \tag{2.5}$$

The negative sign is used here since longitudinal elongation causes lateral contraction, and vice versa. Notice that this lateral strain is caused only by the axial or longitudinal force. Poisson's ratio is seen to be dimensionless and for most nonporous solids it has a value that is generally between 1/4 and 1/3. In particular, an ideal material having no lateral movement when it is stretched or compressed and will have $\upsilon = 0$. The maximum possible value for Poisson's ratio is 0.5.

2.2.7 Tensile Strength

Once past the elastic limit, the material will not relax to its initial shape after the force is removed (Hooke's Law and Modulus of Elasticity). The tensile strength where the material becomes plastic is called yield tensile strength. This is the point where additional deformation (strain) of the material is unrecovered, and the work produced by external forces is not stored as elastic energy but will lead to contraction (Poisson Ratio), cracks and ultimately failure of the construction.

Clearly, this is an important point for the engineering properties of the material since here the construction may lose its loading capacity or undergoes large deformations. On the stress-strain curve below, this point is in between the elastic and the plastic region. The ultimate tensile strength of a material is the limit stress at which the material actually breaks, with sudden release of the stored elastic energy. This point is the fracture marked X on the curve shown in **Figure 4** below.

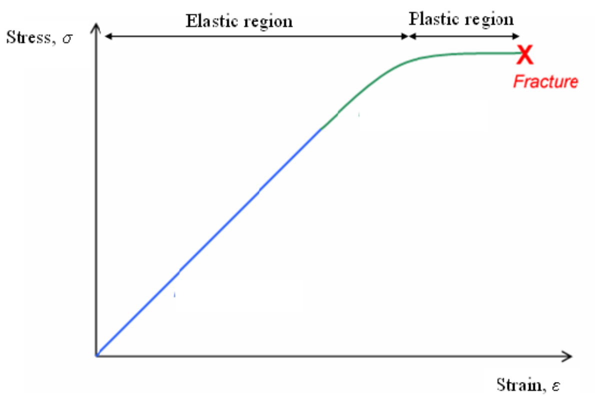


Figure 4 Stress-Strain curve diagram

For steel, the elastic limit is at about 0.2% and the breaking point is at 25% of the total (relative) extension. In steel constructions, the maximum allowable tensile stress at any point in the construction is 2/3 of the yield strength (or 0.2% deformation stress in metals or alloys without clearly defined yield stress). This comes down to a factor of safety of 1.5.

2.2.8 Piping

Piping includes pipe, flanges, fittings, bolting, gaskets, valves, and the pressure containing portions of other piping components. It also includes pipe hangers and supports and other items necessary to prevent over pressurization and overstressing of the pressure-containing components. It is evident that pipe is one element or a part of piping. Therefore, pipe sections when joined with fittings, valves, and other mechanical equipment and properly supported by hangers and supports, are called *piping*.

Nominal pipe size (NPS) is a dimensionless designator of pipe size. It indicates standard pipe size when followed by the specific size designation number without an inch symbol. For example, NPS 2 indicates a pipe whose outside diameter are 2.375 in. The NPS 12 and smaller pipe has outside diameter greater than the size designator (say, 2, 4, 6). However, the outside diameter of NPS 14 and larger pipe is the same as the size designator in inches. For example, NPS 14 pipe has an outside diameter equal to 14 in. the term *schedule* (SCH) was invented to specify the nominal wall thickness of pipe.

2.2.9 Pipe schedule and class

Two important specifications in piping are the material itself and also the pipe thickness as per standard and code. It was obvious that pipe is designated by nominal diameter, but the pipe thickness was designated by the schedule number. The larger the schedule number, the thicker the wall. The purpose of the various wall thicknesses is to meet up the pipe stress requirement. It means that the same pipe size can have different pressure rating according to the schedule of the pipe. Example;

12” Pipe (300mm) Sch 40 SML is translates to;

Nominal diameter:	12” (300mm)
Outside diameter:	12.75”(324mm)
Wall Thickness:	0.406”(10mm)
Inside diameter:	11.938”(303mm)
SML:	seamless pipe

The outer diameter of pipe is always constant for a particular size.

The inner diameter of pipe varies with the wall thickness (schedule no)

Pipe schedule always related to the pressure rating of the pipe. The thicker the pipe, the greater allowable pipe pressure can be applied. Logically, same material but different thickness would have different yield strength. In determining the pipe sizing, the pressure rating should be the most considerable factors. Then the types of fluids running through the pipe will determine the pipe’s material.

Pressure rating also important in determine the selection of flanges. Like pipe schedule, the higher the rating, the thicker the flange.

The wall thickness associated with a particular schedule depends on the pipe size as can be seen from the charts below for some of the more common sized carbon steel pipes encountered. Refer to **Table 1**

Abbreviations used: NB = nominal bore

STD = standard

EH = extra heavy

DBL EH = double extra heavy

Table 1 Different Size of Pipe Sizing

2"NB	OD = 2.375 inch (60.32 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	2.245	2.157	---	---	2.067	---	1.939	---	---	---	1.689	1.503
ID (mm)	57.02	54.79	---	---	52.5	---	49.25	---	---	---	42.9	38.18

3" NB	OD = 3.5 inch (88.9 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	3.334	3.260	---	---	3.068	---	2.900	---	---	---	2.624	2.300
ID (mm)	84.68	82.8	---	---	77.93	---	73.66	---	---	---	66.65	58.42

4" NB	OD = 4.5 inch (114.3 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	4.334	4.260	---	---	4.026	---	3.826	---	3.624	---	3.438	3.152
ID (mm)	110.08	108.2	---	---	102.26	---	97.18	---	92.05	---	87.33	80.06

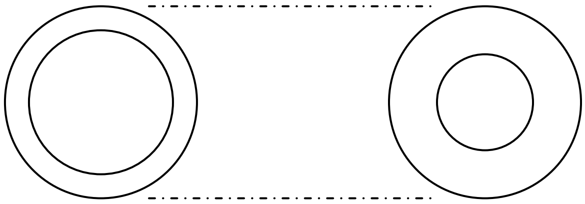
6" NB	OD = 6.625 inch (168.275 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	6.407	6.357	---	---	6.065	---	5.761	---	5.501	---	5.189	4.897
ID (mm)	162.74	161.47	---	---	154.05	---	146.33	---	139.73	---	131.8	124.38

8" NB	OD = 8.625 inch (219.1 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	8.407	8.329	8.125	8.071	7.981	7.813	7.625	7.439	7.189	7.001	6.813	6.375
ID (mm)	213.54	211.56	206.38	205	202.72	198.45	193.67	188.95	182.6	177.83	173.05	161.93

10" NB	OD = 10.750 inch (273 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	10.482	10.42	10.25	10.136	10.02	9.750	9.564	9.314	9.064	8.750	8.500	---
ID (mm)	266.24	264.67	260.35	257.45	254.5	247.65	242.93	236.58	230.23	222.25	215.9	---

12" NB	OD = 12.750 inch (323.85 mm)											
Schedule	5	10	20	30	40 STD	60	80 EH	100	120	140	160	DBL EH
ID (ins)	12.42	12.39	12.25	12.09	11.938 12.000	11.626	11.376 11.750	11.064	10.75	10.50	10.126	---
ID (mm)	315.47	314.7	311.15	307.1	303.22 304.8	295.3	288.95 298.45	281.03	273.05	266.7	257.2	---

From the table, we can conclude that the pipe inner diameter decrease when the pipe schedule increased. For example 4 inch pipe diameter, refer to **Figure 5**



OD = 4.5 inch OD = 4.5 inch

ID = 4.26 inch ID = 3.438 inch

Schedule 10 ***Schedule 160***

Figure 5 Different Pipe Size

For piping class, there are 6 classes that often used;

ANSI 2500 #, ANSI 1500 #, ANSI 900 #, ANSI 600 #, ANSI 300 # and ANSI 150 #

2.2.10 Von Mises Equivalent Stress

Von Mises stress, is a scalar function of the components of the stress tensor that gives an appreciation of the overall 'magnitude' of the tensor. Plastic yield initiates when the Mises stress reaches the initial yield stress in uniaxial tension and, for hardening materials, will continue provided the Mises stress is equal to the current yield stress and tending to increase. Mises stress can then be used to predict failure by ductile tearing. It is not appropriate for failure by crack propagation or fatigue, which depend on the maximum principal stress.

The principal stresses (σ_1, σ_2, σ_3) are calculated from the stress components by the cubic equation:

$$\begin{bmatrix} \sigma_x - \sigma_0 & \sigma_{xy} & \sigma_{xz} \\ \sigma_{xy} & \sigma y - \sigma_0 & \sigma_{yz} \\ \sigma_{xz} & \sigma_{yz} & \sigma_z - \sigma_0 \end{bmatrix} = 0 \qquad (2.6)$$

σ_0 = principal stress (3 values)

The three principal stresses are labeled σ_1, σ_2, and σ_3. The principal stresses are ordered so that σ_1 is the most positive (tensile) and σ_3 is the most negative (compressive). The Von Mises or equivalent stress σ_e is computed as:

$$\sigma_e = \left(\frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right] \right)^{\frac{1}{2}} \tag{2.7}$$

2.2.11 Stress Concentration Factor, **K_t**

Normally failure in machines and structures virtually always initiate at sites of local stress concentration caused by the geometrical or micro structural discontinuities. In reality, it is difficult to design machine without permitting some changes in the cross sections of the members. As an example, rotating shafts must have shoulders designed on them so that the bearing can be properly seated and so that they will take thrust load, and the shafts must have key slots machined into them for securing pulleys and gears. Some other parts require holes, oil grooves, and notches of various kinds. Any discontinuity in a machine part alters the stress distribution on the neighborhood of the discontinuity sp that the elementary stress equations no longer describe the state of stress in the part. These stress concentrations (or can be called by stress raisers) often lead to local stresses many time higher than the nominal net section that would be calculated without considering the stress concentration effects. The theory of stress concentration factor states that K_t is the ratio of the maximum local stress in the region of discontinuities to the nominal net section stress. It simply can be calculated by:

$$K_t = \frac{actual \max stress}{equivalentstress}$$

$$K_t = \frac{\sigma_{\max}}{\sigma_{eq}} \tag{2.8}$$

It should be noted that the value of K_t is valid only for stress levels within the elastic range. The seriousness of the stress concentration depends on the type of loading, type of materials, the size and shape of the discontinuities.

For the investigation of K_t value for welded pipe fitting, we are going to K_t define as below,

$$K_t = \frac{\sigma_{\max}}{\sigma_{equivalent}} \tag{2.9}$$

2.3 Assumption Made Base on the Theory

In order to investigate the value of stress concentration factor, K_t, set assumptions need to be done in order to simplify the problem solution process. These assumptions would help to build a narrow area and manageable study that give a valid value of stress concentration factor. Next, the assumptions made will be represented below.

2.3.1 Three Dimension Stress Study (3-D analysis)

In this study, the semi axial symmetry model for the welded pipe fitting is created by tetrahedral structural solid element (SOLID 92 of ANSYS software). The pipe branch fitting was subject to internal pressure. Owing to the absence of axial symmetry, it is necessary to adopt a large number of node points and generate sufficient meshes to guarantee the accuracy of the result.

The typical configuration and basic nomenclature of the pipe-nozzle connection model is defined as shown in **Figure 8**. The following assumptions were employed.

1) A homogeneous and isotropic material is assumed.
2) The influences of weight and temperature effects are neglected.
3) In the pipe fitting connection model, all the ends of the header and branch pipe are assumed to be fixed. The length of the header and branch are sufficiently long so that the boundary conditions at the ends of the header, as well as the branch, will not affect the stress results.

2.3.2 Material Characteristic

The material of the pipe fitting is assumed to be in homogenous, isotropic and continuous. It is made so that the analysis of the stress on the pipe fitting is simplified without any varieties.

2.3.3 Assumption for Structure of Piping Branch

1) Taking the branch piping junction as two intersecting cylindrical shells and neglecting the elliptic of run and branch pipe and machining error.
2) Ignoring the strengthening effectiveness due to connecting materials at intersections and other components.

CHAPTER 3

METHODOLOGY/ PROJECT WORK

3.1 METHODOLOGY

A 3-dimensional piping branch model as well as a simulation analysis of the entire model is required for this particular project. The 3 dimensional models would serve as an effective tool for presentations whereas the simulation analysis model would serve as an effective tool for verifications of strengths.

The methodology use in modeling involves defining element type and material properties. Then this model is built using ANSYS software. Load and meshing control are then defined. Finally, the stress concentration factor is compute and results are plotted to see the distribution of certain variables, such as component of stress over the entire model. Refer to **Figure 6.**

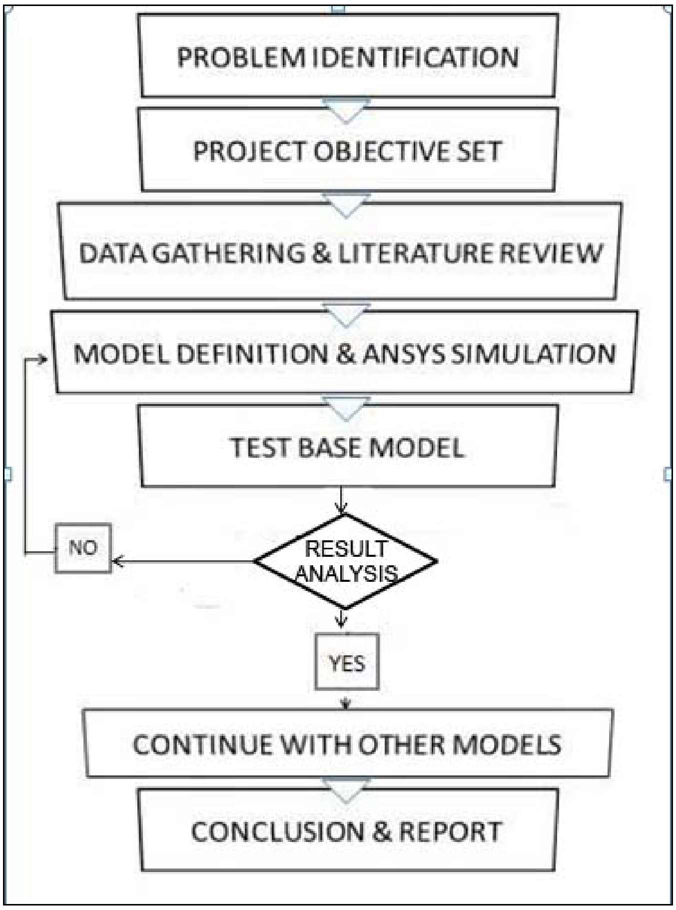


Figure 6 Flow process of the entire project

3.1.1 *Define and Analyze Case*

In the first phase, the cases are defined before start to be analyzed. The project objectives will be the goal for this project. The success of this project is measured from the achievement of the objectives.

3.1.2 *Validation*

The next phase would be the validation of the output result obtained from the analysis through the theoretical method. It was done through research from the reference books to have a better understanding on the expected result from the project. The output was then was verified by comparing the result with the data from the reference books. This stage is important to ensure the input parameters such as geometrical shape, applied load, material properties and elements are correct.

3.1.3 *ANSYS Analysis Flow*

Below are the steps to analyze the t-pipe model. The analysis is divided into separate preprocessing, solution and post processing steps. Please refer to **Figure 7**.

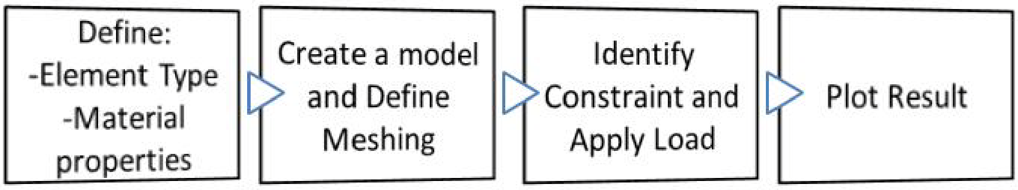


Figure 7 Steps involved during the analysis using ANSYS V9.0 software.

3.1.4 *Preprocessing Phase*

a) Element Type

In this program, each element is identified by a category name followed by a number. For example, three-dimensional solid elements have the category SOLID. SOLID 92 is an element used to model three-dimensional structure solid problems.

b) Material Properties

Most element types require material properties. Depending on the application, material properties can be linear or nonlinear. Linear material properties can be constant or temperature dependent, and isotropic or orthotropic. As with element types and real constants, each set of material properties has a material reference number.

c) Modeling

There are two approaches to construct a finite element model's geometry that are direct generation and solid-modeling approach.

d) Meshing

Meshing is a process to create a finite element model by dividing the geometry into nodes and elements. It can automatically generate the nodes and elements by specifying the element attributes and element size. The element attributes include element types, and material properties. The element sizes control he fines of the mesh. The smaller the element size, the finer the mesh.

3.1.5 Solution Phase

a) Define load

Apply boundary condition, initial conditions and loading such as pressure for this case.

b) Solution

Solve a set of linear and nonlinear algebraic equations simultaneously to obtain nodal results.

3.1.6 Post Processing Phase

a) Read Result

List the result in a tabular form.

b) Plot Result

Contour display is used to see the distribution of certain variables, such as component of stress over the entire model. Besides, it can obtain other important information such as values of von Mises stress.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Analysis Case

In analysis, uncertainties in numerical values are modeled as random variables. Quantities normally modeled as random include loads, material properties, element properties, boundary conditions and dimensions.

Types of variables used in this analysis are:

a) Material properties variability (modulus of elasticity, Poisson's ratio)
b) Element properties (the cross-sectional properties of pipe fitting)
c) Structure dimensions (reinforcement, angle of branch pipe, diameter of branch pipe)
d) Loading (internal pressure applied to the pipe fitting)

4.2 Input Data

Based on the theory and assumptions made, the model that would be investigated using ANSYS is categorized by:

1. different angle of pipe branch
2. different diameter ratios

There are also some information needs to be identify to start with the analysis. It can be summarized by **Table 2** and **Figure 8**

Table 2 Material properties

Data	Description
Material	Carbon Steel
Poisson's Ratio, υ	0.28
Modulus of Elasticity	2.1×10^{11} kpsi

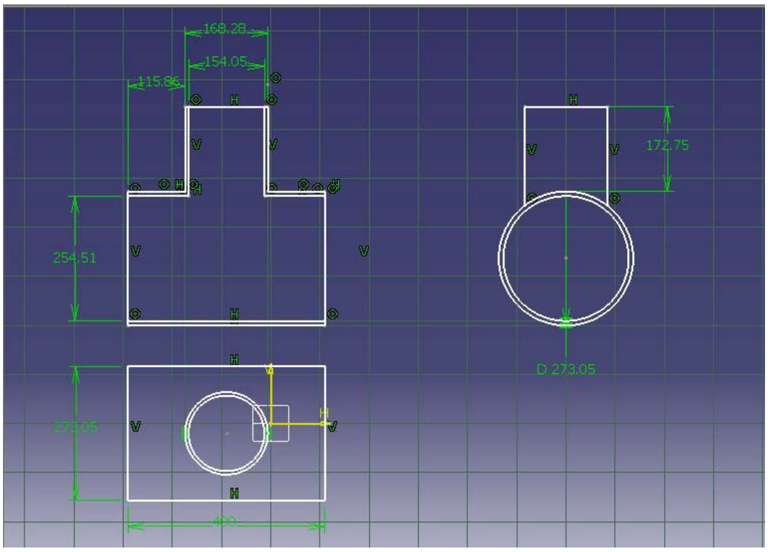


Figure 8 Model input

This project is done by varies the ratio of branch pipe diameter (*d)* to header pipe diameter (*D*). Seven simulations created by varying the *d/D* ratio. The seven *d/D* values are 0.206, 0.246, 0.306, 0.354, 0.402, 0.504 and 0.605 as in **Figure 9.**

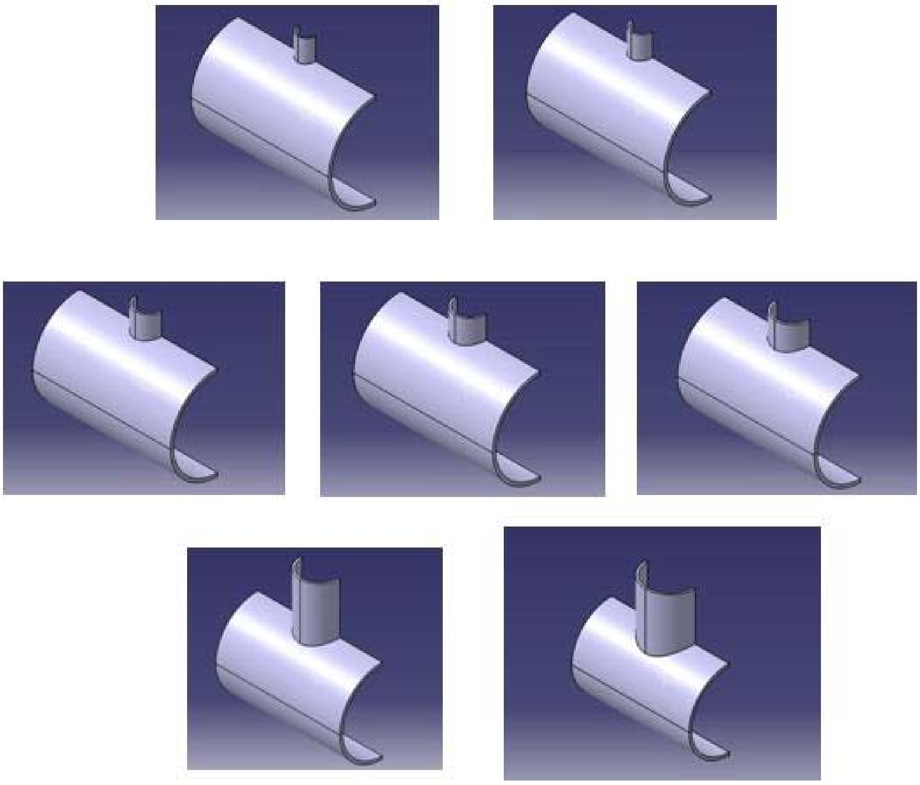


Figure 9 90 degrees pipe branch with seven different diameter ratios

In order to get this ratio, the real size of pipe is obtained by referring ASME 36.10 (Welded and Seamless Wrought Steel Pipe). The dimension of model can be summarized in **Table 3.** For all dimensions, the identification is Standard (STD) and Schedule number is 40.

Table 3 Model parameter

No	**d/D**	**Header**			**Branch**			**t/T**
		NPS	**D**	**T**	**NPS**	**D**	**t**	
1	0.605	**10**	10.02	0.365	**6**	6.065	0.28	0.767
2	0.504	**10**	10.02	0.365	**5**	5.047	0.258	0.707
3	0.402	**10**	10.02	0.365	**4**	4.026	0.237	0.649
4	0.354	**10**	10.02	0.365	**3.5**	3.548	0.226	0.619
5	0.306	**10**	10.02	0.365	**3**	3.068	0.216	0.592
6	0.246	**10**	10.02	0.365	**2.5**	2.469	0.203	0.556
7	0.206	**10**	10.02	0.365	**2**	2.067	0.154	0.422

For the purpose of this project, student has managed to simulate and test all 3 different angles for the pipe branch. The data obtained from all the simulation are used to analyze the effect of internal pressure due to different diameter ratios and angle of branch.

4.2.1 Sample Calculation

Stress Reaction from Pressure Exerted in the Pipe

Using the detail defined form **Table 2** and **Table 3;** we need to calculate the value of *uniform stress reacted* from the pressure exerted inside the header pipe. Due to internal pressure, the element of pressure vessel will experience circumferential stress

$$\sigma_c = \left(\frac{pr}{t}\right) = 4118 \text{ psi}$$

p = applied load = 300 psi

r = internal radius of main pipe = 5.01 inch

t = thickness of main pipe = 0.365 inch

From equation 2.7, von Mises equivalent stress is calculated as:

$$\sigma_e = \left(\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]\right)^{\frac{1}{2}}$$

Applying:

Principal Stress 1 (σ_1) = $\left(\frac{pr}{t}\right)$ = *4118 psi*

Principal Stress 2 (σ_2) = $\left(\frac{pr}{2t}\right)$ =*2059 psi*

Principal Stress 3 (σ_3) = *0 psi*

So, we get $\sigma_e =$ 3566.13 psi

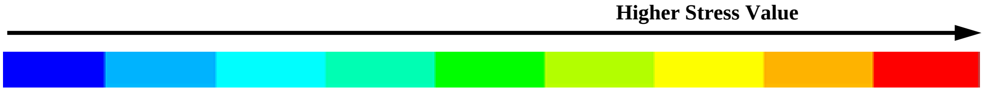
The value is applied to Equation 2.8 to get the stress concentration factor. The maximum stress is obtained by ANSYS simulation for every dimension

$$K_t = \frac{\sigma_{\max}}{\sigma_{equivalent}} \quad \text{,if } \sigma_{\max} = 27712 \text{ psi}$$

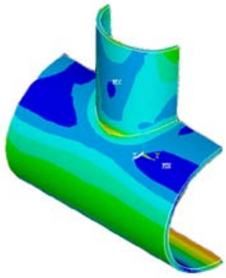
$$K_t = \frac{27712}{3566} = 7.8$$

4.3 Result for 90°Tee Fitting With Internal Pressure 300 psi

Higher Stress Value



Minimum Stress **Maximum Stress**



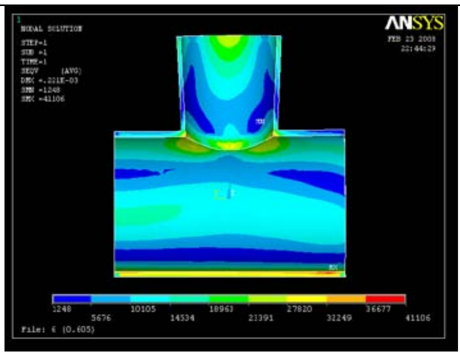
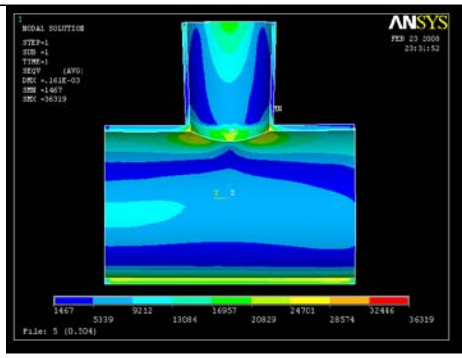
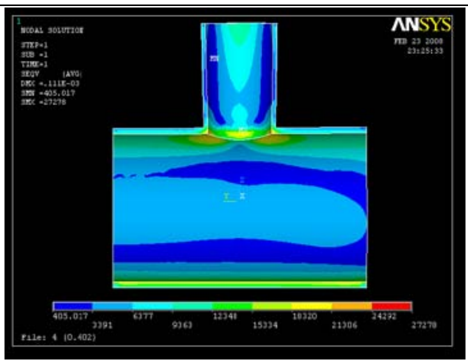
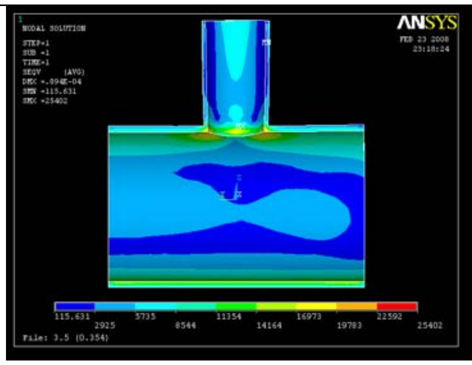
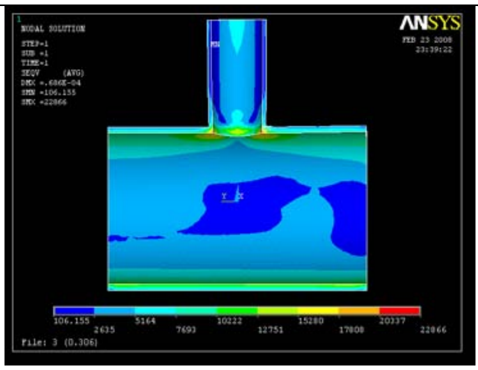
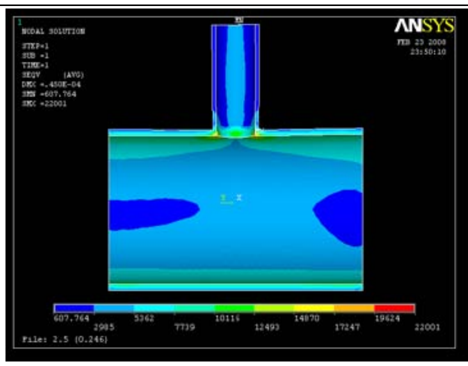
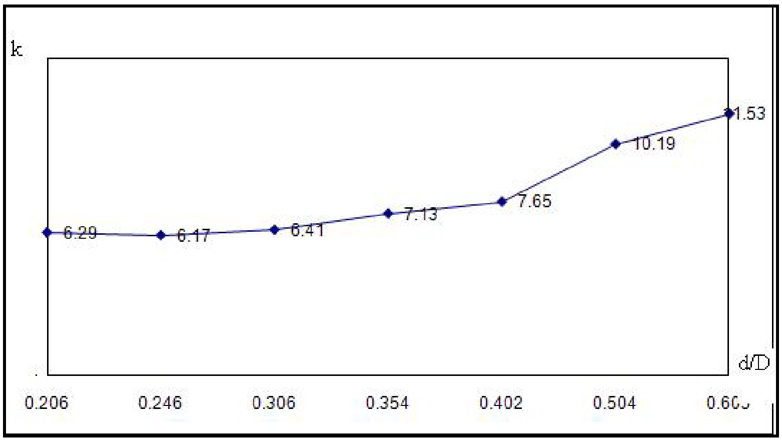
D/d	Inside Wall	D/d	Inside Wall
0.605		0.504	
0.402		0.354	
0.306		0.246	

Figure 10 Von Mises Stress Simulated by ANSYS for 90° tees fitting. The red color in the figure and indicators shows the highest area experience stress

Table 4 Stress concentration factor for 90^o tees fitting with internal pressure

No	d/D	Equivalent stress (psi)	Maximum stress (psi)	Stress concentration factor (Kt)
1	0.605	3566	41106	11.5
2	0.504	3566	36319	10.2
3	0.402	3566	27278	7.6
4	0.354	3566	25420	7.1
5	0.306	3566	22866	6.4
6	0.246	3566	22001	6.2
7	0.206	3566	22433	6.3

Referring to the **Figure 9** simulated tees branch pipe with internal pressure is showed. The simulation started with d/D equal to 0.206 and finished d/D equal to 0.605. From the **Table 4**, the highest maximum value of stress concentration factor is 11.5 at d/D equal to 0.605.



Graph 1 Stress concentration factor versus d/D for welded 90^o tees fitting

Overall, the stress concentration is increasing linearly. This can be observed from **Graph 1**. From **Figure 11**, the simulated ANSYS show that all the maximum stress is concentrated at the most 90^o between the intersection of the header pipe and branch pipe. As the bent sharp is reduced the stress is less concentrated.

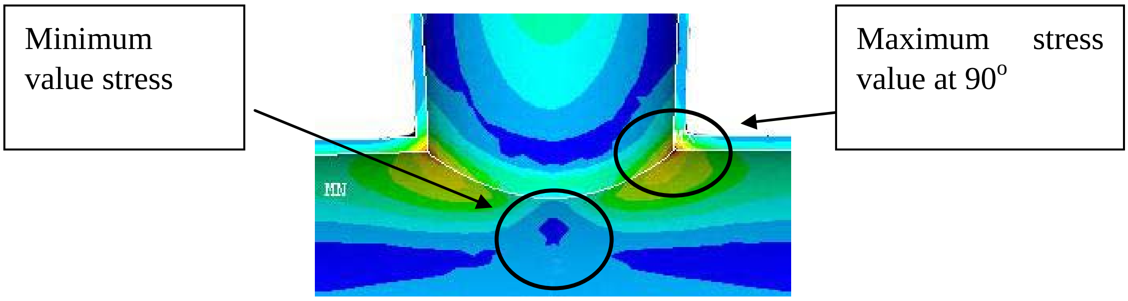


Figure 11 Maximum and minimum stress for welded 90° tees fitting

The location of stress maximum area is same for all the ratio d/D. However the distribution of stress at the welding area is higher for higher ratio of d/D. It can be observe from **Figure 12**, the figure show that ratio d/D = 0.605, has a higher stress concentration and experienced higher stress distribution along the joint area due to internal pressure compare to d/D = 0.206, which experience lower stress concentration. So, it is concluded that due to internal pressure, the joint area for higher ratio d/D effected more that smaller ratio d/D.

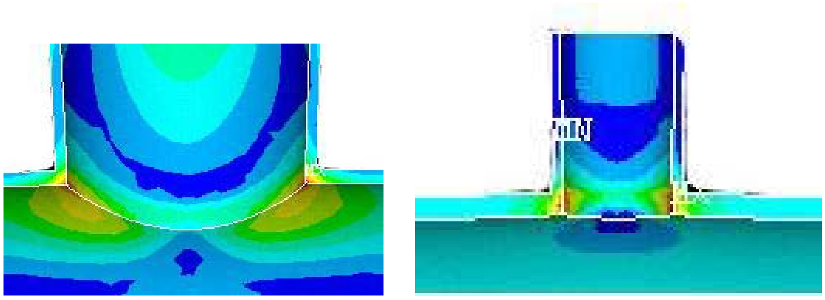


Figure 12 Stress distributions along joint

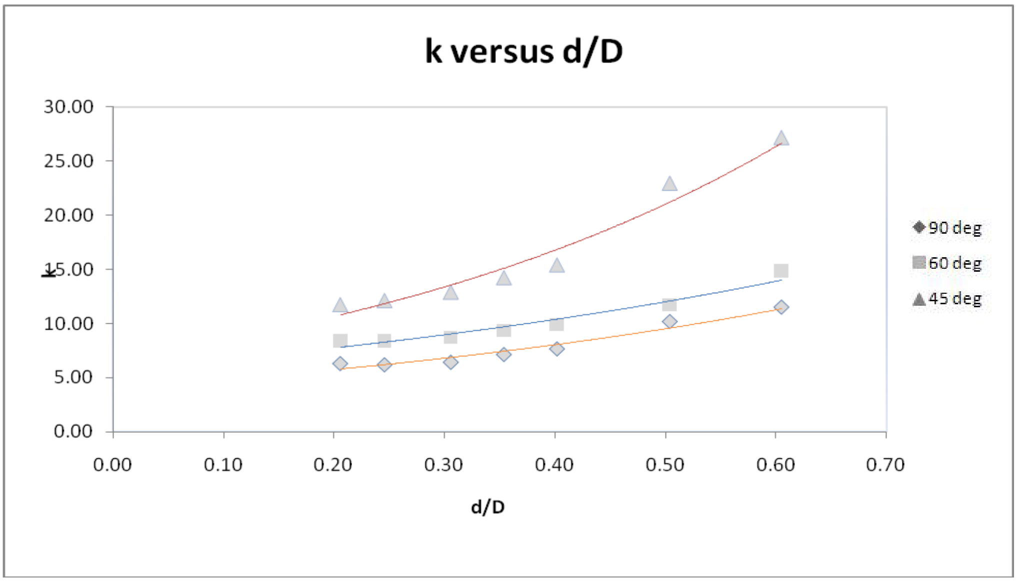
Another thing that can be observed from the simulation the area covered different colours on the pipe. If we take the lowest stress value (light blue area) as our reference in this observation, we can see the difference of it area distribution for different diameter ratios. Branch that have a bigger diameter ratio ratios experience a significantly greater effect on the main pipe compared to branch with smaller diameter ratios. The stress in the smaller diameter ratio branch is more concentrated on the joint without much influencing the main pipe.

4.4 Result for 45° and 60° Tee Fitting With Internal Pressure 300 psi

Table 5 Stress concentration factor for 45° and 60° tees fitting with internal pressure

No	d/D	Equivalent stress (psi)	Stress concentration factor (Kt) for 45°	Stress concentration factor (Kt) for 60°
1	0.605	3566	27.0	14.9
2	0.504	3566	23.0	11.7
3	0.402	3566	15.4	9.9
4	0.354	3566	14.1	9.3
5	0.306	3566	12.8	8.7
6	0.246	3566	12.0	8.4
7	0.206	3566	11.7	8.4

From the **Table 5**, the highest maximum value of stress concentration factor is 27.0 at d/D equal to 0.605 for 45° branch while the highest maximum value of stress concentration factor is 14.9 at d/D equal to 0.605 for 60° branch connection. Based on **Graph 2,** we can see that the pattern is almost the same for the three different branch angles that is the stress concentration factor increased linearly as the diameter ratio increases.



Graph 2 Comparison of Stress concentration factor versus d/D for 45°, 60° and 90°

Overall, the stress concentration is increasing linearly. This can be observed from **Graph 1**. The simulated ANSYS show that all the maximum stress for all three different branch angles have the same pattern. We can observe that stress is concentrated the most between the intersection of the header pipe and branch pipe. As the bent sharp is reduced the stress is less concentrated.

4.5 Stress distribution at Critical Area

Based on the result being obtained, the area where maximum stress are concentrated are being focused in order to analyze the trend of the stress distribution. The area is as shown in **Figure 13.**

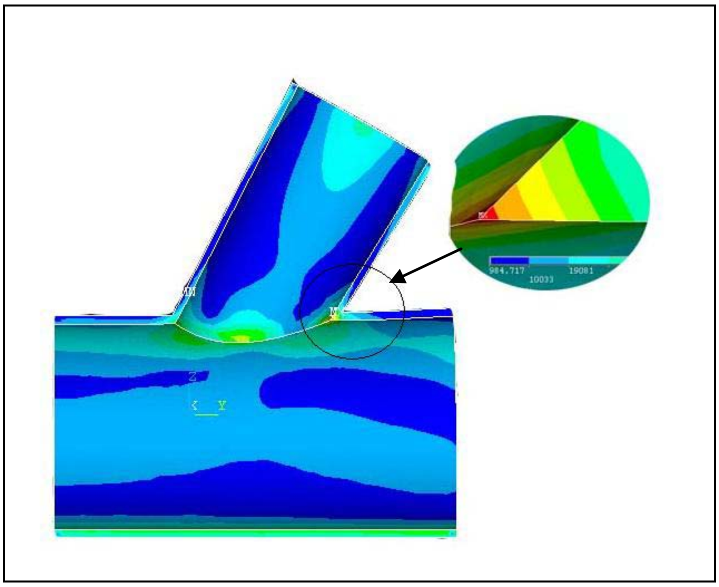
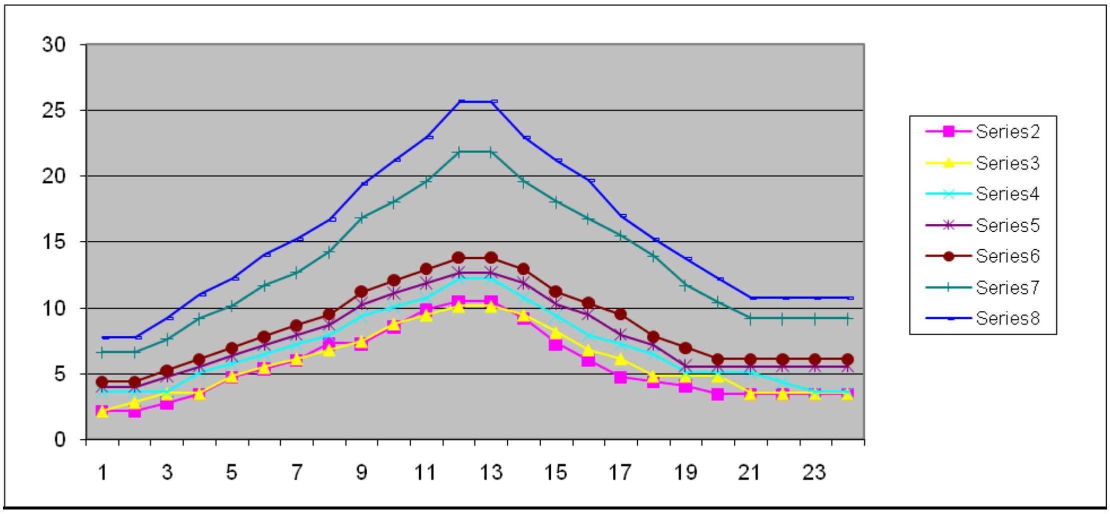
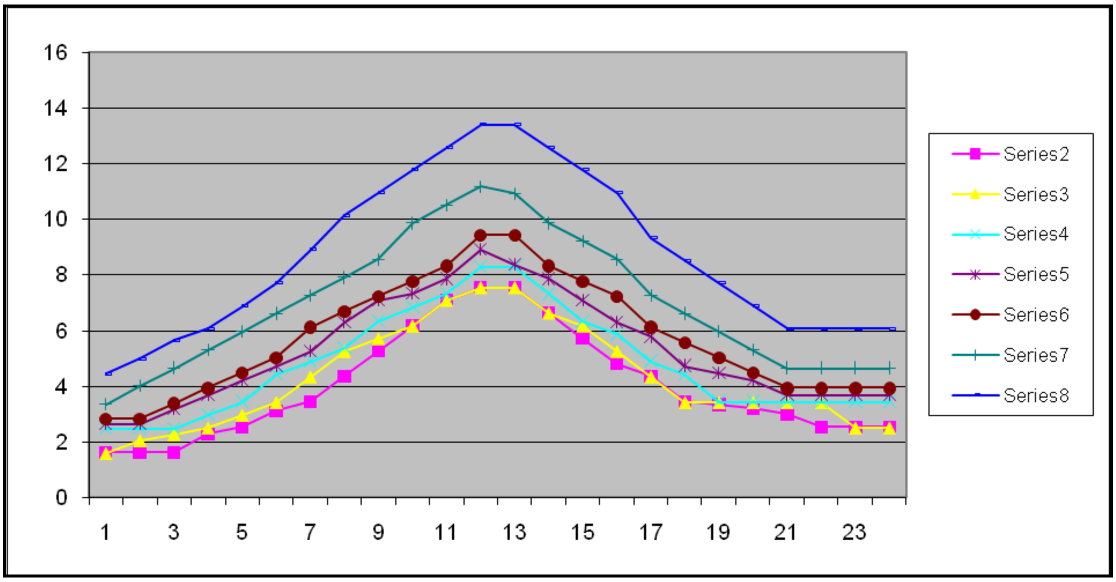


Figure 13 Stress Concentrated Area for 45^{o} Angle

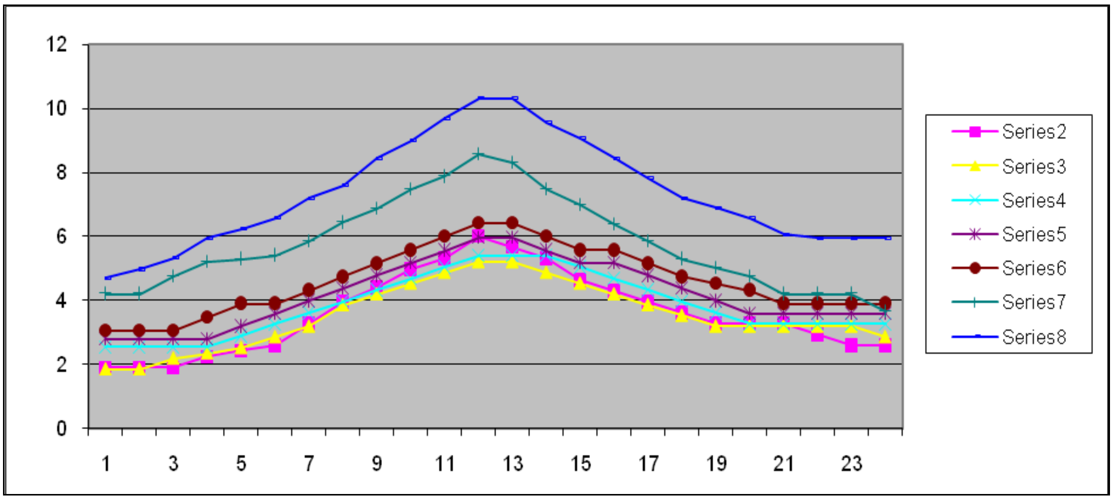
By taking several value of parameter near the edge of the pipe joint, graph of the stress distribution can be plotted and analysis can be done for the stress distribution for 45^{o}, 60^{o} and 90^{o} pipe angle. This can be referred to **Graph 3, Graph 4** and **Graph 5** where the peak of the graph is at the edge between the main pipe and branch pipe that is where the maximum stresses are concentrated.



Graph 3 Stress Distributions at Critical Area for 45^o



Graph 4 Stress Distributions at Critical Area for 60^o



Graph 5 Stress Distributions at Critical Area for 90^o

CHAPTER 5

CONCLUSION

On this report, student has defined the model of investigation and discussed the result obtained from ANSYS simulation. The analysis results of the various angle branch pipes with different angle demonstrate a good relationship between each of them. The general concept of stress state that the stress concentration factors increase as the angle of pipe branch is reduces. Here we can conclude that branch with higher dimension ratio obtained higher stress concentration factor. Branch with bigger diameter ratio also experience more stress on the main pipe compared to branches with smaller diameter ratios where it can sometimes be neglected. The graph is important medium in showing the pattern of the stress for all type of design. The results obtained from the graph show the correct shape but it concentration factor value is still quite high. Further investigation and analysis need to be done in order to determine the factors that lead to this result.

RECOMMENDATION

Recommendations have been made that is;

Cut off the inside corner of intersection to reduce the effect of internal pressure

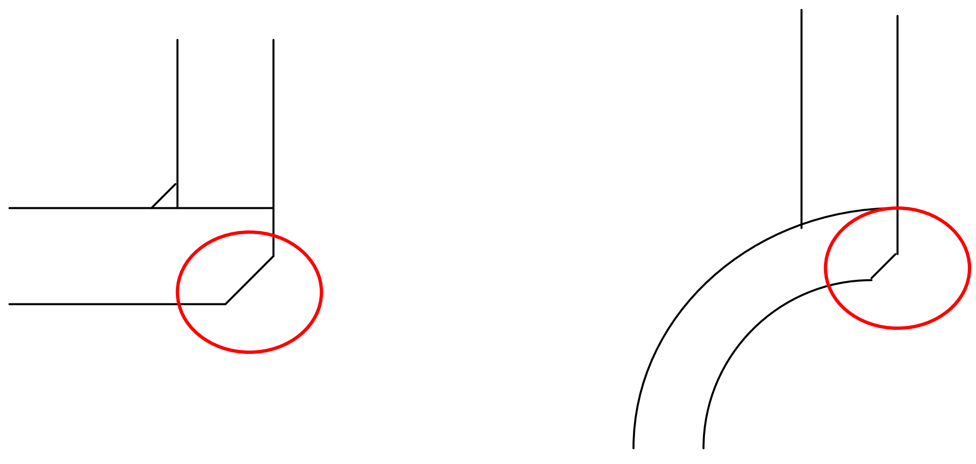


Figure 19: Inside corner of intersection grounded off.

By cutting of the edge where the stress are concentrated, the stress distribution can become much better where there is not much differences between maximum and minimum stress value.

REFERENCES

1. ASME Standard B31.3; *Process Piping*

2. ASME B 36.10 : *Welded and Seamless Wrought Steel Pipe*

3. ASME B31. 1; *Power piping*

4. Beer, Johnston, DeWolf, Mechanics of Materials 2004, Third edition, Mc Graw Hill

5. Andrew Pytel and Jaan Kiusalass, Mechanics of Materials, 2003, Thomson Learning

6. Irving H. Shames and James M. Pitarresi, Introduction to Solid Mechanics, 2000 ,Third Edition, Prentice Hall

7. Walter D.Pilkey, Peterson's Stress Concentration Factor, 2nd Edition, A. Wiley Production

8. Saeed Moaveni, Finite Element Analysis Theory and Application with ANSYS, 2nd Edition, 2003, Pearson Education Inc, New York.

9. http://www.mech.uwa.edu.au/DANotes/SSS/failure/theories.html

10. http://www.ndted.org/EducationResources/CommunityCollege/Materials/Mechanical /Tensile.htm

11. http://www.sciencedirect.com

12. http://www.engin.brown.edu/courses/en175/Notes/Elastic_material/Elastic_material.htm

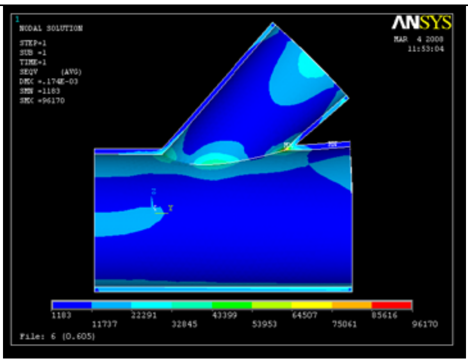
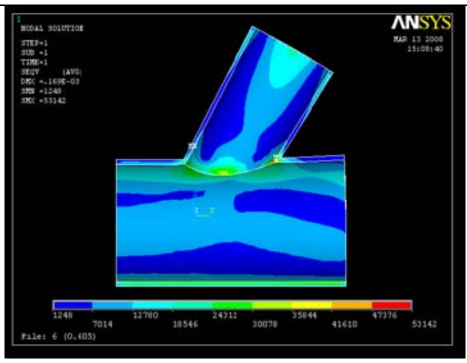
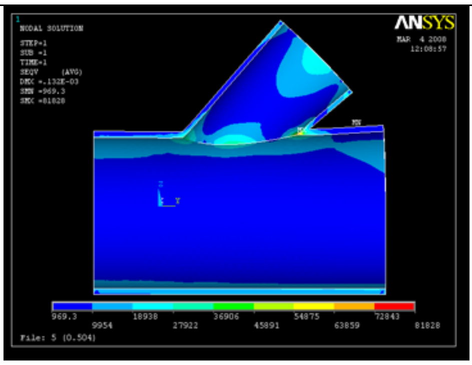
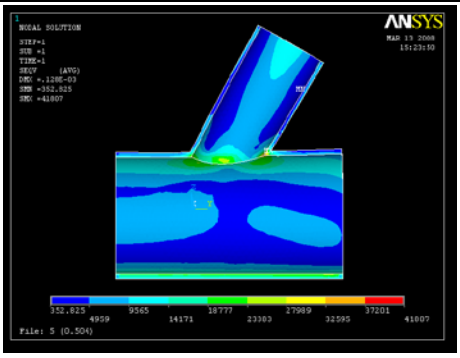
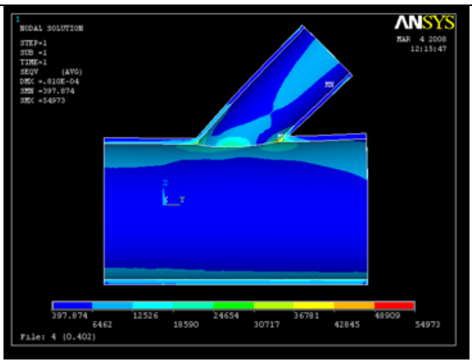
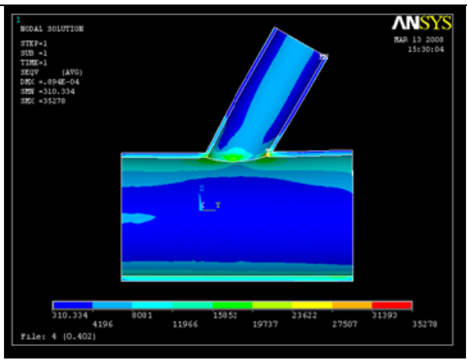
APPENDICES

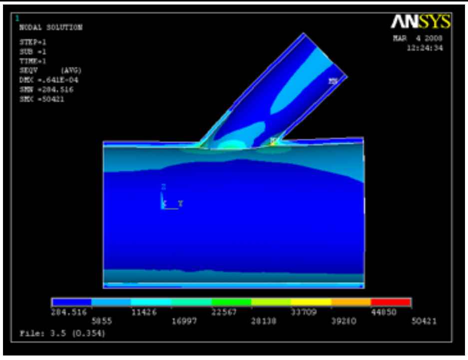
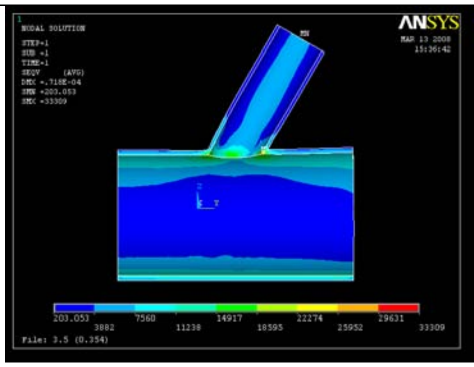
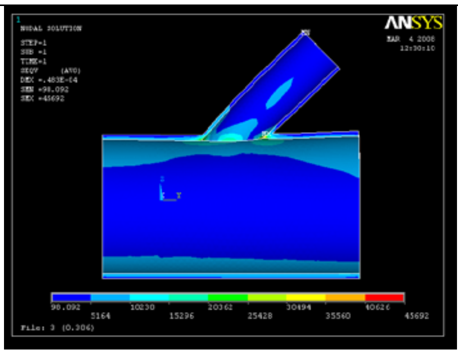
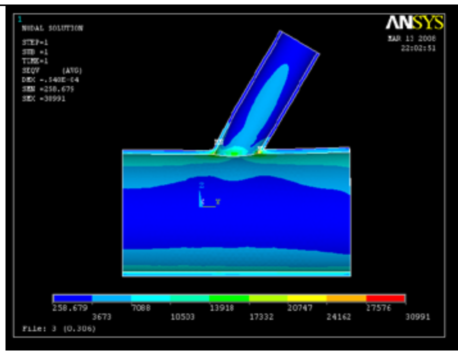
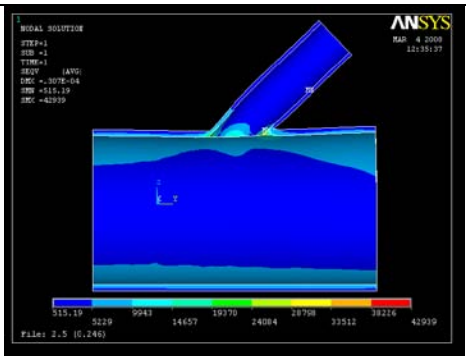
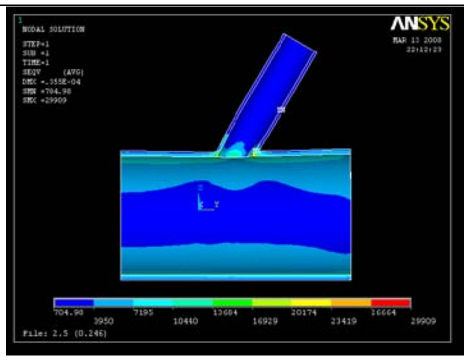
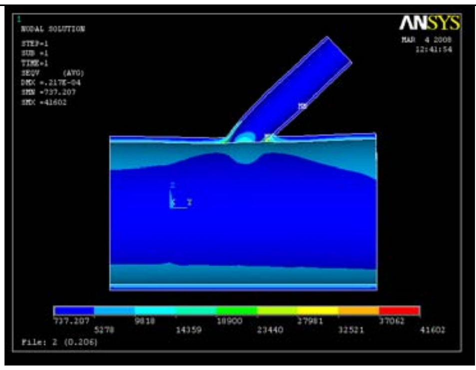
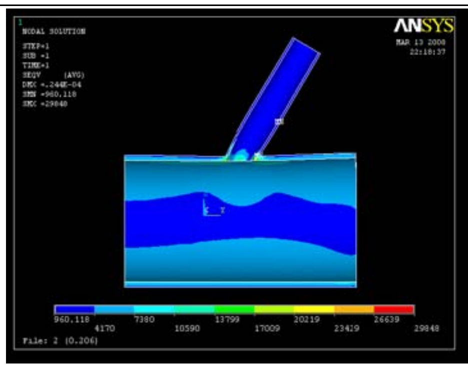
APPENDIX A

RESULT FOR 45^{O} AND 60^{O} TEE FITTING WITH INTERNAL PRESSURE 300 PSI

Higher Stress Value

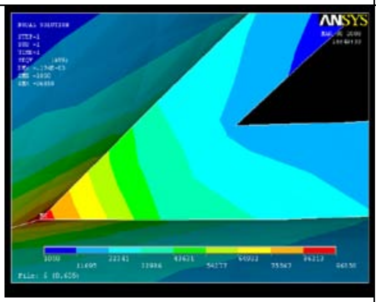
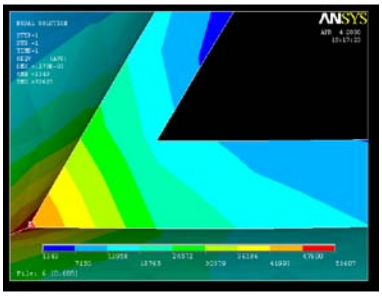
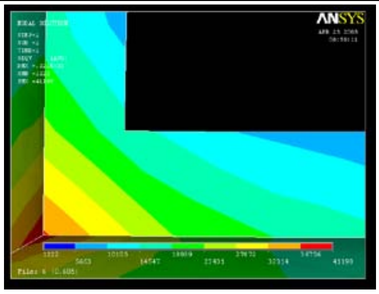
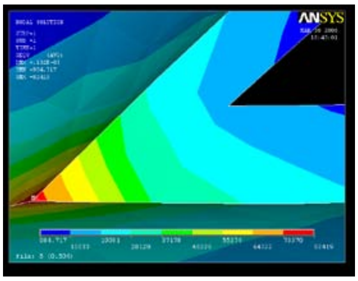
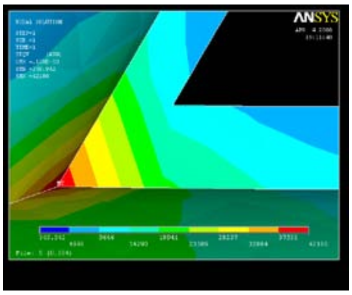
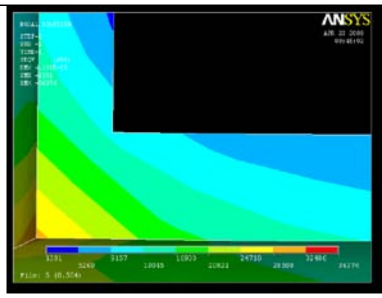
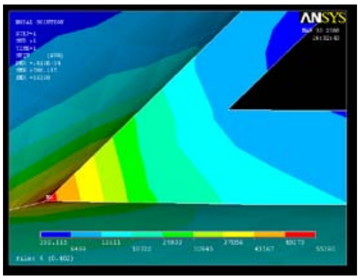
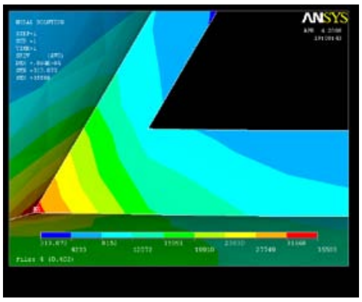
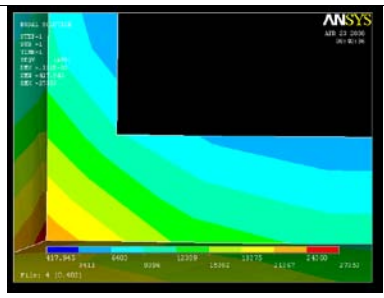
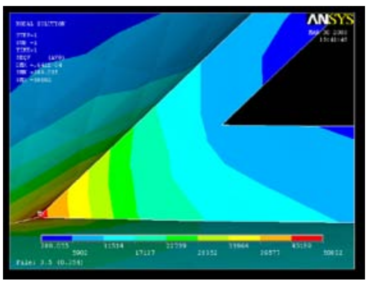
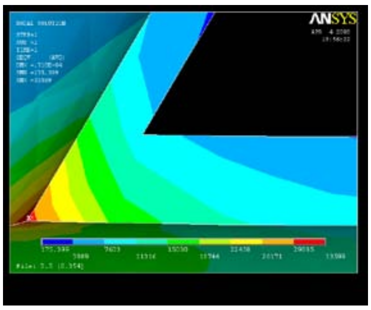
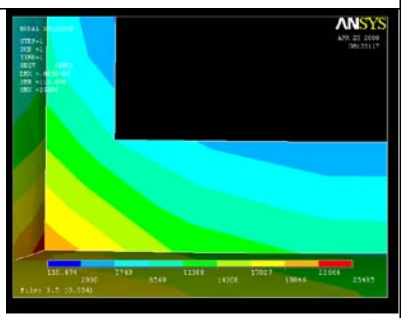
Minimum Stress **Maximum Stress**

D/d	45^{o}	60^{o}
0.605		
0.504		
0.402		

0.354		
0.306		
0.246		
0.206		

APPENDIX B

STRESS DISTRIBUTION AT CONCENTRATED AREA FOR 45^{O} 60^{O} AND 90^{O}

D/d	45^{o}	60^{o}	90^{o}
0.605			
0.504			
0.402			
0.354			

0.306