BRIDGE SLAB STRUCTURE EVALUATION USING ABAQUS CAE FINITE ELEMENT MODELING

(Case Study: Bengkalis Bridge)

Name	: Benget Rio Sanses Simanjuntak
Student Number	: 4204201282
Advisor	: M. Gala Garcya, MT

ABSTRACT

The Slab Design for Bridge Deck needs to be strong enough to withstand the load that works during bridge service life. In bridge design, one of the factors that need attention is bridge displacement. The displacement value of the bridge must not exceed the permissible deflection value. Therefore, it is necessary to evaluate the displacement of the bridge. Bridge slab modeling using Abaqus CAE software in this research is expected to get results that are close to displacement tests in the field using the dial gauge tool. Based on research, displacement testing in the field using a dial gauge tool showed displacement results of 0.37 mm with a palm oil truck loading of 10.87 tons. Meanwhile, the results of the Abaqus CAE modeling obtained a displacement result of 0.59 mm with the same loads. This means that the bridge can still withstand a greater load because the permitted displacement value is 4.75 mm, which is converted to a field load of 139.55 tons and for the Abaqus CAE analysis of 87.51 tons.

Keywords: Bridge Experimental Test, Displacement, Finite Element Analysis, Slab Evaluation