

ABSTRACT

The assembly process of the sub-assembly frame components in a 70 m³ capacity vessel dump truck is currently performed manually without specialized positioning tools. This method causes the setting time to take up to 6 working hours per unit and carries a high risk of dimensional inaccuracy and structural deformation during the welding process. This research aims to design a jig setting tool to position the components automatically and precisely, analyze its structural strength, and estimate the reduction in assembly lead time. The tool design was carried out using Computer-Aided Design (CAD) software, with SS400 structural steel as the primary constituent material. The structural feasibility evaluation was analyzed through linear static simulation based on the Finite Element Method (FEM) using Computer-Aided Engineering (CAE) software. The loading simulation results indicate that the jig setting structure is safe and highly rigid. The maximum equivalent stress (Von Mises stress) occurring in the structure is 45,223 MPa, far below the yield strength limit of the SS400 steel material (245 MPa). The maximum total deformation is recorded at 0,14669 mm, which still meets the maximum assembly tolerance limit of 0,5 mm. Furthermore, the minimum safety factor value is at 5,41, which convincingly exceeds the required minimum safety limit of 1,25. Operationally, the implementation of this jig setting design is estimated to be able to reduce the total assembly time from 360 minutes to 180 minutes per unit. This represents a 50% increase in productivity and efficiency of the fabrication line.

Keyword: *finite element analysis, jig setting, tool design*

INTISARI

Proses perakitan komponen *sub assembly frame* pada *vessel dump truck* berkapasitas 70 m³ saat ini masih dilakukan secara manual tanpa alat bantu pemosisian khusus. Metode ini menyebabkan waktu penyetelan (*setting*) memakan waktu hingga 6 jam kerja per unit dan berisiko tinggi terhadap ketidakpresisian dimensi serta deformasi struktur saat proses pengelasan. Penelitian ini bertujuan untuk merancang alat bantu *jig setting* guna memosisikan komponen secara otomatis dan presisi, menganalisis kekuatan strukturnya, serta mengestimasi penurunan *lead time* perakitan. Perancangan alat bantu dilakukan menggunakan perangkat lunak *Computer-Aided Design* (CAD) dengan material penyusun utama baja struktural SS400. Evaluasi kelayakan struktur dianalisis melalui simulasi statis linier berbasis *Finite Element Method* (FEM) pada perangkat lunak *Computer-Aided Engineering* (CAE). Hasil simulasi pembebanan menunjukkan bahwa struktur *jig setting* dinyatakan aman dan sangat kaku. Nilai tegangan ekuivalen maksimum (*Von Mises stress*) yang terjadi pada struktur adalah sebesar 45,223 MPa, jauh di bawah batas tegangan luluh material baja SS400 (245 MPa). Deformasi total maksimum tercatat sebesar 0,14669 mm, yang masih memenuhi batas toleransi maksimal perakitan sebesar 0,5 mm. Selain itu, nilai rasio keamanan (*safety factor*) minimum berada di angka 5,41, yang secara meyakinkan melampaui batas aman minimum yang disyaratkan yaitu 1,25. Secara operasional, pengimplementasian rancangan *jig setting* ini diestimasi mampu memangkas total waktu perakitan dari 360 menit menjadi 180 menit per unitnya. Hal ini merepresentasikan peningkatan produktivitas dan efisiensi lini fabrikasi sebesar 50%.

Keyword: *finite element analysis, jig setting, perancangan alat*