Abstract

Windmill is one of the tools created to generate energy by utilizing wind gusts. With the wind blowing, the blades on the windmill will rotate due to the drag force and lift force. It should take a stronger structure to withstand the load and there should be a safety system to keep the tool running according to its function by protecting the components from excessive loads caused by wind. At this final project will discuss the design and structure of the windmill brake system to be installed on the bridge Suramadu conditioned on the largest wind speed.

The method used to design a safety system at this windmill is to simulate the windmills structure with ANSYS program and other programs that support, such as AutoCAD or CATIA. With the program will be known image of stress that occur in the structure of the windmill so that it can be determined from the structure and design of the brake system to be used for the windmill.

The resulti by designing this windmills is the construction of the structure and the brake system are suitable for attachment to a windmill that will be installed on the bridge Suramadu so windmills operate as its function and not damaged because the force caused by the wind that is received by the windmill.

Keywords: windmills, structure, ANSYS, brake systems
(Halaman ini sengaja dikosongkan)