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

ACKNOWLEDGEMTS

First and foremost, I would like to thanks to 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

FIGURE LIST

TABLE LIST

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.

CHAPTER I

INTRODUCTION

1.1 Background

The increasing number of infrastructures construction also make the development of new type of column. Column/pier are one important element of infrastructure

Since the construction industry developed significantly, the demands for better structural members that better in strength were increased. Column is of the main structural member, where column will transmit the weight of the structure above through compression to structure below. So, in other word column is a compression member. There are two type of column that commonly used in coinstruction, reinforced concrete column and steel column. Many researchers started developed new type column that combine the rigidity and stiffness of concrete with the strength and ductility of steel section. Steel Encased Concrete composite column and concrete filled steel tubular column are the common type of composite column that been used in construction project.

Steel encased concrete composite column will be used in this final project. It offers high strength ductility, fire protection for the steel encased section and more simple in beam and column connection, with concrete encased steel composite columns contractor can also design smaller cross section area of column.

Contractor also competing to find more efficient and accurate way to do designing of member in construction. In every designing process, testing is need to determine whether the material is feasible or not. For concrete composite column compression test is needed to maintain the quality of the material. The tests are usually carried out using several sample of concrete and steel section. The cost for testing concrete and steel section is not cheap.

The solution for this problem were to predict the axial strength of the column using design code and using simulation study. In order to design a safe and proper composite column, manual or code is needed. SNI 1729-2015 is the standard or code that is used in Indonesia, the objective of this final project is to get better understanding on how accurate SNI 1729-2015 in predicting the axial compression of steel encased composite column. SNI 1729-2015 will also compared with Eurocode4 (EC4). Different methods for the design of composite column will resulted in different result of composite column axial capacity. For the simulation study ABAQUS was used to design the model and conduct the compression of the sample.

In this final project the result of the design codes and simulation study was compared to find which one give the most accurate axial compression capacity of the concrete encased steel composite column. If the result give small difference with the previous experiment result (Soliman, Arafa, & Elrakib, 2013), design codes and simulation study could decrease the cost for testing in laboratory.

1.2 Problem Statement

The problem that will discussed in this final project are how different is SNI 1729-2015 compare with Eurocode4, how to verify the compression test in laboratory using ABAQUS software and to compare the peak load from ABAQUS result with design code result in predicting the ultimate strength of concrete encased steel composite column.

1.3 The Objectives

The objective of this final project is:

1. To know compare SNI 1729-2015 axial compression capacity result with Eurocode4.
2. To compare the design code result with numerical analysis result in predicting axial load capacity of concrete encased steel composite column.
3. To verify numerical analysis result with experiment result.

1.4 Problem Limitation

The limitation of this project are:

1. This project will only focus on the encased composite column with inner I-section steel.
2. The dimension and properties of specimens will the same based on (Soliman et al., 2013) experiment.
3. This project will only focus on axial load capacity under centric axial compression.

4. ABAQUS was used to do compression test of concrete encased steel composite column.
5. SNI 1729-2015 and Eurocode4 equation will be used in this final project.
6. This final project using short column for the specimen.

1.5 Originality

The scope of this final project are to identify which methods will provide the best prediction about the axial compression capacity of steel encased composite column, review the difference between the codes and the verifying the ABAQUS result with previous experiment. This final project originated from the Review of Design Codes of Concrete Encased Short Columns under Axial Compression by (Soliman et al., 2013). The difference of this final project with the previous research are in this final project the code that will be reviewed and analyse are (SNI-1729, 2015) and Eurocode 4, this final project also conducting simulation using ABAQUS.