

# **MODELING OF REINFORCED CONCRETE BEAM WITH THE ADDITION OF SIKACIM CONCRETE ADDITIVE BASED ON EXPERIMENTAL AND FINITE ELEMENT ANALYSIS USING ABAQUS CAE**

Student Name : Septian Rizki Andi

Nim : 4204191246

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

## **Abstract**

Effective testing of concrete structures in the field requires a lot of precision and understanding the response and behavior of loads, namely the comparison between experimental test results of reinforced concrete beams and the results of analysis in the CAE abaqus software. Concrete with high quality is quite difficult to work. Therefore, the chemical Sikacim Concrete Additive is added to make work easier and maintain the quality of the concrete. This research will be a recommendation on the reinforcement placement arrangement that can increase the load value of the test object. Modeling reinforced concrete beams in the abaqus application for this study is expected to get results close to experimental tests in the field. The result with the addition of Sikacim Concrete Additive of 1% by weight of cement to the compressive strength of concrete has increased from normal concrete, namely from 25.81 MPa to 28.98 MPa. The results of flexure testing of reinforced concrete beams in the field obtained a maximum load value of 123.18 KN with a deflection of 1,07 cm. While the modeling of reinforced concrete beams in the abaqus application obtained a maximum load value of 155,738 KN with a deflection of 0.565 cm. the deflection of these 2 variations is a better recommendation compared to field testing and abaqus modeling without variations. The deflection in variation 1 is 0.485 cm with a load of 155.73 KN and variation 2 is 0.413 cm with a load of 178.477 KN.

**KeyWord:** Reinforced Concrete, *Finite Element*, Experimental Test.