

# **EXPERIMENTAL BASED REINFORCED CONCRETE BEAM ELEMENT MODELING AND FINITE ELEMENT ANALYSIS USING ABAQUS CAE**

Student Name :Nofri Bernardo

Nim : 4204191250

Responsibility : Zev Al Jauhari, MT & M. Gala Garcya, MT

## **Abstract**

Effective testing of concrete structures in the field requires a lot of accuracy and understanding of the response and behavior of concrete loads. Flexibility in beams tends to be smaller, so reinforcing steel is used to increase the displacement of the beams. The arrangement of the reinforcement is not in accordance with the requirements for the load received, so it is necessary to create a model according to the load to be applied without conducting multiple trials (trial errors). The abaqus CAE software is a modeling application that belongs to the Finite Element. Modeling reinforced concrete beams in the abaqus application for this study is expected to obtain results close to experimental tests in the field. That is a comparison between the experimental test results of reinforced concrete beams and the results of analysis in the abaqus CAE software .The results of flexure testing of reinforced concrete beams in the field obtained a maximum load value of 106.22 KN with a deflection of 1.126 mm. While the modeling of reinforced concrete beams in the abaqus application obtained a maximum load value of 146.203 KN with a deflection of 4.83 mm.

**Keyword** :Reinforced Concrete, *Finite Element*, Experimental Test