

**NUMERICAL STUDY OF CONCRETE ENCASED
STEEL COMPOSITE COLUMN SUBJECTED TO
CONCENTRIC LOADING**

Final Project Report
In fulfillment of the requirement for the degree
Bachelor of Civil Engineering

by:
I MADE GOVINDA BENDESA MAS
Student Number: 151316143



**INTERNATIONAL CIVIL ENGINEERING PROGRAM
DEPARTEMENT OF CIVIL ENGINEERING
FACULTY OF ENGINEERING
UNIVERSITAS ATMA JAYA YOGYAKARTA
YOGYAKARTA
2019**

STATEMENT

I signed below stating that the final project with the tittle :

**“ NUMERICAL STUDY OF CONCRETE ENCASED STEEL
COMPOSITE COLUMN SUBJECTED TO CONCENTRIC
LOADING ”**

It is the result of my own work and not a result of plagiarism for other people's work. Ideas, research data, and quotes directly or non-directly derived from writings or ideas of other expressly provided in this final project. If it is proven later that this final project is result of plagiarism, the graduation certificate that I received will be canceled and returned to Universitas Atma Jaya Yogyakarta.

Yogyakarta

Statement maker,



(I Made Govinda Bendesa Mas)

APPROVAL

Final Project Report

**NUMERICAL STUDY OF CONCRETE ENCASED STEEL
COMPOSITE COLUMN SUBJECTED TO CONCENTRIC LOADING**

by:

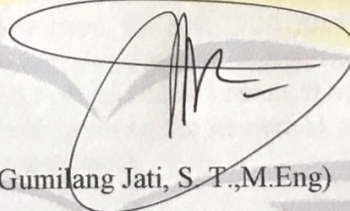
I Made Govinda Bendesa Mas

Student Number: 151316143

Has been approved by,

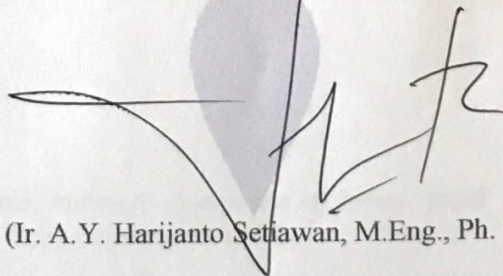
Yogyakarta, ... 20 - 1 - 2020

Supervisor



(Dinar Gumilang Jati, S. T., M. Eng)

Department of Civil Engineering Chairman,



(Ir. A. Y. Harijanto Setiawan, M. Eng., Ph. D.)

APPROVAL

Final Project Report

NUMERICAL STUDY OF CONCRETE ENCASED STEEL COMPOSITE COLUMN SUBJECTED TO CONCENTRIC LOADING

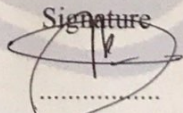
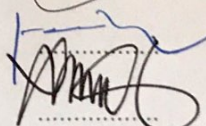
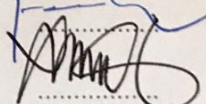


by:

I MADE GOVINDA BENDESA MAS

Student Number: 151316143

Has been examined and approved by:

	Name	Signature	Date
Chairman	: Dinar Gumilang Jati, S. T., M.Eng		20/1/2020
Member	: Dr. Ir. Ade Lisantono, M.Eng		January 21, 2020
Member	: Ir. J. Tri Hatmoko, M.Sc		21/01/2020

ACKNOWLEDGEMENTS

First and foremost, I would like to thank God for the blessing that given to me, so I can prepare and finish my final project report. The purpose of the final project with the title “Numerical Study of Concrete Encased Steel Composite Column Subjected to Concentric Loading” is to complete the requirement of undergraduate program (S-1) in International Civil Engineering Program, Department of Civil Engineering, Faculty of Engineering, Universitas Atma Jaya Yogyakarta. For the completion of this final project, I also would like to express my gratitude towards:

1. Dinar Gumilang Jati, S. T., M.Eng., as my supervisor, for his advice, patient, and kindness that given to me during the counseling. His support and advice have been invaluable.
2. Gung Mas, for the love, and relentless support so I can finish this final project.
3. My big family for the love, support, and advices that given to me to finish this final project.
4. All the lectures and staffs in the Civil Engineering Department, both in lecture and nonacademic for all the knowledge and guidance.
5. My friend in ICEP batch 2015, who encourage, accompany and support me to finish this final project.

6. Yopang, Deny, Ian, Joni and Tomy for all the help and advices that help me finish this final project.

Finally, I as the writer of this report realized that this report have many mistakes and may have space for further improvement. For that, any suggestion and critics will be accepted. Thank you.

Yogyakarta, January 2020

The Author

I Made Govinda Bendesa Mas
Student Number: 151316143

TABLE OF CONTENTS

COVER	i
STATEMENT	ii
APPROVAL.....	iii
APPROVAL.....	iv
ACKNOWLEDGEMTS	v
TABLE OF CONTENTS.....	vii
FIGURE LIST.....	ix
TABLE LIST	x
ABSTRACT.....	xi
CHAPTER I INTRODUCTION.....	1
1.1 Background	1
1.2 Problem Statement	2
1.3 The Objectives.....	2
1.4 Problem Limitation.....	3
1.5 Originality	3
CHAPTER II LITERATURE REVIEW.....	4
2.1 Concrete Encased Steel Composite Column	4
2.2 Design Codes.....	5
CHAPTER III BASIC THEORY.....	7
3.1 Composite Column.....	7
3.2 Steel Encased Composite Column	7
3.3 SNI 1729-2015	8
3.4 Eurocode 4.....	10
3.5 ABAQUS.....	11
3.5.1 Concrete Stress-Strain.....	12
CHAPTER IV METHODOLOGY	14
4.1 General	14
4.2 Procedure.....	14

4.3	Design Code	16
4.4	Simulation Study	16
4.4.1	Modelling of Concrete Encased Steel Composite Column.....	16
4.4.2	Compression Test using ABAQUS.....	17
CHAPTER V RESULT AND DISCUSSION		18
5.1	Composite Column.....	18
5.2	Design Code Prediction of Compressive Strength Capacity.....	19
5.2.1	SNI 1729-2015.....	20
5.2.2	Eurocode4	20
5.3	Simulation Study with ABAQUS.....	21
5.3.1	Part	21
5.3.2	Material Property	23
5.3.3	Assembly.....	24
5.3.4	Step.....	25
5.3.5	Interaction	25
5.3.6	Load and Boundary	25
5.3.7	Mesh.....	25
5.3.8	Job	26
5.4	Design Code Axial Compression Capacity Result.....	26
5.4.1	SNI 1729-2015.....	26
5.4.2	Eurocode4	27
5.5	ABAQUS Result	28
5.6	Combined Result	30
CHAPTER VI CONCLUSION.....		31
6.1	Conclusion.....	31
6.2	Suggestion	32
REFERENCES		33

FIGURE LIST

No	Figure Name	Page
2.1.	Concrete encased composite column cross-section (Soliman et	5
2.2.	al., 2013)	5
3.1.	Specimen C4 failure in experiment	12
3.2.	Concrete stress-strain under compression	13
4.1	Concrete stress-strain under tension	15
5.1	Research Flow chart	19
5.2	Composite column type	22
5.3	Concrete part	22
5.4	Steel part	23
5.5	Reinforcement bar and transverse ties part	23
5.6	Plate part	25
5.7.	Part assembly	28
5.8.	Previous experiment and ABAQUS failure region	29
5.9.	Load of the displacement	29
5.10.	Longitudinal Displacement	29
5.11	ABAQUS load displacement graph	30
	Result comparison	

TABLE LIST

No	Table Name	Page
2.1.	Experiment results (Soliman, Arafa, & Elrakib, 2013)	4
2.2.	Comparison experiment result and design code (P_u/P_{calc}) (Soliman	6
4.1.	et al., 2013) Steel Dimension	16
4.2.	Material Properties	17
5.2.	Material Properties	24
5.3.	Result	30
6.1.	Result comparison	31

ABSTRACT

NUMERICAL STUDY OF CONCRETE ENCASED STEEL COMPOSITE COLUMN SUBJECTED TO CONCENTRIC LOADING. I Made Govinda Bendesa Mas, Student Number: 151316143, Year 2019, Structure Concentration, International Civil Engineering Program, Faculty of Engineering, Universitas Atma Jaya Yogyakarta

This final project aim is to compare the axial load capacity of concrete encased steel composite column using two different design codes and conducting numerical analysis, where the result was verified with previous experiment. For the design codes, SNI 1729-2015 and Eurocode4 were used in this final project. ABAQUS software was used for the numerical analysis and conducting compression test. In this final project one specimen from previous experiment, where all the property and dimension is the same. ABAQUS result and design code was compared with previous experiment result.

This final project used secondary data from previous research about concrete encased steel composite column subjected to concentric loading. The column was a square cross section with 200 x 200 mm dimension and 1400 mm of height. This column were longitudinally reinforced by 4 bars of 12 mm diameter high-grade steel with 8 mm of stirrups diameter. The steel section dimension were 100 mm depth of steel section and 50 width of flange with 6.6 thickness of flange and 4.5 thickness of web.

The result of this final project were the predicted axial load capacity using design codes equation were conservative, the SNI 1729-2015 result was 938 KN and Eurocode4 result was 957 KN, where the experiment result was 1050 KN. When the design code compared with the numerical analysis result, the numerical analysis gave closer result to the experiment result, where the numerical analysis result was 1108 KN or 5% higher than the experiment result.

Keywords: concrete encased composite column, axial compression capacity, numerical analysis, design codes, ABAQUS.