

CHAPTER 3
METHODOLOGY

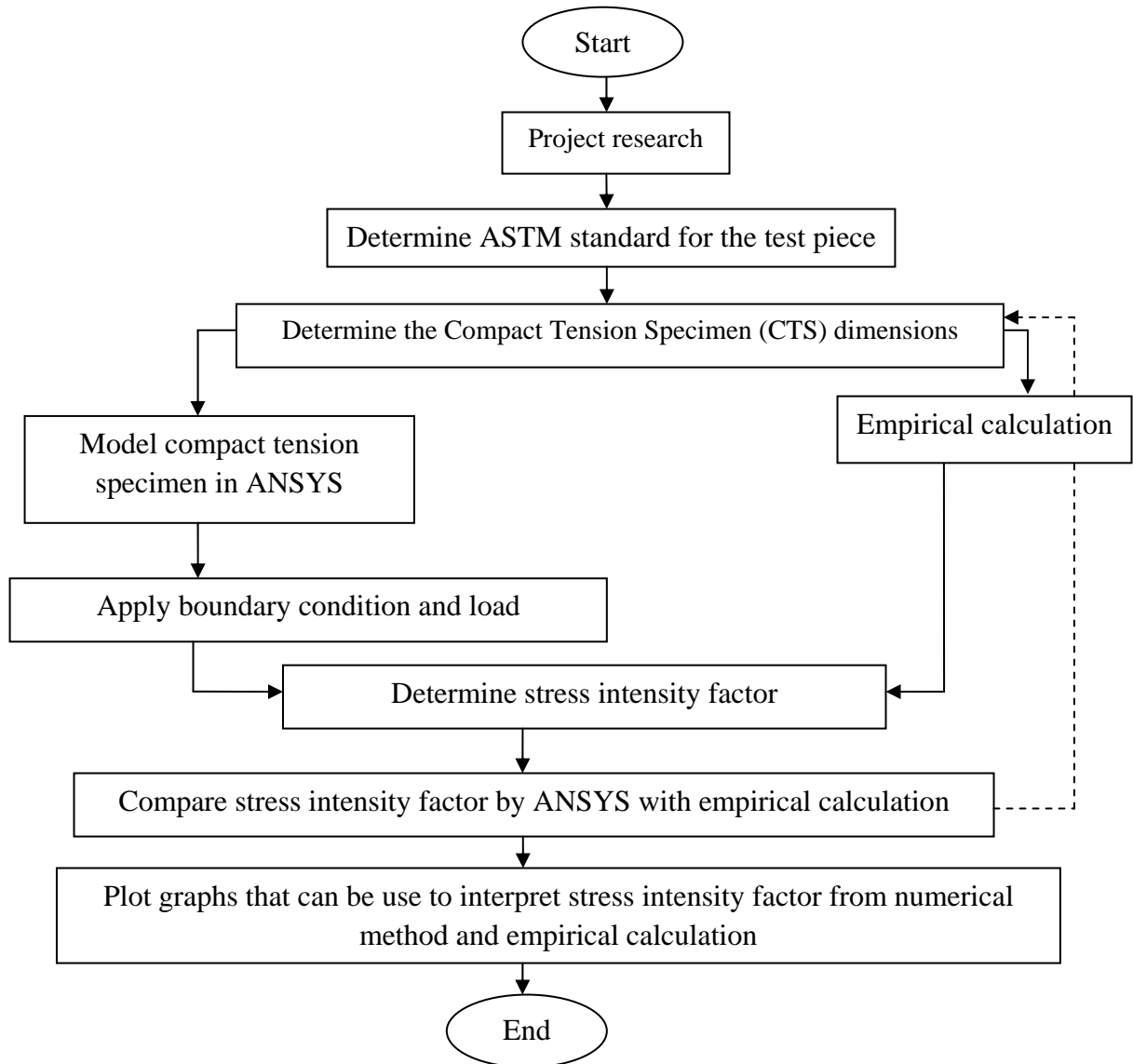


Figure 3.1: Methodology of Project

After the author had understands the concept of this project, the author had develop methodology of the project as in Figure 3.1.

3.1 Project research

Project research had been done to get the rough idea and to get basic understanding about the theoretical part of this project. Project start based on reading from textbook, discussion with lecturer and assist by PhD student. Besides that, internet source also useful to get the relevant source. The American Society of Testing and Material (ASTM) specification and configuration for compact tension specimen used in this project.

3.2 Determination of stress intensity factor by using empirical calculation

Referring to ASTM E399-90, for compact tension specimen there are specific requirement to calculate stress intensity factor. This requirement had been stated in section 2.7 and 2.8 by using equations 2.8 to 2.12.

3.3 Determination of stress intensity factor by using numerical method (ANSYS)

ANSYS software is a commercial finite element code used in industry to solve large scale problems. It can be used to solve a wide variety of problems including linear and nonlinear structural response, buckling, modal analysis, and fluid flow problems. Information about these elements and the type of analysis available in ANSYS can be found in various sources and manuals. **ANSYS Help** is also available interactively within ANSYS.

Author used ANSYS to extract the value of the stress intensity factor for a compact tension specimen ASTM E399-90 standard. Basically in this project, author had to perform a 2-D structural analysis then proceeds with 3-D structural analysis and reproduces the results indicated in equation 2.8.

The most important utilities that the author had used and considered in ANSYS are:

- a) Use PLANE 2 and SOLID 95 as singular elements
- b) Half specimen modeled using symmetry boundary conditions.
- c) KSCON command to automatically create local crack tip mesh with quarter-point nodes.
- d) Used “**skewed ¼ pt element**” to generate singular elements at the crack tip
- e) The **KCALC** command used to extract the value of the stress intensity factors

3.4 Procedures in using ANSYS

The procedures in using ANSYS can be summarizing by figure 3.2 and the details can be referring to Appendix 2.

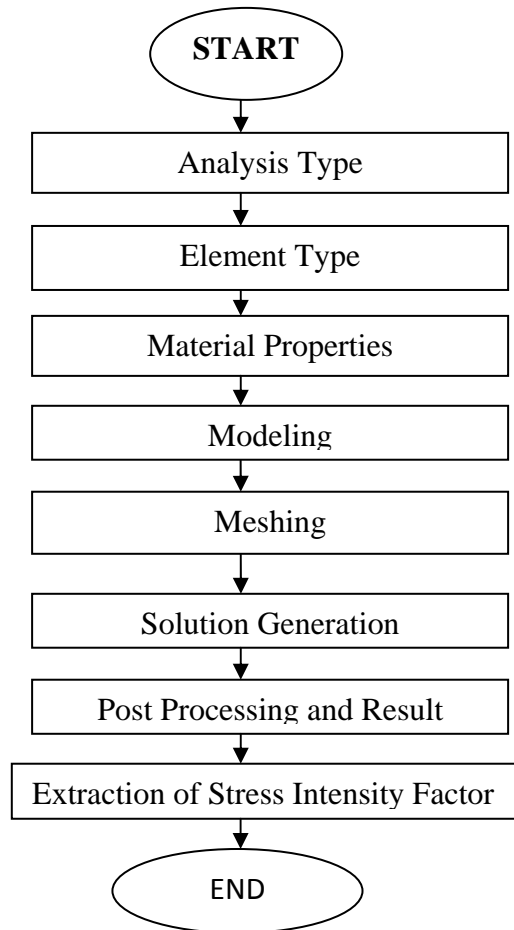


Figure 3.2: Procedure in ANSYS

3.4.1 Singularity element for 2-D Model

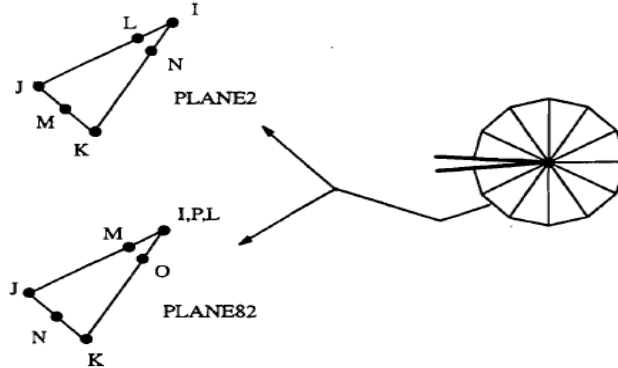


Figure 3.3: Singular element for 2-D models

The most important region in a fracture model is the region around the edge of the crack. The stresses and strains are singular at the crack tip, varying as $1/\sqrt{r}$. To pick up the singularity at the crack, the elements around the crack tip and crack front should be quadratic, with the middle nodes placed at the quarter points. Such elements are called singular elements.

The recommended element type for a 2-D fracture model is PLANE2 or called the Triangular 6node2 solid as show in figure 3.3. The element has a quadratic displacement behavior and is well suited to model irregular meshes.

3.4.2 Singularity element for 3-D element

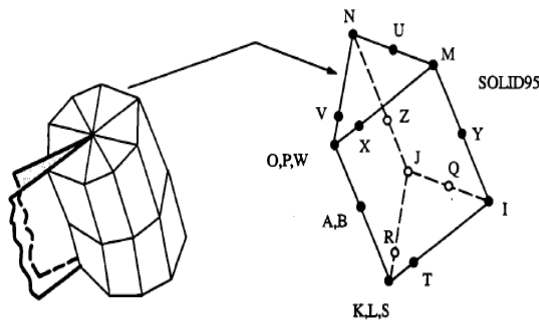


Figure 3.4: Singular element for 3-D model

The recommended element type for 3-D models is SOLID95 which is 20-node 95 element. This type of element is show by figure 3.4. It is a higher order element and it can tolerate irregular shapes without as much loss of accuracy. Besides that, SOLID95 elements have compatible displacement shapes. The first row of elements around the crack front should be singular elements. Generating a 3-D fracture model is considerably more involved than a 2-D model.

3.4.3 Mesh Generation

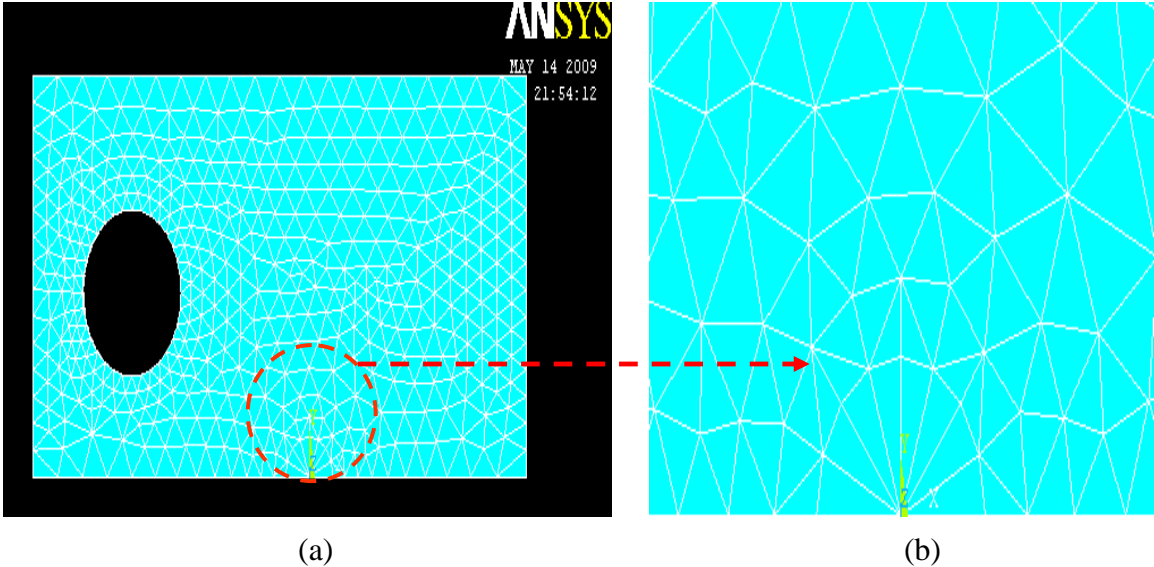


Figure 3.5: (a) Predefine mesh for 2-D model (b) Crack tip

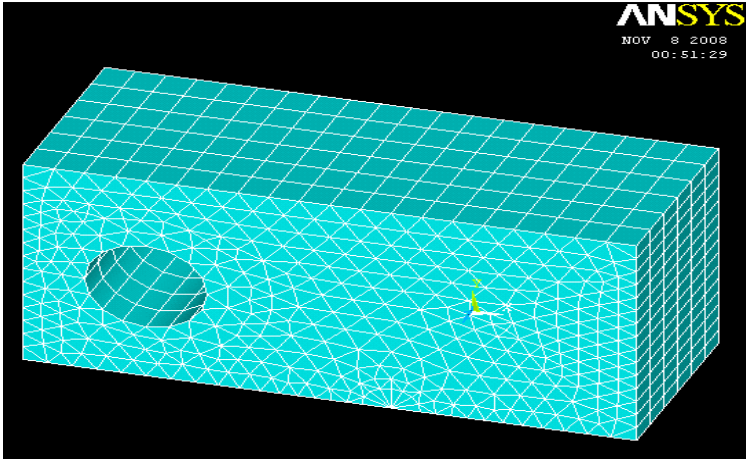


Figure 3.6: Predefine mesh for 3-D model

The most important procedure that will affect the result is mesh generation. Mesh generation applied after 2-D model successfully build and by using **KSCON** command:

Main Menu> Preprocessor> Meshing> Size Cntrl> Concentrate KPs> Create

This command is applied at the first row of elements around the crack tip. Crack tip should have radius of approximately $a/8$ or smaller (a is size of crack) while the number of element around the crack is eight to produce accurate result. To produce singularity at the crack tip, author had use “**skewed ¼ pt element**”.

Figure 3.5(a) shows meshing for 2-D model that used PLANE 2 element while figure 3.6 shows meshing for 3-D model that used SOLID 95 element. The typical mesh at the crack tip should look like presented by figure 3.5(b)

3.4.4 Solution Generation

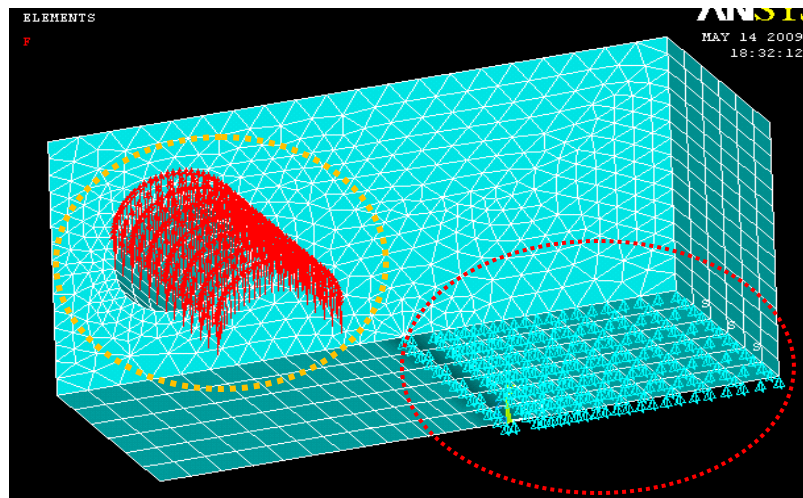


Figure 3.7: Define load and symmetry boundary condition

There are three command that should be consider in Solution Generation which are define load, define symmetry boundary condition and **SOLVE** that shows by figure 3.7. Referring to ASTM E399-90, load should be applied at the hole with positive direction. Load is indicated by the red arrow. The lower right corner of the rectangle is constraint,

so boundary condition should be apply at that area. Boundary condition is indicated by green arrows.

3.4.5 General Postprocessing

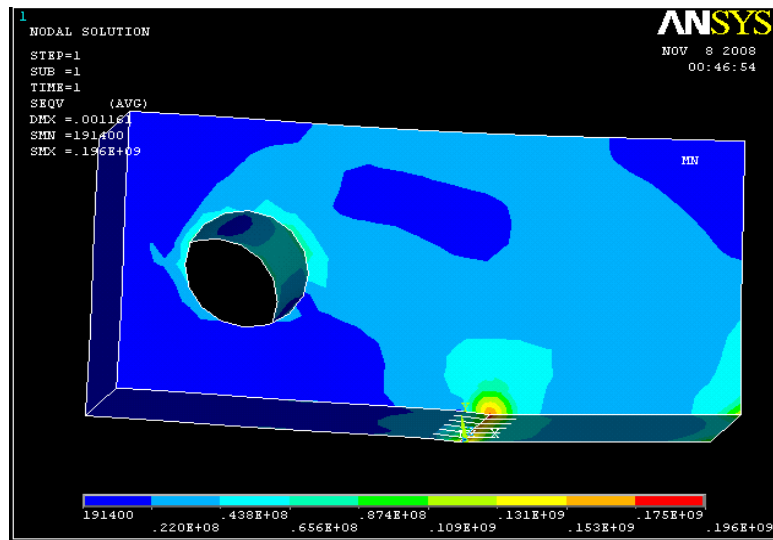


Figure 3.8: Von Mises Stress

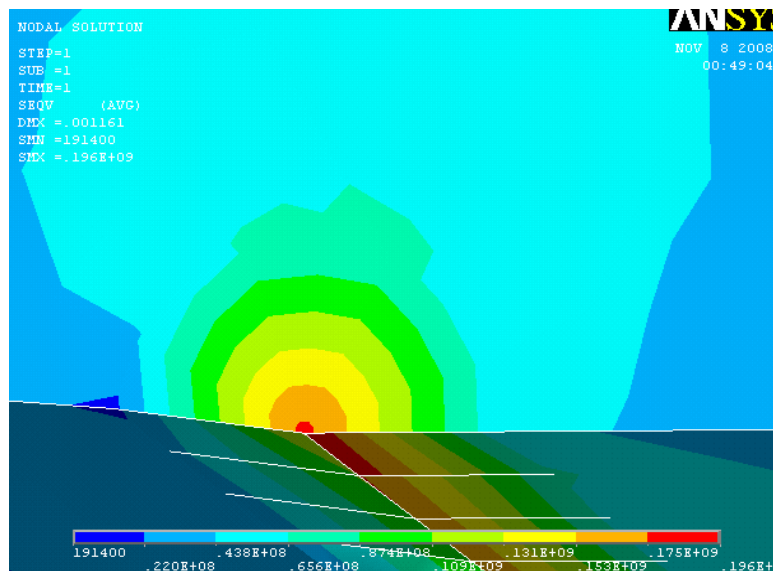


Figure 3.9: Stress distribution around the crack tip for compact tension specimen

General Post Processing command plot the result by using Von Misses stress and produce stress distribution at crack tip. The results show at figure 3.8 and 3.9, had proved that maximum stress occur at the crack tip of the specimen. The red color in the figure shows the maximum stress. It is because of the edge of the specimen cause by the crack which will lead to the sudden failure of material. The stress decrease as the distance from the crack tip is increase. Stress concentrate on the crack tip can lead to material failure if the stress intensity factor exceeds the fracture toughness value which is different according to material

3.4.6 Extraction of Stress Intensity Factor

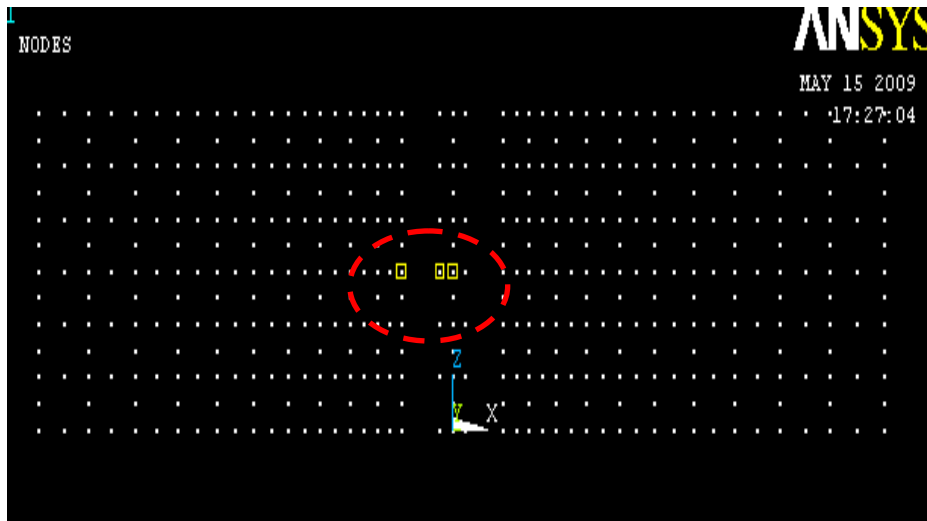


Figure 3.10: Define path at the crack tip

Extraction of stress intensity factor is the last step to produce the result. As explained in the appendix 2, the author had to define a path in the vicinity of the crack tip as showed in figure 3.10. Then pick successively three nodes behind the crack tip with the first one at the crack tip, the second close to the crack tip, and the third one a little further. Finally, extract the stress intensity factor with KCALC command as follow:

General Postprocessor < Nodal Calcs < Stress Intensity Factor