CHAPTER I

INTRODUCTION

1.1.   Introducing to The Tubular Structure

Tubular members are widely used in both onshore and offshore structures. Their attributes such as the high strength-to-weight ratio, low drag coefficient, and the ability to use their internal space have made them particularly useful in the offshore industry over many years. A typical example of the use of tubular members in an offshore situation is the fixed offshore platform (see Figure 1.1), where tubular members form a space frame to support the topside structure. Tubular connection design is a major factor in the design of a structure and can even be the limiting factor in terms of the strength of the structure.

Circular hollow sections are widely used in the construction of offshore structures in Malaysia. These sections offer many advantages over other sections. The sections have the ability to distribute load consistently. From the architect point of view, it has a minimum amount of surface area to unclean matter effect, rust and other spoil. With a circular form, it has an advantage in reducing the effect of wind, wave and blast loadings.
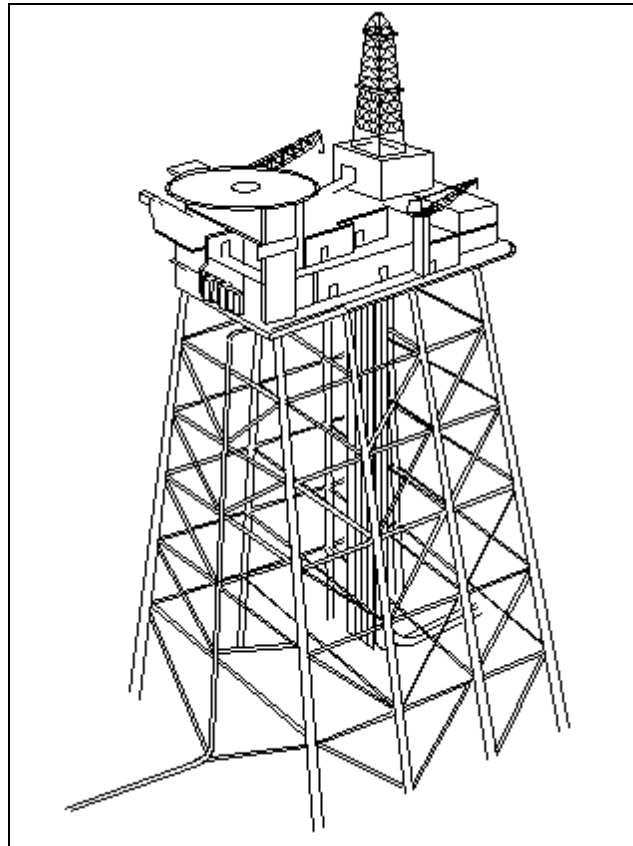
Figure 1.1: A typical jacket structure [1]

1.2.   Problem of Study

Structural tubular joints are widely used in the construction of offshore structures. As these structures are located in hostile environment, these joints represent structural weak spots and so it is desirable to develop reliable methods of determining their static collapse loads.

It is impractical to test actual joints due to their massive sizes and also in view of the associated testing costs. Testing small-scale steel joint models of various

shapes was widely carried out in the past. However, the manufacture of these joint models needs highly skilled welders and the exact shape of fillet welds is not repeatable.

An attractive alternative is to carry out finite element analysis using a suitable software to obtain the static strength results. If the results are good, this method can be used to performed a parameter study to investigate the effect various geometric parameters on a joint static strength.

1.3.   Objectives of Study

The static strength of tubular DT joints will be studied using LUSAS finite element software [2]. Objectives of the study are:

a)   to create a good finite element DT joint model.

b)   to carry out a mesh convergence study.

c)   to define the maximum load attained during the elastic plastic response of the joint.

d)   to compare the static strength results of tubular DT joint between finite element software and previous experimental test.

e)   to performed parameter study to investigate the effect of the various geometric parameters on the static strength of DT joints.

f)   to develop a simple formula to calculate the static strength of DT joints.

1.4.   Scope of Study

The LUSAS finite element software will be used to determine the static strength of tubular DT joint. In this study, the static strength of a tubular DT joint is defined as the maximum load attained during the elastic plastic response of the joint and this is shown in Figure 1.2. Data for the analysis are taken from a previous experimental testing on a similar DT joint performed by Kang et al.[3].
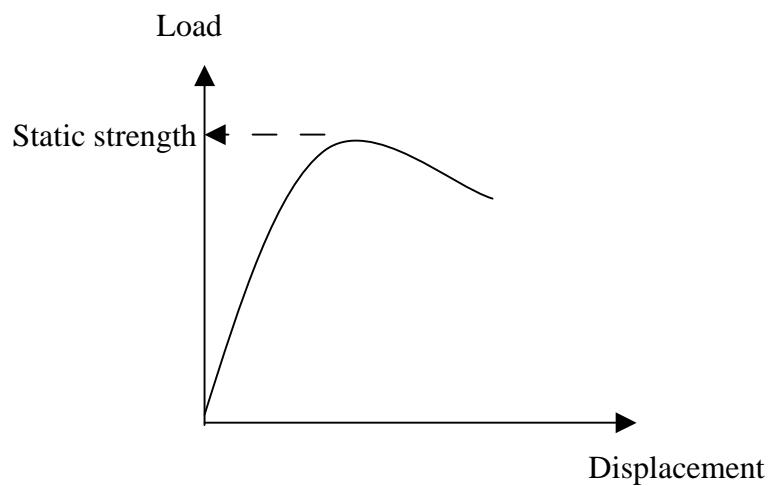

Figure 1.2: An elastic plastic response of the joint

To obtain the static strength by using LUSAS software, the dimensions of the DT joint finite element model had been based on the dimensions and data of the actual joint test performed by Kang et al.[3]. Before the comparison of the static strength result predicted by the LUSAS software and that of the actual test, a mesh refinement was conducted by performing non-linear analysis of the DT joint model under brace compression loading using different element density applied to the chord area near the brace wall.