

# Koresponden B IComera

SINTA | Auth: X | Edit item: Kor | Vol 5, No 1 | Akun Google | Email Terkirim | Akun Google | Email Terkirim | (35,382 unread) | +

mail.yahoo.com/d/search?keyword=icomera/messages/53808/AJFFkhAKqvYrYp1mwhkcFOIDko:2

HOME MAIL NEWS FINANCE SPORTS ENTERTAINMENT LIFE SEARCH SHOPPING YAHOO PLUS MORE...

yahoo/mail icomera Advanced

Compose

Inbox 35K  
Unread  
Starred  
Drafts 180  
Sent  
Archive  
Spam  
Trash  
Less  
Views Show  
Folders Hide  
+ New Folder  
A Sembiring 535  
AI 6  
Banpros 42  
BKSTM 567  
CNC praktikum 2  
Commonwealth life 52  
Conference 725  
danish 7  
dari spam1 32  
Diksi

Submission paper (5) Yahoo/Drafts

h7f1973@yahoo.com <h7f1973@yahoo.com>  
To: icomera.ub@gmail.com  
Fri, Jul 31, 2020 at 5:42 PM

Dear ICOMERA organizer,  
Herewith, I send our paper to joint in your conference ICOMERA 2020 in Malang. Please check our paper submission.  
Thank you very much.

Best Regards  
Hendra, Dr. Eng.  
Associate Professor Mechanical Engineering

ICOMERA-2020-Anizar Indriani.docx  
524 KB

ICOMERA MEUB <icomera.ub@gmail.com>  
To: h7f1973@yahoo.com  
Mon, Aug 3, 2020 at 3:43 PM

Yth. Bapak/Ibu Dosen,  
pertama kami menyampaikan terima kasih atas ketertarikan Bapak/Ibu di ICOMERA 2020. Sekarang kami sedang melakukan review dan pengecekan tuntas atas paper yang Bapak/Ibu kirimkan. Kami berharap untuk dapat menyerahkan proceeding ke IOP paling lambat Desember 2020 sehingga proceeding dari spam1

ICOMERA-2020-Anizar Indriani.docx Page 1 of 7

### Finite Element Method for Stress Analysis in the Frame Holder of Generator Translation and Rotation Motion on Vertical Direction Mechanism for Sea Wave Power Plant

Y. Subartono<sup>1</sup>, A. Indriani<sup>2</sup>, Hendra H.<sup>3</sup>, Rispani<sup>4</sup>, and Henedrita H.<sup>5</sup>  
<sup>1</sup>Industrial Engineering Dept, University of Bengkulu, Indonesia  
<sup>2</sup>Industrial Engineering Dept, University of Sultan Ageng Tjakra Cirebon, Indonesia  
<sup>3</sup>Industrial Engineering Dept, University of Bengkulu, Indonesia  
<sup>4</sup>Post Graduate Industrial Engineering, University of Nusa Tenggara, Indonesia  
<sup>5</sup>Dr. H. Subartono, Universitas Ibnu Sina, Bengkulu, Indonesia  
\*y.su@iainbh.ac.id

**Abstract:** Finite element method has been used to calculate stress analysis in many field of engineering or industry (metal, automotive, power plant and etc.). To reduce cost of product processing, FEM can be applied for design of product and also design of product based on selected following the requirement such: Requirement design of product is always good, effects and efficient based on demand and function of product. Stress analysis depends on the properties of material: load, static and dynamic, stress, processing of product or component. In the metal processing FEM is applied for stress analysis in the processing of component to selected good material due to load and thermal effect on the part. Also in metal processing for automotive component for selected thermal stress on the structure of engine component. In the power plant, stress analysis of the structure or frame holder of generator due to static and dynamic process. Dynamic of properties of material frame holder of generator is elastic, high stiffness, high compression resistance, high strength and etc. In this research focus on the stress due to static process in the frame holder of generator translation and rotation motion on vertical direction by using of material: cast iron, stainless steel, steel and thermoplastic resin has been used for varying of material. In previous study, frame holder of generator has been used for cast iron (CI) and horizontal direction of thrust force mechanism is applied in generator and get the maximum value load of cast iron of generator is a block size, the deflection value is 0.28 mm (7). By using the frame holder of generator with vertical direction of thrust force mechanism is found that the maximum stress on stress value is 11488.106 MPa and displacement is 0.79 mm for cast iron and using cast iron to frame holder of generator material.

**Keywords:** Material, Frame holder of Generator, FEM, Stress Analysis, Displacement, Vertical Direction Mechanism

#### 1. Introduction

Finite element method (FEM) is one method to calculate stress analysis in the design of product by simulation or mathematics modeling. FEM analysis can be found on component analysis in metal industry, automotive industry, manufacture, building, structure and etc. FEM can be applied for reduce cost of product manufacturing by making the better design of product using

SINTA | Auth: X | Edit item: Kor | Vol 5, No 1 | Akun Google | Email Terkirim | Akun Google | Email Terkirim | (35,382 unread) | +

mail.yahoo.com/d/search?keyword=icomera/messages/53808/AJFFkhAKqvYrYp1mwhkcFOIDko:2

HOME MAIL NEWS FINANCE SPORTS ENTERTAINMENT LIFE SEARCH SHOPPING YAHOO PLUS MORE...

yahoo/mail icomera Advanced

Compose

Inbox 35K  
Unread  
Starred  
Drafts 180  
Sent  
Archive  
Spam  
Trash  
Less  
Views Show  
Folders Hide  
+ New Folder  
A Sembiring 535  
AI 6  
Banpros 42  
BKSTM 567  
CNC praktikum 2  
Commonwealth life 52  
Conference 725  
danish 7  
dari spam1 32  
Diksi

Submission paper (5) Yahoo/Drafts

ICOMERA MEUB <icomera.ub@gmail.com>  
To: h7f1973@yahoo.com  
Mon, Aug 3, 2020 at 3:43 PM

Yth. Bapak/Ibu Dosen,  
pertama kami menyampaikan terima kasih atas ketertarikan Bapak/Ibu di ICOMERA 2020. Sekarang kami sedang melakukan review dan pengecekan tuntas atas paper yang Bapak/Ibu kirimkan. Kami berharap untuk dapat menyerahkan proceeding ke IOP paling lambat Desember 2020 sehingga proceeding dapat terbit di sekitar Februari 2021. Untuk memperlancar komunikasi, selanjutnya komunikasi diutamakan melalui whatsapp. Oleh karena itu perkenankan kami untuk meminta bantuan Bapak/Ibu untuk mengirimkan pesan ke WA panitia di nomor 083831102198 dengan isi sebagai berikut:  
Nama contact person:  
Asal institusi contact person:  
Nama semua author:  
Judul lengkap paper:  
Kami menyampaikan banyak terima kasih atas partisipasi Bapak/Ibu. Kita berharap tidak ada kendala komunikasi sehingga proceeding dapat diterbitkan sesuai jadwal di atas

Homat kami  
Panitia ICOMERA 2020  
> Show original message

h7f1973@yahoo.com <h7f1973@yahoo.com> To: icomera.ub@gmail.com  
Thu, Aug 13, 2020 at 1:10 PM

h7f1973@yahoo.com <h7f1973@yahoo.com>  
To: Hema Dewita  
Fri, Aug 21, 2020 at 8:15 PM  
> Show original message

ICOMERA-2020-Anizar Indriani.docx Page 1 of 7

### Finite Element Method for Stress Analysis in the Frame Holder of Generator Translation and Rotation Motion on Vertical Direction Mechanism for Sea Wave Power Plant

Y. Subartono<sup>1</sup>, A. Indriani<sup>2</sup>, Hendra H.<sup>3</sup>, Rispani<sup>4</sup>, and Henedrita H.<sup>5</sup>  
<sup>1</sup>Industrial Engineering Dept, University of Bengkulu, Indonesia  
<sup>2</sup>Industrial Engineering Dept, University of Sultan Ageng Tjakra Cirebon, Indonesia  
<sup>3</sup>Industrial Engineering Dept, University of Bengkulu, Indonesia  
<sup>4</sup>Post Graduate Industrial Engineering, University of Nusa Tenggara, Indonesia  
<sup>5</sup>Dr. H. Subartono, Universitas Ibnu Sina, Bengkulu, Indonesia  
\*y.su@iainbh.ac.id

**Abstract:** Finite element method has been used to calculate stress analysis in many field of engineering or industry (metal, automotive, power plant and etc.). To reduce cost of product processing, FEM can be applied for design of product and also design of product based on selected following the requirement such: Requirement design of product is always good, effects and efficient based on demand and function of product. Stress analysis depends on the properties of material: load, static and dynamic, stress, processing of product or component. In the metal processing FEM is applied for stress analysis in the processing of component to selected good material due to load and thermal effect on the part. Also in metal processing for automotive component for selected thermal stress on the structure of engine component. In the power plant, stress analysis of the structure or frame holder of generator due to static and dynamic process. Dynamic of properties of material frame holder of generator is elastic, high stiffness, high compression resistance, high strength and etc. In this research focus on the stress due to static process in the frame holder of generator translation and rotation motion on vertical direction by using of material: cast iron, stainless steel, steel and thermoplastic resin has been used for varying of material. In previous study, frame holder of generator has been used for cast iron (CI) and horizontal direction of thrust force mechanism is applied in generator and get the maximum value load of cast iron of generator is a block size, the deflection value is 0.28 mm (7). By using the frame holder of generator with vertical direction of thrust force mechanism is found that the maximum stress on stress value is 11488.106 MPa and displacement is 0.79 mm for cast iron and using cast iron to frame holder of generator material.

**Keywords:** Material, Frame holder of Generator, FEM, Stress Analysis, Displacement, Vertical Direction Mechanism

#### 1. Introduction

Finite element method (FEM) is one method to calculate stress analysis in the design of product by simulation or mathematics modeling. FEM analysis can be found on component analysis in metal industry, automotive industry, manufacture, building, structure and etc. FEM can be applied for reduce cost of product manufacturing by making the better design of product using