CHAPTER 3

RESEARCH METHODOLOGY

3.1 General

The methodology for this research is going to be discussed in this chapter thoroughly by showing the step of analyzing I-beam girder of spans 20m, 10m and 5m. It is mainly about the finite element analysis. For this project, the use of ANSYS computer program was used to create and analyze the structure of this 3D structure. The analysis of the fail elements also can be analyzed by using this program. The material, cross section and element type of the I-beam are all been provided in the program. The specification of the simulation is shown in Table 3.1.

Table 3.1: Element type for working model

Specification (Element Type)	Used
Concrete	Solid65
Reinforcing steel	Link8
Pre-stressing steel	Link8

3.2 Finite Element Model

Finite element analysis program, ANSYS has been use in this study to simulate the behavior of 3 different spans of beams. It has its own specialized element and nomenclature and also procedures that have to be used properly.

In order to create the model of finite element analysis in ANSYS, there are multiple tasks which need to be completed in order for the model to run properly. These models can be created using command prompt line input or the Graphical User Interface (GUI) and GUI has been utilized to create the model. The rest of this chapter might discuss the procedure were used to model the beam. All these procedures include definition of real constants, material properties and parameters for nonlinear analysis.

3.2.1 Element Types

The element types for this model are shown in table 3.1. The solid65 element was used to model the concrete. This element has eight nodes with three degrees of freedom at each node with translations in the nodal x, y and z directions. This element is capable of plastic deformation, cracking in three orthogonal directions and crushing. A schematic diagram of this element is shown in Figure 3.1. Element type number 2 is refers to Link8 element. This type of element has been used for all steel reinforcement in the beam. It is assume to be bilinear isotropic. This bilinear isotropic is based on the von Mises failure criteria. It requires the yield stress and hardening modulus of the steel be known. This element is a 3D spar element and has two nodes with three degrees of freedom in the translation of x, y and z directions. This element is also capable of plastic deformation. This element is shown in Figure 3.2. The descriptions for each of the element type are actually laid out in the ANSYS element library.
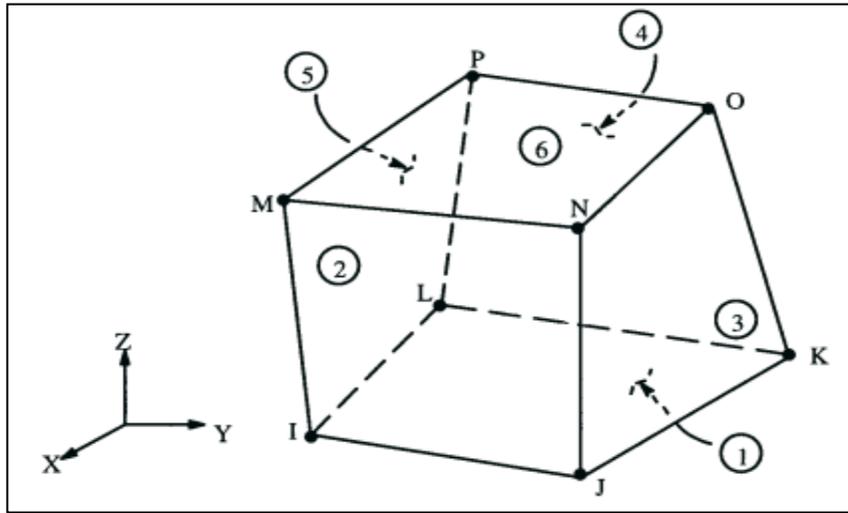
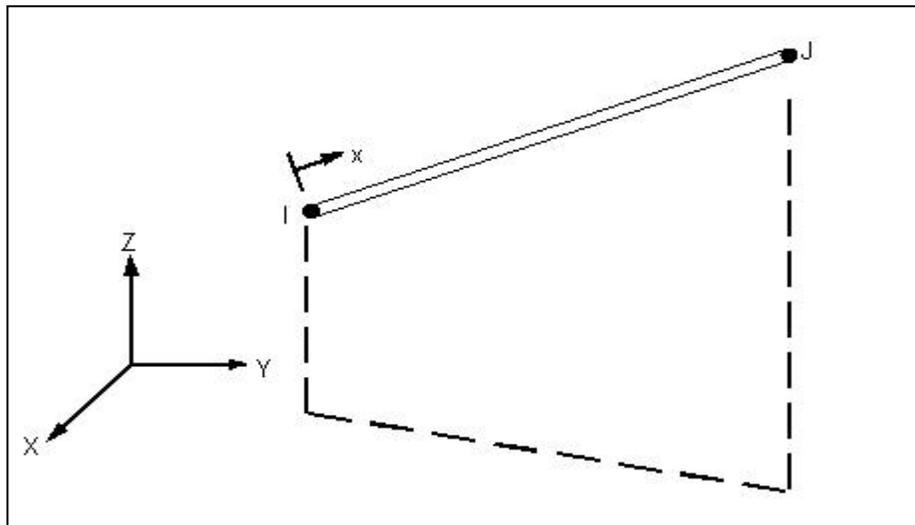Figure 3.1: Solid*65* – Reinforced concrete solid

Source: SAS, 2009

Figure 3.2: Link8 – Reinforcement of concrete

Source: SAS, 2009