Structural Analysis: Pipe Rack of Segment Cc1-Cc3

Rizal Purnawan

Civil Engineer, Independent Researcher, Jakarta, Indonesia

Corresponding author. E-mail: rizalpurnawan23@gmail.com; ORCID: 0000-0001-8858-4036

14 November 2023

Abstract

This document contains the structural analysis report of a design of Pipe Rack Structure of segment Cc1-Cc3 in Smelter Ferronickel Kolaka held by PT. PP (Persero) Tbk. This document is intended for a portfolio to demonstrate the competency and experience of the author. We do not intend to share sensitive information in the project, and hence, we do not disclose as much sensitive information as possible.

Contents

1 INTRODUCTION

This document presents the calculation sheet for the design and analysis of Piperack grid Cc1-Cc3 of Project Smelter Ferronickel Kolaka. All the preliminary data used for conducting the design and analysis of the structure as well as the substructure are presented in this section as follows.

1.1 Codes, Standards and Other Data

The following codes and standards are legitimate for the use in this calculation and design:

A. Codes

SNI 1726:2019	Precedures of Design and Planning for Earthquake-Resistant Structures of Building and Non-Building (*Tata Cara Perencanaan Ketahanan Gempa untuk Struktur BangunanGedung dan Nongedung*)
SNI 1727:2020	Minimum Design Loads and Associated Criteria for Buildings and Other Structures (*Beban Desain Minimum dan Kriteria Terkait untuk Bangunan Gedung dan Struktur Lain*)
SNI 2847:2019	Specification of Concrete Structures for Buildings and Explanations (*Persyaratan Beton Structural untuk Bangunan Gedung dan Penjelasan*)
SNI 1729:2020	Specification of Steel Structures for Buildings (*Spesifikasi untuk Bangunan Gedung Baja Struktural*)
SNI 2052:2017	Concrete Steel Reinforcement (*Baja Tulangan Beton*)
SNI 8460:2017	Requirements for Geotechnical Designs (*Persyaratan Perancangan Geoteknik*)
JIS G3101 SS400	Spesification of Steel Material
ASCE 7-16	Minimum Design Loads and Associated Criteria for Buildings and Other Structures
ACI 318-19	Building Code Requirements for Structural Concrete
ASTM-A615	American Society for Testing Material: Specification for Carbon Steel
AISC 360-16	Specification for Structural Steel Buildings
AISC 341-16	Seismic Provisions for Structural Steel Buildings

B. Other Data

Soil Investigation	Soil Investigation for Ferronickel Smelter Development Project Kolaka, Southeast Sulawesi Indonesia (PT. SOILENS)
PSHA	Seismic Hazard Assessment for Ferronickel Smelter Factory Development Project Kolaka, Southeast Sulawesi Indonesia
922A-E941-023-2	Pipeline Layout Flowchart (REV C)
922A-E941-023-18	Section of Piping Arrangement
922A-E941-023-1	Line List Comprehensive Pipelines Network (Area 023)
922A-E941-023-11	Pipeline Weight Table Comprehensive Pipelines Network (Area 023)

1.2 Units of Measurements

The units of measurements used in this document are the International System of Units, known as SI (*Système International*).

1.3 Material Specifications

Table 1: Material Specifications

Material	Property	Notation	Measure	Unit
Concrete Structure	Compressive Strength	f_c'	29.00	MPa
Leveling Concrete	Compressive Strength	f_c'	12.00	MPa
Reinforcing Deformed Bar	Yield Strength	f_y	420	MPa
Reinforcing Deformed Bar	Tensile Strength	f_u	525	MPa
Structural Steel	Yield Strength	F_y	245	MPa
Structural Steel	Tensile Strength	F_u	400	MPa

1.4 Unit Weights

Table 2: Unit weights

Material	Measure	Unit
Reinforced concrete	23.50	kN/m^3
Leveling concrete	21.60	kN/m^3
Steel	76.99	kN/m^3
Water	9.81	kN/m^3

1.5 Computational Software

The following software and programs are used in the development of this document and its contents:

a. SAP2000
SAP2000 is used for modelling the upper structure as well as the structural components of the foundation. Members capacity control is also conducted using the dedicated algorithm within SAP2000.

b. Microsoft Excel
Excel is used for conducting manual calculations such as calculating the values of the input loads for the model,for computing the reinforcement of pile cap, computing the connections of steel members and computing the foundation stability analysis.

c. Overleaf LaTeX
This report is arranged in Overleaf, a collaborative cloud-based LaTeX editor used for writing, editing and publishing scientific documents. In addition, Overleaf is open-source.

2 METHODOLOGY

The upper structure and the substructure will be analyzed and modelled separately in SAP2000. The design and analysis of the upper structure will be conducted with SAP2000 namely to execute modal analysis, execute seismic analysis, assess the capacity of the members and serviceability evaluation such as deflections. The analysis for the joint connections will be conducted manually in spread sheets. After the modelling of the upper structure, we model the foundation in SAP2000 to compute the soil pressure due to bearing mechanism. Additional analysis of the foundation such as evaluations on overturning and sliding is conducted manually in spread sheets.

Upper Structure Work Flow

1. START
2. Collect all preliminary data including arrangement drawing, loading data, material specifications, code, standards, etc.
3. Generate the preliminary design.
4. Model the upper structure in SAP2000.
5. Input all loading conditions, fine tune the structural requirements such as member moment releases, and modal case setting including determining the estimate number of modes.
6. Run analysis in SAP2000.
7. Perform modal analysis.
   If the modal participating mass ratio is close to 1 (exceeds 0.9), proceed.
   Else, increase the number of modes and repeat from step 5.
8. Perform seismic analysis.
   Base shear control:
   If sufficient, then proceed.
   Else increase the scale factor for the response spectrum and repeat from step 5.
   Storey drift control:
   If sufficient, then proceed.
   Else increase the scale factor for the response spectrum and repeat from step 5.
9. Perform vertical deflection control:
   If sufficient, then proceed.
   Else, increase dimensions of the members or modify the configuration and repeat from step 3.
10. Perform the member capacity evaluation:
    If sufficient, then proceed.
    Else, increase dimensions of the members or modify the configuration and repeat from step 3.
11. DONE

Substructure Work Flow

1. Prepare the preliminary data including the support reaction of the upper structure.
2. Generate the preliminary design for the foundation.
3. Model the foundation in SAP2000 using foundation properties.
4. Input all loading conditions from the support reaction of the upper structure.
5. Run analysis in SAP2000.
6. Evaluate bearing capacity.
   If sufficient, then proceed.
   Else increase the dimension or the depth of the foundation and repeat from step 2.

7. Evaluate overturning.
   If sufficient, then proceed.
   Else increase the dimension or the depth of the foundation and repeat from step 2.
8. Evaluate sliding.
   If sufficient, then proceed.
   Else increase the dimension or the depth of the foundation and repeat from step 2.
9. DONE

2.1 Design Criterion

The following criterion shall be met in the design:

1. Loads to be considered in the design include dead loads, superimposed dead loads, piping loads consisting of the pipe materials as well as the fluid content within the pipe, wind loads and seismic loads. Initially, three schema are taken into account which consist of the erection stage (PE), the operating stage (PO) and the test stage (PT). However, the test used for the pipe is the hydro-test, which uses the same fluid (water) and the same circumstance for PO, thus, PT is omitted and we are left with two schema namely PE and PO. PE considers the weight of the pipe materials only without its fluid content. While PO considers the weight of the pipe materials as well as its fluid contents.
2. The analysis method of Allowable Strength Design (ASD) is used for evaluating the substructure.
3. The analysis method of Load Resistance Factor Design (LRFD) is used for evaluating the upper structure. The design of steel structure as well as the connections use SNI 1729:2020 as well as AISC 360-16. While the design of concrete structure such as pedestals and sloof use SNI 2847:2019 as well as ACI 318:19.
4. The upper structure is modelled in SAP2000 as 3D frame structure by considering the structure being an ordinary concentrically braced frames (OCBF) in both horizontal axes directions.
5. SAP2000 incorporates 2 types of conventions regarding coordinate systems, the X-Y-Z system and the 1-2-3 system. The global coordinate system uses the X-Y-Z while the local coordinate uses the 1-2-3. For the global coordinate system, the Z-direction refers to the vertical direction while X-Y plane refers to the horizontal plane. The mathematical concept underlying this convention is the concept of 3 dimensional Euclidean vector space $\mathbb{R}^3$ (Rudin, 1976).

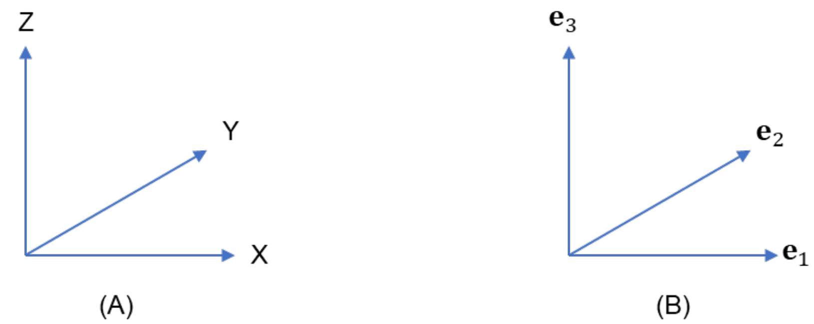


Figure 1: Equivalence of SAP2000 coordinate systems (A) with the standard basis of $\mathbb{R}^3$ (B)

2.2 Overview of the Structure

The location of the pipe rack is as indicated in figure 2. A more detail arrangement of the pipe rack is presented in figure 3. It can be seen from the figure that the piperack has a longer span of 16 meters, and another span of 4 meters hanging on the larger pipe rack. The cross sectional arrangement of the pipe rack is presented in figure 4.

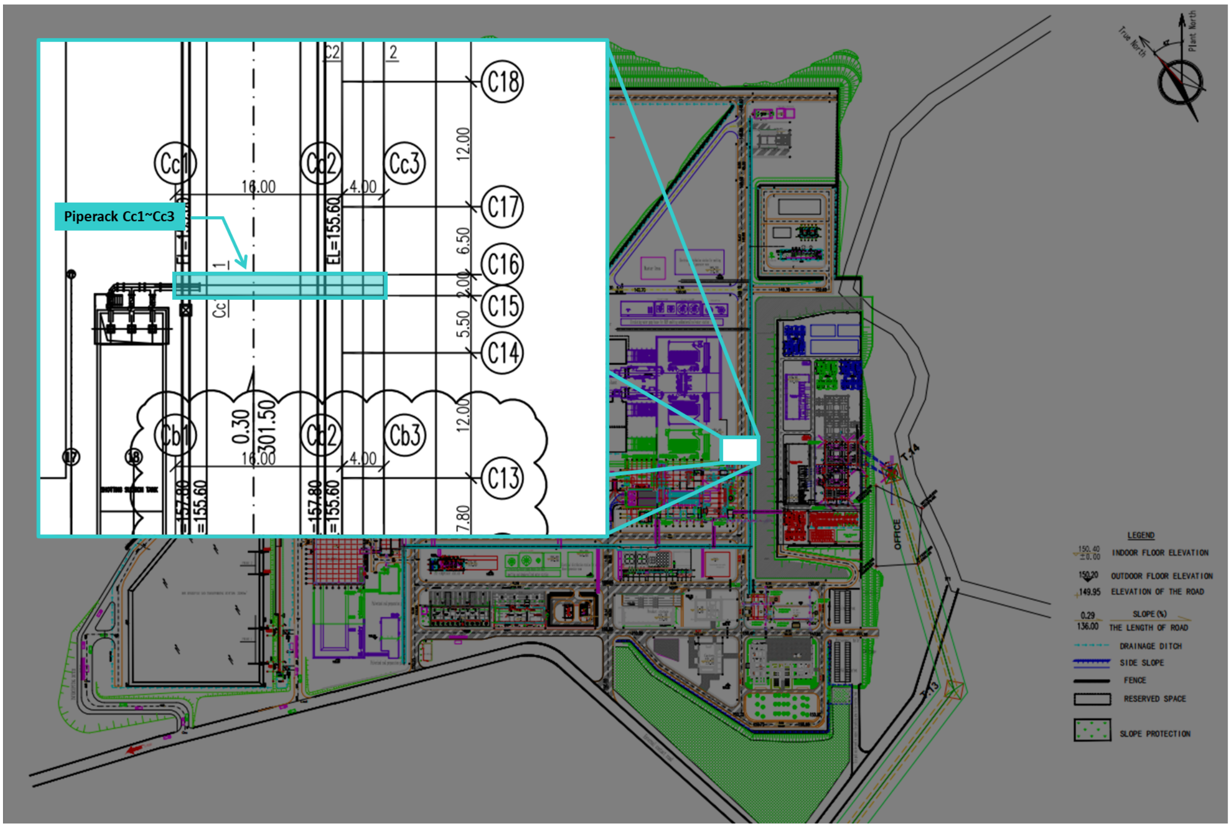


Figure 2: Location of Pipe Rack Cc1-Cc3

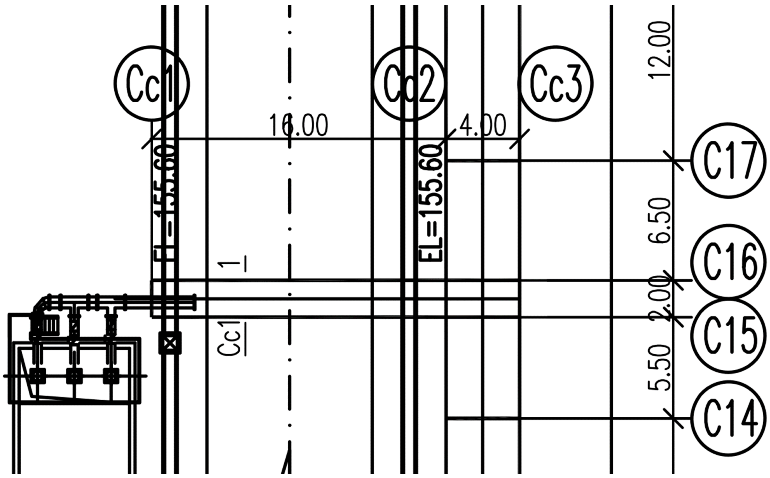


Figure 3: A more detail arrangement of Pipe Rack Cc1-Cc3 from the input data

It can be seen from the cross sectional arrangement in figure 4 that the only additional load in terms of the structure's main functionality is a pipe with a code 023-E941-CR4-0009-600-BACD1.

2.3 Fundamental Loads

The fundamental loads applied to this structure are described as follows. A notable information is that the live load does not present in the loading data (observe figure 4). Thus it is neglected.

1. Dead Load

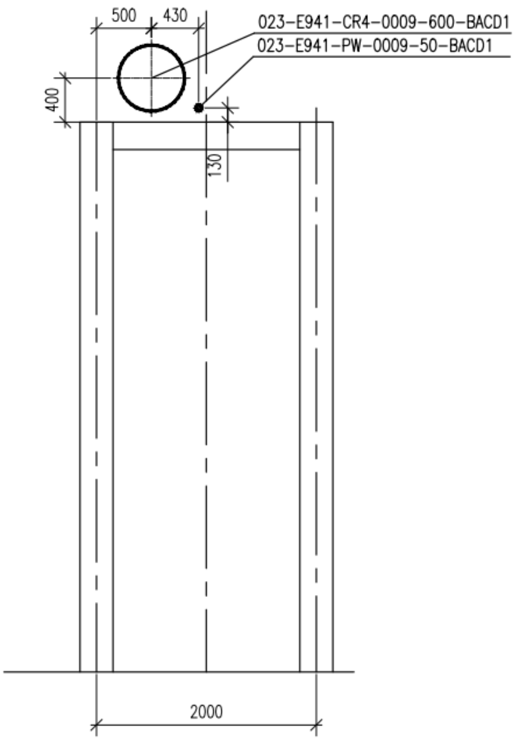


Figure 4: Cross sectional arrangement of Pipe Rack Cc1-Cc3 from the input data

The dead load is defined as the self weight of the structure, additional loads such as superimposed loads as well as the piping loads, which are considered permanently attached to the structure. The piping load PE and PO is assumed to be dead loads. And thermal expansion equivalent load of the pipe is also assumed to be a dead load acting in the horizontal direction. The dead load is denoted by **D**.

2. Wind Load
   The design for wind loads will follow the guideline in SNI 1727:2020 section 26 and section 27. Two orthogonal horizontal wind direction will be considered as well as their opposite directions. These decompositions of the wind loads will be denoted by **Wx**[+], **Wx**[-], **Wy**[+], **Wy**[-].

3. Seismic Load
   The loading condition for earthquake uses the response spectrum analysis. The method for generating the spectrum function will be described comprehensively on a later section. The seismic consideration will follow the guidelines in SNI 1726:2019. The directions of interest for the seismic loads are X and Y direction of the global coordinate system. And the seismic loads are denoted by **EQ-X** and **EQ-Y**. However, in the seismic load combinations, the seismic load cases to be included are **Ev** (vertical earthquake) and **Eh** (horizontal earthquake). SNI 1726:2019 section 7.4.2.1 asserts that

$$\mathbf{Eh} = \rho Q_E \,,$$

where ρ is the redundancy factor and Q_E is the horizontal seismic load, that is, $Q_E =$ **EQ-X** or $Q_E =$ **EQ-Y**. And SNI 1726:2019 section 7.4.2.2 asserts that

$$\mathbf{Ev} = 0.2 S_{DS} \mathbf{D} \,,$$

where S_{DS} is the short period spectral acceleration parameter obtained from the PSHA study. On the other hand, the seismic load case in terms of overstrength factor **Emh** is given by

$$\mathbf{Emh} = \Omega_0 Q_E \,,$$

where Ω_0 is the overstrength factor, determined using SNI 1726:2019 table 12.

4. Modal
   Modal analysis is required to determine the fundamental period of structures as well as their natural behaviour. SAP2000 provides a dedicated computational environment to execute modal analysis.

2.4 Load Combinations

The load combinations in use are set in accordance with SNI 1727:2020 as well as the project design basis. For the upper structure, the load combinations are arranged in accordance with SNI 1727:2020 section 2.3.1 for base combinations and section 2.3.6 for the seismic effect including the overstrength factor. While for the substructure, the load combinations are arranged in accordance with SNI 1727:2020 section 2.4.1 for the base combinations and section 2.4.5 for the seismic effect including the overstrength factor.

1. LRFD Combinations

Load Combination	Remark
1.4**D**	
1.2**D** + 1.0**W**	The wind load **W** will include **Wx**[+], **Wx**[-], **Wy**[+] and **Wy**[-].
0.9**D** + 1.0**W**	The wind load **W** will include **Wx**[+], **Wx**[-], **Wy**[+] and **Wy**[-].
1.2**D** + **Ev** + **Eh**	Seismic without overstrength. **Eh** will consider both X and Y directions.
0.9**D** - **Ev** + **Eh**	Seismic without overstrength. **Eh** will consider both X and Y directions.
1.2**D** + **Ev** + **Emh**	Seismic with overstrength. **Emh** will consider both X and Y directions.
0.9**D** - **Ev** + **Emh**	Seismic with overstrength. **Emh** will consider both X and Y directions.

2. ASD Combinations

Load Combination	Remark
D	
D + 0.6**W**	The wind load **W** will include **Wx**[+], **Wx**[-], **Wy**[+] and **Wy**[-].
D + 0.75(0.6**W**)	The wind load **W** will include **Wx**[+], **Wx**[-], **Wy**[+] and **Wy**[-].
0.6**D** + 0.6**W**	The wind load **W** will include **Wx**[+], **Wx**[-], **Wy**[+] and **Wy**[-].
1.0**D** + 0.7**Ev** + 0.7**Eh**	Seismic without overstrength. **Eh** will consider both X and Y directions.
1.0**D** + 0.525**Ev** + 0.525**Eh**	Seismic without overstrength. **Eh** will consider both X and Y directions.
0.6**D** - 0.7**Ev** + 0.7**Eh**	Seismic without overstrength. **Eh** will consider both X and Y directions.
1.0**D** + 0.7**Ev** + 0.7**Emh**	Seismic with overstrength. **Eh** will consider both X and Y directions.
1.0**D** + 0.525**Ev** + 0.525**Emh**	Seismic with overstrength. **Eh** will consider both X and Y directions.
0.6**D** - 0.7**Ev** + 0.7**Emh**	Seismic with overstrength. **Eh** will consider both X and Y directions.

3 MODELLING AND DESIGN

This section will cover the modelling of the upper structure and the substructure as well as the method of determining the considered loads together with their input in the model.

3.1 Model of the Upper Structure

The model of the upper structure is composed of 3D space frames which includes steel structures for the truss members and concrete structures for the pedestal and the sloof. In both the horizontal axes, the system of ordinary concentrically braced frames (OCBF) is used. The overview of the model is presented in figure 5.

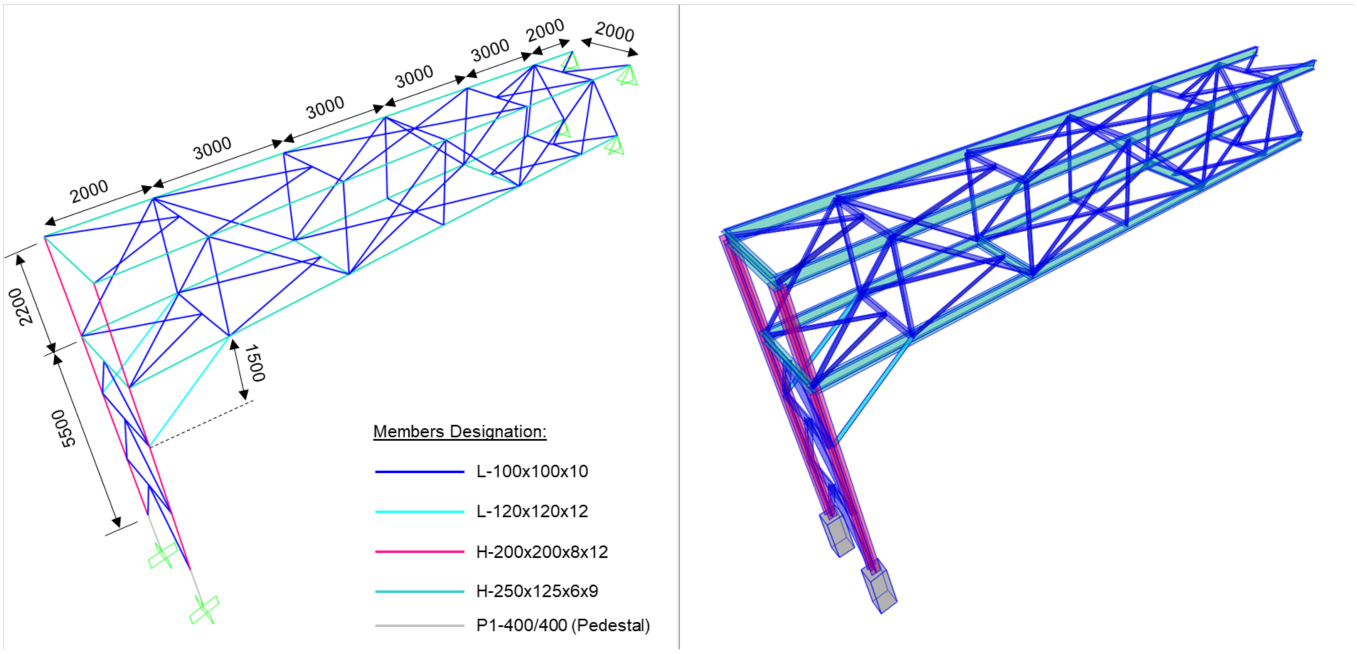


Figure 5: Overview of the Pipe Rack Cc1-Cc3 upper structure model in SAP2000

At the right end, (as indicated) the model is supported to the connecting main pipe rack. The supports in the model are as illustrated in figure 5. As a preliminary, the dimensional configurations of the frame elements in use are as presented in figure 5. A detail list of the dimensions of the steel profile is presented in table 3.

Table 3: Dimensional configurations of the upper structure

Section	Type	Remark
L-100x100x10	L-Section	Longitudinal truss members, vertical column braces
L-120x120x12	L-Section	Diagonal braces
H-200x200x8x12	H-Section	Column members
H-250x125x6x9	H-Section	Longitudinal members, transversal beams
P1-400x400	Concrete rectangular section	Pedestal

The braces are expected not to withstand moments. And the both ends of the horizontal beams are pinned connected, which means that the joints does not withstand moments. Thus, moments releases on these frames are applied. The setting of the moments releases is presented in figure 6. Frames with released moments are indicated in figure 7.

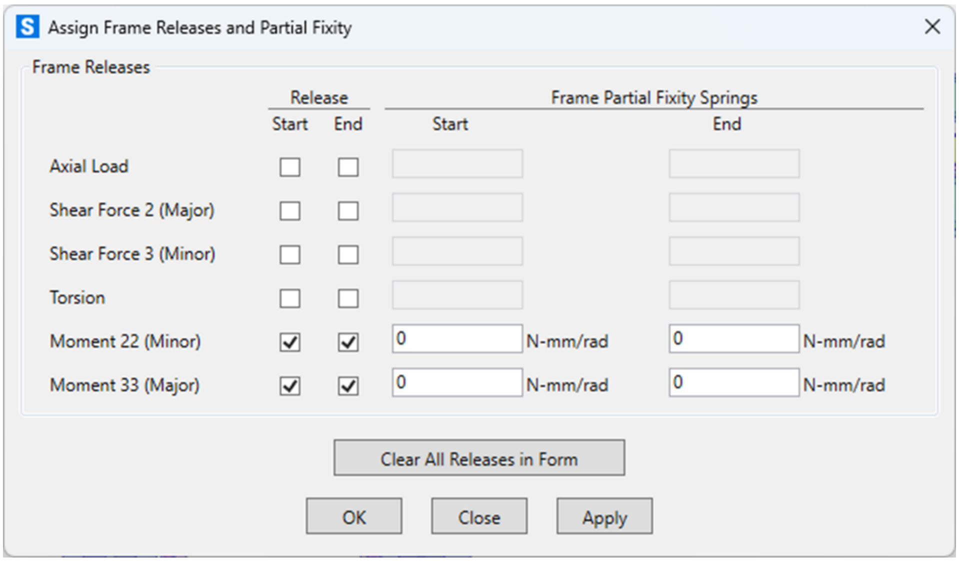


Figure 6: Setting of moments releases for braces and horizontal beams

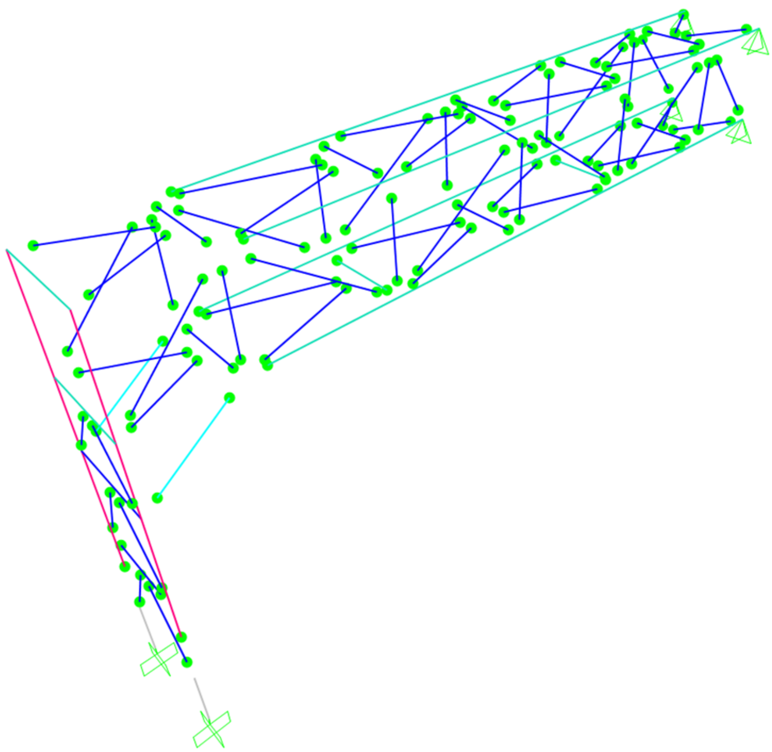


Figure 7: Frames with released moments

3.2 Loads Input and Settings

3.2.1 Dead Load

By default, SAP2000 computes the self weight of the structural elements, which is considered as being a dead load. Another dead loads considered in this design are the weight of the pipes, the weight of the fluids inside the pipes and the friction load of the pipe along the pipe longitudinal axis. These additional dead loads will be modeled as point loads on the horizontal beams, the places where the pipe is anchored. A comprehensive calculation on the piping loads is presented in the attachment. Here we summarize the result of the calculation which will be input in the model.

Table 4: Summary of Piping Loads

Grid to be Applied	023-E941-CR4-0009-600-BACD1			023-E941-PW-0009-50-BACD1		
	Empty Pipe (N)	Fluid (N)	Expansion (N)	Empty Pipe (N)	Fluid (N)	Expansion (N)
A	3752.33	7357.50	999.88	49.05	19.62	6.18
B	0.00	0.00	0.00	147.15	58.86	18.54
C	9005.58	17658.00	2399.72	147.15	58.86	18.54
D	0.00	0.00	0.00	147.15	58.86	18.54
E	9005.58	17658.00	2399.72	147.15	58.86	18.54
F	0.00	0.00	0.00	147.15	58.86	18.54
G	3752.33	7357.50	999.88	49.05	19.62	6.18

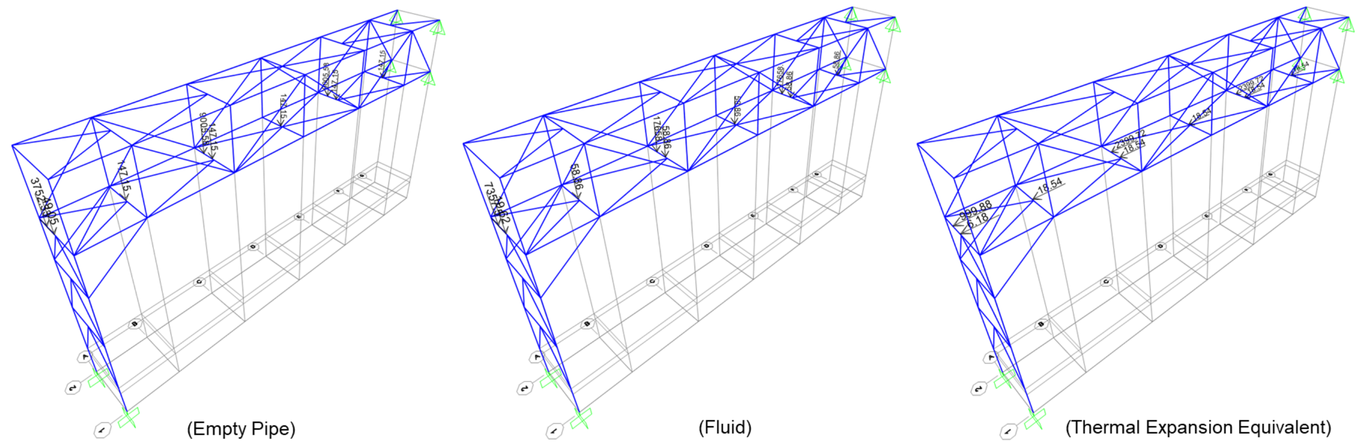


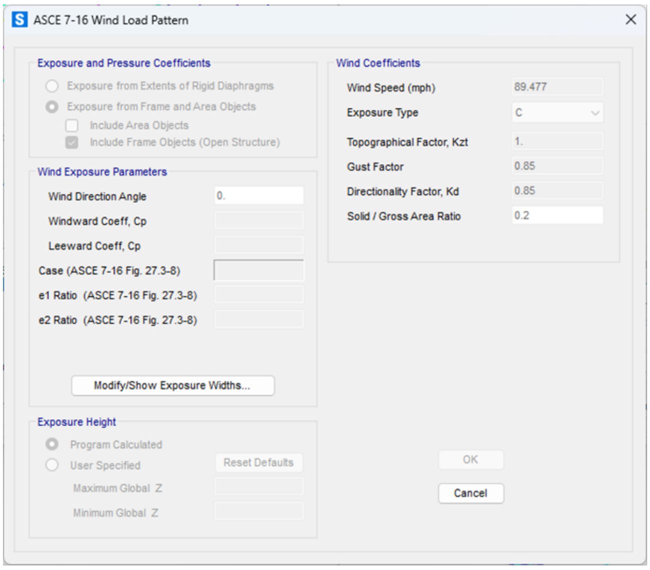
Figure 8: Input of additional dead loads with the unit in Newton

3.2.2 Wind Loads

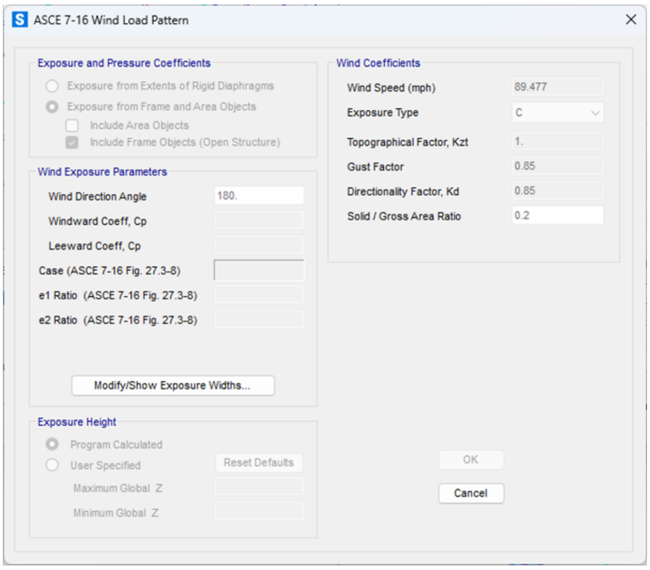
The wind loads will be automatically computed using the dedicated algorithm within SAP2000. SAP2000 provides a computational environment for wind loads according to ASCE 7-16, which is equivalent to SNI 1727:2020. Thus, we can use this computational environment. The wind load parameters are given in table 5. The parameters are to be input in the SAP2000 model. Some illustrations of the input is given in figure 9. The generated wind loads scenario in SAP2000 for all considered directions are presented in figure 10.

Table 5: Wind load parameters

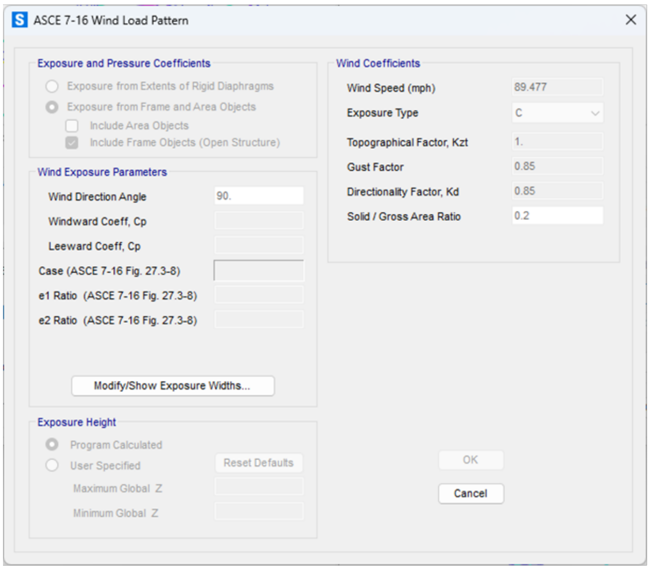
Notation	Item	Value	Unit	Remark
	Exposure classification	C		SNI 1727:2020 sect. 26.7.2
V	Basic wind speed	40.00	m/s	
h	Total height of building	5.60	m	
	Type of building	Open		
f_{nx}	Building natural frequency in x-direction	14.58	Hertz	Computed in SAP2000
f_{ny}	Building natural frequency in x-direction	6.95	Hertz	Computed in SAP2000
	Building rigidity type	Rigid		SNI 1727:2020 sect. 26.2
G	Gust factor	0.85		For rigid building
C_p	Widward coefficient	0.80		
C_p	Leeward coefficient	0.50		
K_{zt}	Topographical factor	1.00		SNI 1727:2020 sect. 26.8.2
K_d	Wind directionality factor	0.85		SNI 1727:2020 tab. 26.6-1



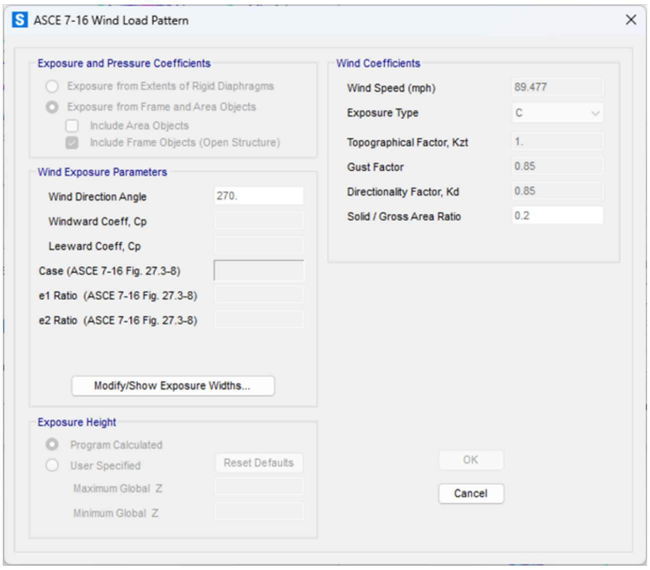
(Input for **Wx[+]**)



(Input for **Wx[-]**)



(Input for **Wy[+]**)



(Input for **Wy[-]**)

Figure 9: Wind parameters input in SAP2000 model

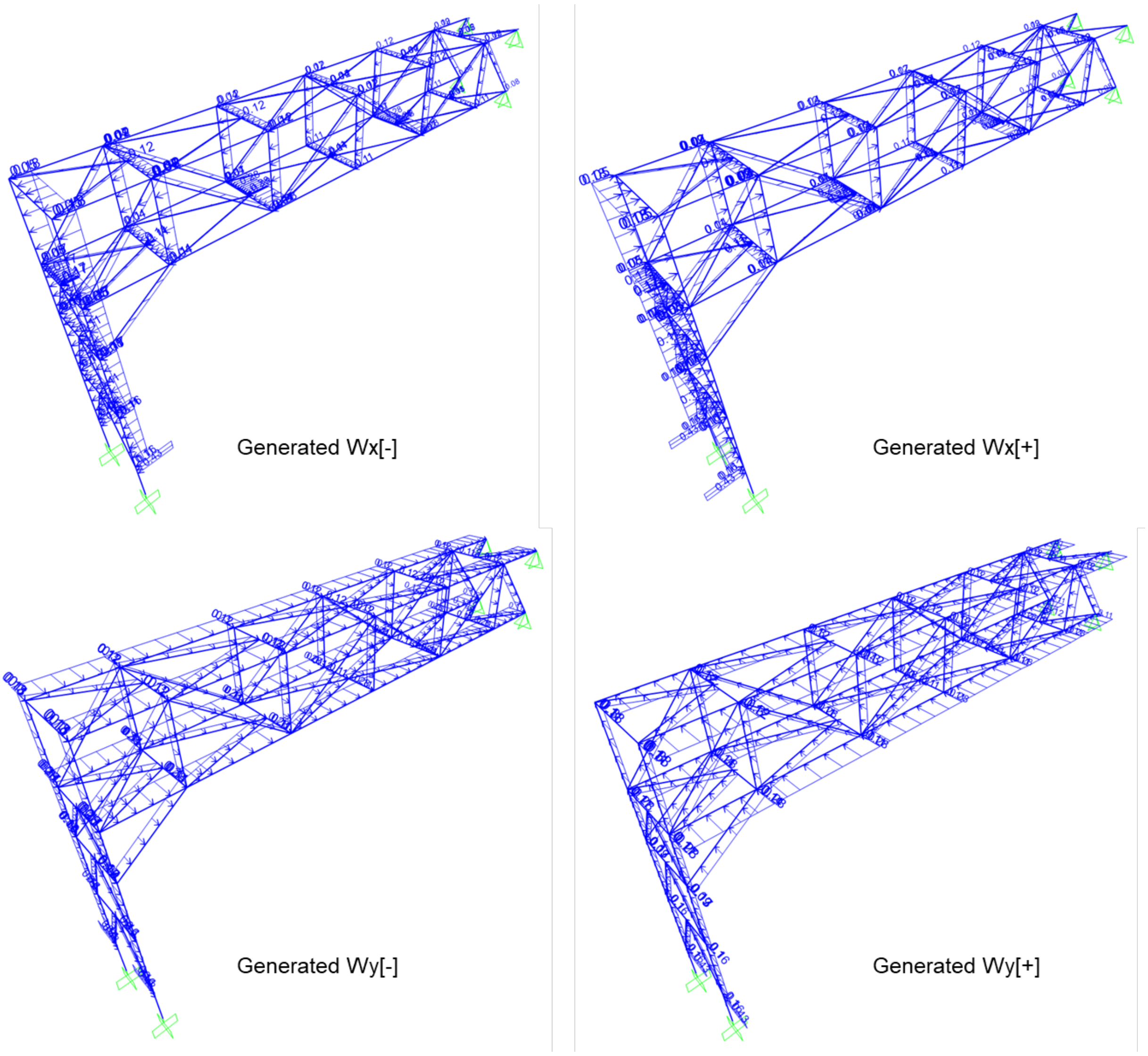


Figure 10: Generated wind loads in SAP2000 (unit in N/mm)

3.2.3 Seismic Load

For the seismic load, we use response spectrum analysis, which is regulated in SNI 1726:2019. SAP2000 provides a dedicated algorithm for computing structural responses from a given design spectra. The seismic parameters for the response spectrum is obtained from the soil report as well as the Probabilistic Seismic Hazard Analysis (PSHA) of Kolaka. These parameters are divided into two parts, for x-direction and for y-direction, and are given in tables 6 and 6. The response spectrum function is computed in accordance with SNI 1726:2019 section 6.4. A formal framework as well as the computation of response spectrum is demonstrated as follows.

Mathematically, a response spectrum function can be modelled as a map $S_a : \mathcal{T} \to \mathbb{R}$, where $\mathcal{T} \subseteq [0, \infty)$ is the time domain of interest with the unit of seconds during earthquake and $\mathbb{R}$ is the set of all real numbers (Rudin, 1976) representing the numeric value of the spectral acceleration in the unit of g (gravitational acceleration). As presented in tables 6 and 6, several parameters are already known are predetermined from either soil report, PSHA of Kolaka or SNI 1726:2019. Among those predetermined parameters, we have $S_{DS}, S_{D1} \in \mathbb{R}$ and $T_L \in \mathcal{T}$, which would be instrumental in determining the map S_a. And follows from SNI 1726:2019 section 6.4, S_a is defined by

$$\forall T \in \mathcal{T} : \; S_a(T) := \begin{cases} S_{DS}\left(0.4 + 0.6\frac{T}{T_0}\right) & : T < T_0 \\ S_{DS} & T_0 \leq T \leq T_s \\ \frac{S_{D1}}{t} & : T_s < T \leq T_L \\ \frac{S_{D1} T_L}{t^2} & : T > T_L \end{cases}$$

where

$$T_0 = 0.2 T_s \quad T_s = \frac{S_{D1}}{S_{DS}} .$$

The graph representation of $S_a : \mathcal{T} \to \mathbb{R}$ with $\mathcal{T} = [0, 10]$ is given by figure 11.

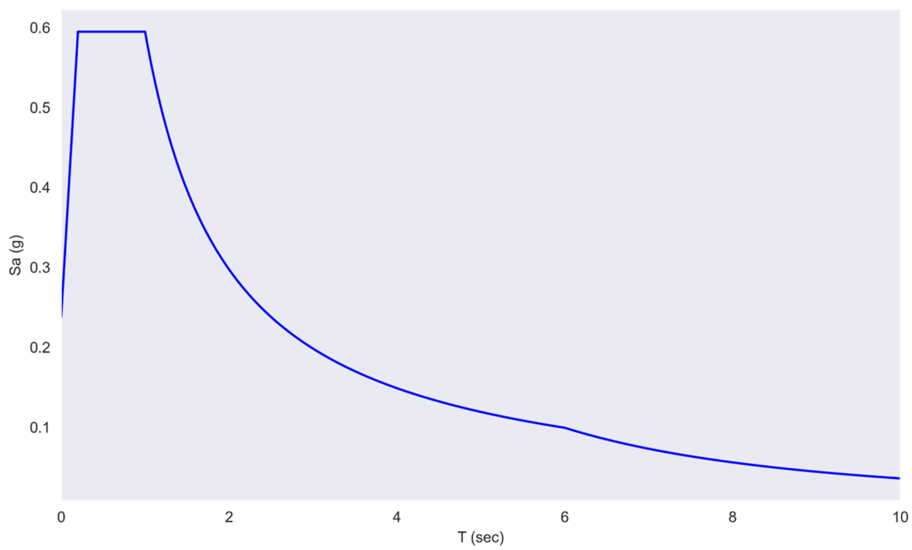


Figure 11: Response spectrum of Kolaka

The generated response spectrum is then implemented in the SAP2000 model, as illustrated in figure 12. The load cases for the seismic loads is then set. We take into consideration of two main directional loads, namely along the X and Y directions. For each direction, a 30% seismic load from the other direction is considered. In addition, the response scale factor gI_e/R is also applied in the setting, since our response spectrum is in the unit of g, while our SAP2000 model uses the unit of mm for length and seconds for time. Note that $g =$ 9810 mm/s^2. The setting in the seismic load cases is given in figure 13.

Table 6: Seismic parameters

Notation	Item	Value	Unit	Remark
	Structural Type	OCBF		
	Risk Category	III		SNI 1726:2019 table 3
I_e	Importance Factor	1.250		SNI 1726:2019 table 4
Class	Site class	SE		SNI 1726:2019 table 5
S_s	Spectral acceleration at t = 0.2 s	0.578	g	PSHA Kolaka
S_1	Spectral acceleration at t = 1.0 s	0.179	g	PSHA Kolaka
F_a	Site coefficient for short period of 0.2 s	1.543		PSHA Kolaka
F_v	Site coefficient for long period	4.981		PSHA Kolaka
S_{DS}	Spectral acceleration parameter of short period with 5% damping	0.595	g	SNI 1726:2019 sect. 6.3
S_{D1}	Spectral acceleration parameter at 1 s	0.594	g	SNI 1726:2019 sect. 6.3
T_L	Long period transtition	6.000	s	SNI 1726:2019 figure 20
	Determining Structural Fundamental Period with Approximation			
C_t		0.073		SNI 1726:2019 table 18
x		0.750		SNI 1726:2019 table 18
h_n	Structural height from the ground	5.600	m	
T	Structural fundamental period	0.287	sec	SNI 1726:2019 sect. 7.8.2.1
R	Response modification coefficient (in y-direction)	3.250		SNI 1726:2019 table 28
Ω_0	Overstrength factor (in y-direction)	2.000		SNI 1726:2019 table 28
C_d	Deflection factor (y)	3.250		SNI 1726:2019 table 28
$C_{s,\min}$	Minimum seismic response coefficient	0.033	g	SNI 1726:2019 sect. 7.8.1.1 eq. (34)
$C_{s,\max}$	Maximum seismic response coefficient	3.601	g	SNI 1726:2019 sect. 7.8.1.1 eq. (32, 33)
C_s	Seismic response coefficient	0.229	g	SNI 1726:2019 sect. 7.8.1.1 eq. (31)
C_s	Seismic response coefficient in use	0.229	g	
ρ	Redundancy factor	1.000		SNI 1726:2019 sect. 7.3.4.2
gI_e/R	Response scale factor for model in SAP2000 (in y-direction)	3773.077		g is in mm/sec for the model

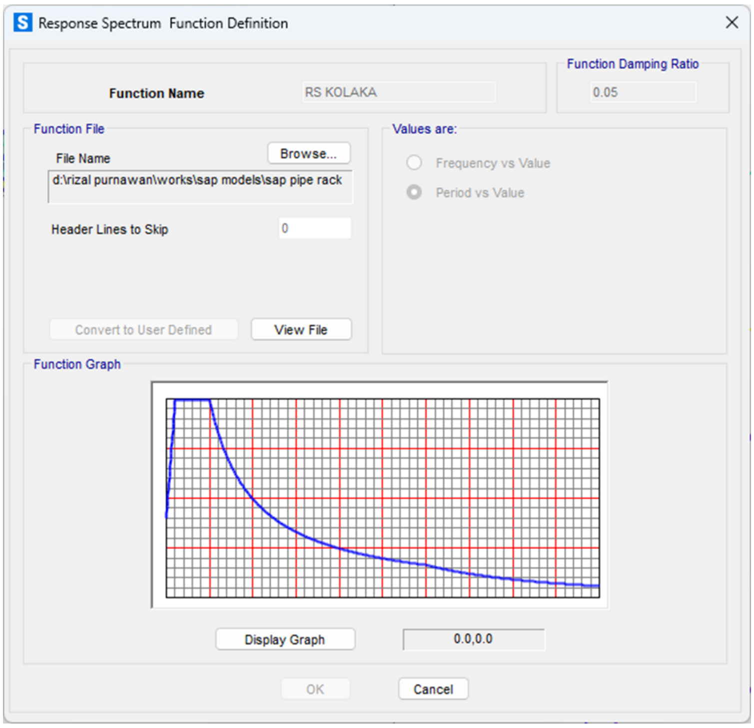


Figure 12: Response spectrum input in SAP2000

3.2.4 Modal

Modal load case shall be set in order to execute the modal analysis on the structure. A sufficient mode shapes shall be determined such that the participating mass ratio of the modal analysis has a considerable value as close as possible to 1. The number of mode shapes is actually needs to be set as many as the number of degree of freedom in the system.

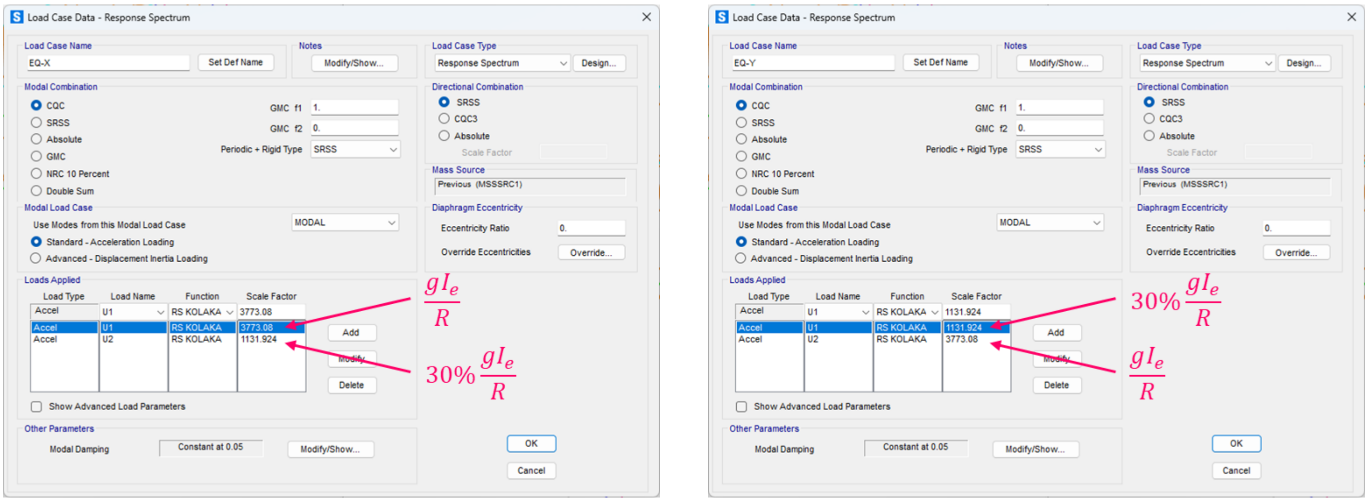


Figure 13: Seismic paramters input in SAP2000 model

However, for a very complex structure such as ours, it will not be convenient to calculate the number of degree of freedom. In exchange, we guess a considerably large number of mode shapes of about 180. The setting is shown in figure 14. On the later section, we will describe the underlying theory describing modal analysis. In addition, the mass source shall also be set. For a simple explanation, according to Newton's second law, force is given by the multiplication of mass and acceleration. Mass source specifies the mass which will be included into force with the acceleration provided from a response spectrum, time history, etc. For this building, since there are no live loads on the structure, then we specify the mass source to be of dead loads which include the structural self weight, the pipe weight as well as the fluid weight.

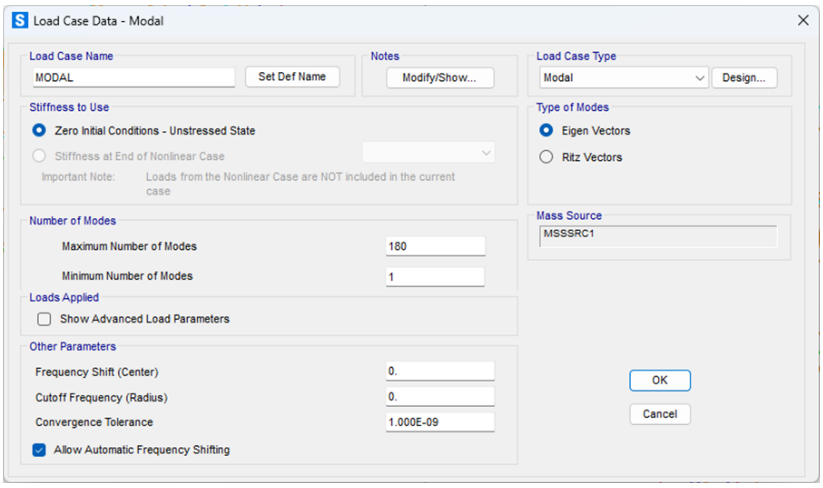


Figure 14: Modal input as well as mass source setting in SAP2000 model

3.3 Foundation Model

We model the foundation separately. There will be two shallow foundations in use for the columns at the left end of the upper structure model (see figure 5). We will use the dedicated modelling template provided in SAP2000 for the foundation called foundation properties. The parameters to be input in the foundation properties include the foundation material properties, the foundation geometric dimensions and the modulus of subgrade reaction of the soil below the foundation slab. The parameter input is presented in figure 15.

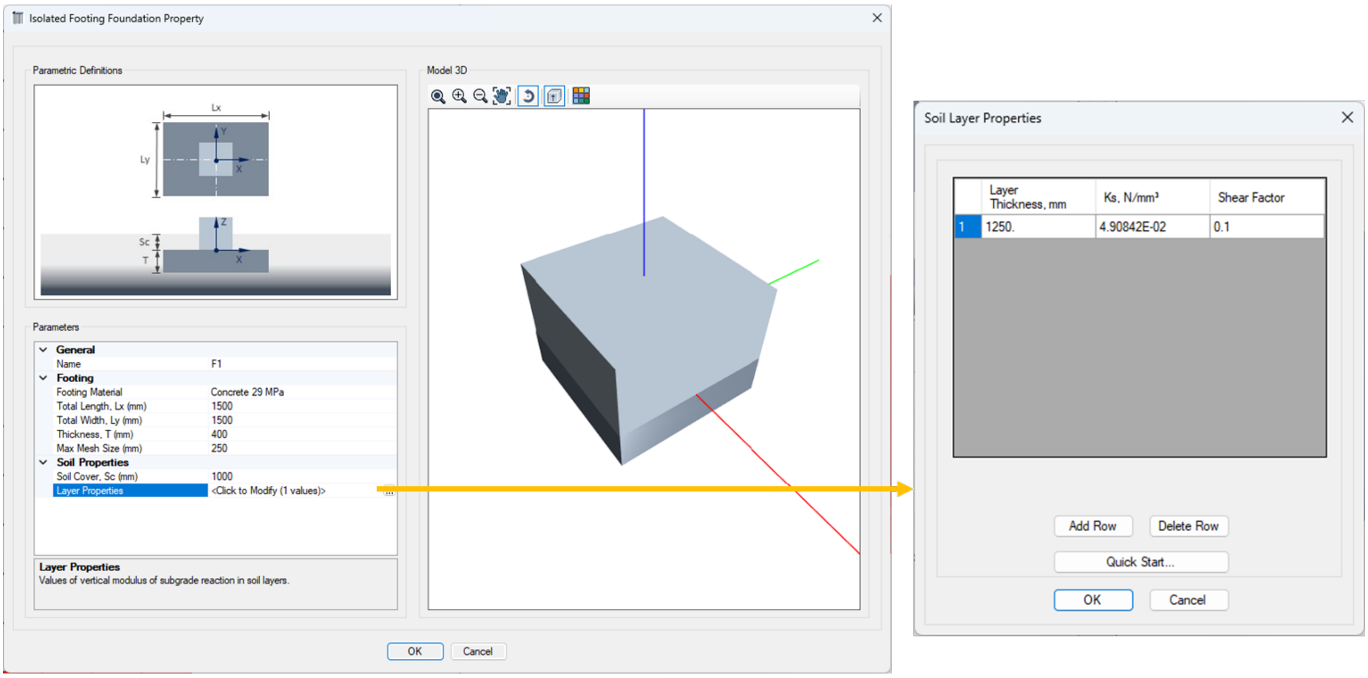


Figure 15: Foundation properties in SAP2000

The modulus of subgrade reaction is computed in accordance with (Das, 2011) as given by

$$k_s = \frac{E_s}{B(1-\nu_s^2)}$$

where B is the width of the foundation, ν_s is the Poisson's ratio of soil and E_s is the Young's modulus of soil. The data of the soil properties is obtained from the soil report, which is given in the attachment. There is only 1 soil layer above the peridotite in this area. The summary of the soil data is presented in table 7.

Table 7: Summary of the soil properties

Borelog	Depth (m)	Unit Weight, γ (kN/m^3)	Young's Modulus, E_s (MPa)	Poisson's Ratio ν_s	Footing Width, B (mm)	Subgrade Modulus, k_s N/mm/mm^2
BH.5.1.38	0-1.250	18.00	67.00	0.30	1500.00	0.04908

The model of the foundation in SAP2000 is shown in figure 16. We also input all the critical support reactions from the upper structure to the model.

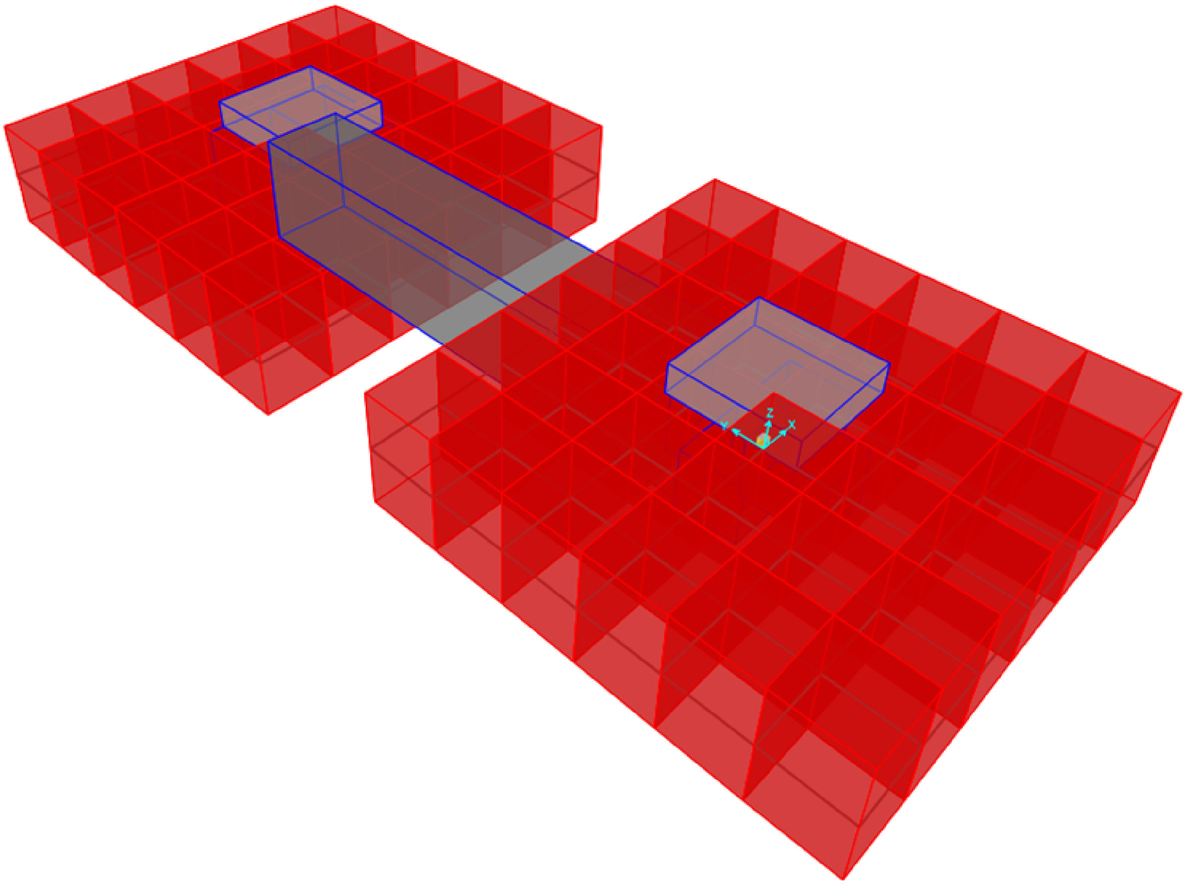


Figure 16: Foundation model in SAP2000

4 GENERAL ANALYSIS

This section covers the modal analysis, seismic analysis and deflections of the whole structure of pipe rack Cc1-Cc3.

4.1 General Theory

4.1.1 Theoretical Basis of Modal Analysis

Modal analysis is the typical common dynamic simulation for many other dynamic simulations, which is mainly used to determine the vibration characteristic of linear elastic structures such as natural frequency or equivalently fundamental period.

The origin of modal analysis can be described from the equation of motion of a classical mechanical system. Suppose we are given a system with n degree of freedoms, for some $n \in \mathbb{N}$. Then the corresponding equation of motion of elements in the system is given by the matrix differential equation

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{f}$$

where $\mathbf{M}, \mathbf{C}, \mathbf{K} \in \mathbb{R}^{n\times n}$ are mass, damping and stiffness matrices, and $\mathbf{u}, \mathbf{f} \in \mathbb{R}^n$ are the displacement and force vectors respectively. Note that $\dot{\mathbf{u}} \in \mathbb{R}^n$ is the first time derivative of $\mathbf{u}$ which is the velocity vector, and $\ddot{\mathbf{u}}$ is the second time derivative of $\mathbf{u}$ which is the acceleration vector.

The point of interest in the modal analysis is at the state that the force is zero and the system is undamped, that is, $\mathbf{f} = \mathbf{0}$ and $\mathbf{C}$ is the zero matrix. And we obtain

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{0}\,,$$

which is a second order linear homogeneous matrix differential equation. Now suppose that the system undergoing a harmonic motion. Then the solution of $\mathbf{u}$ takes the form

$$\mathbf{u} = \mathbf{\Phi} \sin\left(\omega t + \theta\right),$$

where $\mathbf{\Phi}$ is the amplitude of the harmonic motion, ω is the angular frequency and θ is the phase angle.. And then the acceleration becomes

$$\ddot{\mathbf{u}} = -\omega^2 \mathbf{\Phi} \sin\left(\omega t + \theta\right).$$

And by substituting the displacement and acceleration equations above into the equation of motion with undamped system and zero force, we obtain

$$\begin{aligned} \mathbf{K}\mathbf{\Phi}\sin\left(\omega t + \theta\right) - \omega^2 \mathbf{M}\mathbf{\Phi}\sin\left(\omega t + \theta\right) &= \mathbf{0} \\ \therefore (\mathbf{K} - \omega^2\mathbf{M})\mathbf{\Phi} &= \mathbf{0}\,, \end{aligned}$$

which is an eigenvalue problem (Roman, 2005), with ω^2 being the eigevalue and $\mathbf{\Phi}$ being the eigenvector. A trivial solution to the problem is that $\mathbf{\Phi}$ is a zero vector. However, this will not give us more insights and may not be convenient. Other solutions for nonzero $\mathbf{\Phi}$ can be given if

$$\det\left(\mathbf{K} - \omega^2\mathbf{M}\right) = 0\,,$$

where $\det$ denotes the determinant of a square matrix. The number of solutions to the eigenvalue problem is equal to the number of degree of freedoms in the system. Each solution is commonly known as "mode". The eigenvector $\mathbf{\Phi}$ is known as the mode shape. While the square root of the eigenvalue, ω, is the angular natural frequency. The natural frequency in the unit of Hertz (or equally cycle/second) is given by

$$f := \frac{\omega}{2\pi}\,.$$

And the fundamental period is given by

$$T = \frac{1}{f} = \frac{2\pi}{\omega}\,.$$

For a very complex structure, we may have a very large number of degree of freedoms. And computing all the solutions is not convenient and may not be possible. Instead, we need to consider only modes which contribute to the significant deformations of the structure. It can be identified by the modal participating mass ratio. A relatively large number of modal participating mass ratio indicates a large excitation on the structure. The sum of modal participating mass ratio gives an indication that we have included significant modes. As the sum of modal participating mass ratio close to 1, then it means that the significant modes has already been included.

4.1.2 Theoretical Basis of Seismic Analysis

Some verification are needed when using the response spectrum method for computing seismic loads. The verification includes the base shear control and the storey drift limit.

Follows from SNI 1726:2019 section 7.9.1.4.1, if the base shear from the model with response spectrum is less than the base shear computed with the method of static equivalent, then the base shear from the model with response spectrum shall be scaled such that it exceeds the value of the static equivalent. Otherwise, the model is already acceptable. Supposing that we perform the analysis in an iteratuve manner. Suppose V_t is the base shear from our model with response spectrum at the current iteration, or iteration t. And suppose V is the base shear computed with static equivalent method, which is given in accordance with SNI 1726:2019 section 7.8.1 by

$$V = C_s W$$

where C_s is the seismic response coefficient already given in tables 6 and 6, and W is the total weight of the structure. Then

$$V_t < V \implies V_{t+1} := MV_t := \frac{V}{V_t} V_t \,.$$

shall be satisfied. Or alternatively, we can upscale the response spectrum with $M := V/V_t$.

For the storey drift limit, the drift between each adjacent storeys shall be less than the required limit stated in SNI 1726:2019 section 7.12.1. Suppose Δ_x is the story drift at a level x, then

$$\Delta_x < 0.020 h_x$$

according to 1726:2019 section 7.12.1, where h_x is the building height up to level x. And the lateral displacement at level x shall be given by

$$\delta_x = \frac{C_d \delta_{xe}}{I_e} \,,$$

where C_d is the amplification factor which is given in table 6, δ_{xe} is the elastic displacement due to seismic loads at level x computed in the model and I_e is the importance factor which is also given in table 6. Thus,

$$\Delta_x = \delta_x - \delta_{x-1} \,,$$

where δ_{x-1} is the lateral displacement at one level below level x.

4.2 Modal Analysis

In modal analysis, we need to be sure that the model attains a sum of the modal mass participating ratio close to 1 (or 100%). The summary of the sum of the modal mass participating ratio is presented in table 8. The result shows that the model has reached a 100% sum of the modal mass participating ratio, which means that the significant mode shapes have been taken into consideration by the model. Also note that it turns out that there are only 147 modes in the output of the analysis, less than our anticipated modes of 180. It implies that the degree of freedom of the structure is 147. The significant mode shapes in terms of UX and UY is shown in figure 17.

Table 8: Summary of the sum of modal mass participating ratio

Mode	Period (sec)	Frequency (Hertz)	SumUX (Unitless)	SumUY (Unitless)	SumUZ (Unitless)	SumRX (Unitless)	SumRY (Unitless)	SumRZ (Unitless)
1	0.1439	6.9499	0.000	0.760	0.006	0.038	0.000	0.000
2	0.1095	9.1354	0.002	0.800	0.006	0.210	0.000	0.180
3	0.1013	9.8687	0.004	0.810	0.680	0.220	0.066	0.180
4	0.0760	13.1643	0.035	0.830	0.680	0.260	0.067	0.660
5	0.0686	14.5798	0.520	0.840	0.680	0.260	0.074	0.680
⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮
151	0.0004	2277.9043	1.000	1.000	1.000	1.000	1.000	1.000
152	0.0004	2604.1667	1.000	1.000	1.000	1.000	1.000	1.000
153	0.0004	2604.1667	1.000	1.000	1.000	1.000	1.000	1.000
154	0.0003	3484.3206	1.000	1.000	1.000	1.000	1.000	1.000
155	0.0003	3484.3206	1.000	1.000	1.000	1.000	1.000	1.000
156	0.0002	4807.6923	1.000	1.000	1.000	1.000	1.000	1.000

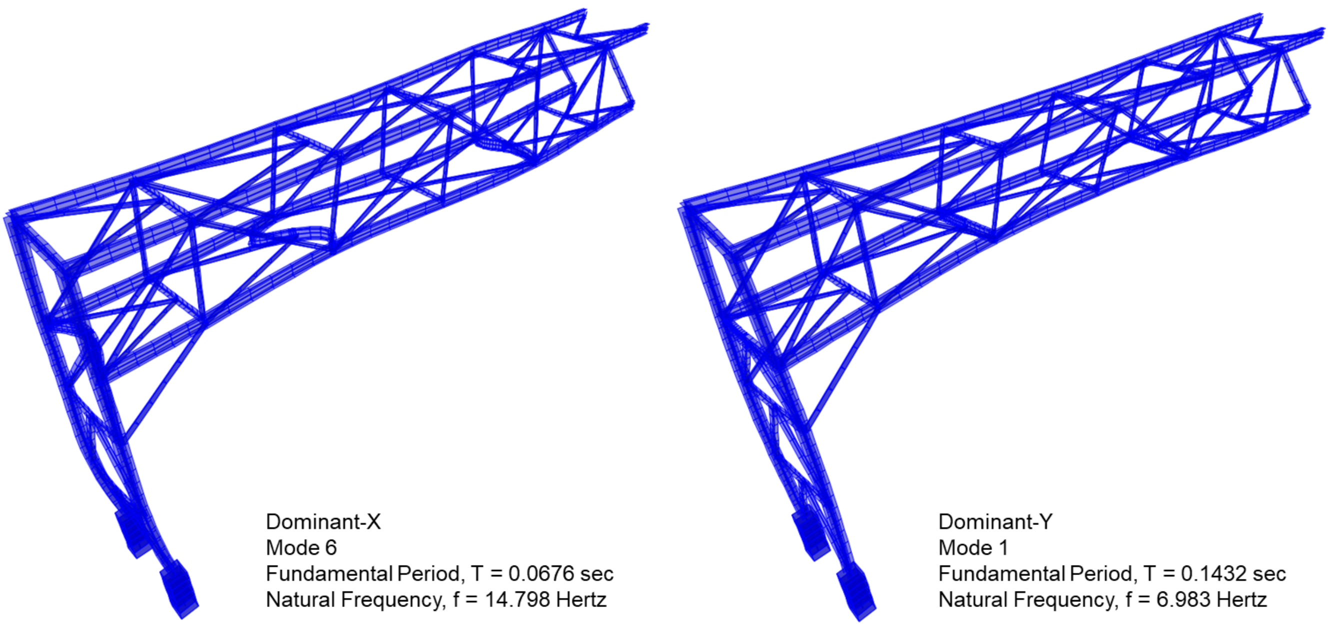


Figure 17: Significant mode shapes for X and Y directions

4.3 Seismic Analysis

First, we will check the base shear from the response spectrum analysis and compare it with the base shear manually computed using the static equivalent method. The seismic base shear can be exported from the SAP2000 model, and is given in table 9. And for computing the base shear using the static equivalent method, we need the weight of the structure as well as additional loads included in the seismic design. We can obtain this from the base reaction of the SAP2000 model accounting for the dead load and pipe operating load, which are loads included in our seismic design. The base reaction of these loads is presented in table 10.

Table 9: Seismic base reaction from SAP2000

Load Case	FX (kN)	FY (kN)
EQ-X	9.997	5.284
EQ-Y	3.040	17.540

Table 10: Gravitational base reaction from SAP2000

Load Case	FZ (kN)
TOTAL	127.520

The static equivalent base shears in both x and y directions are given as follows:

$$V_{sx} = C_s W = 0.229 \cdot 127.520 = 29.202 \text{ kN}$$
$$V_{sy} = C_s W = 0.229 \cdot 127.520 = 29.202 \text{ kN}$$

Then we obtain

$$V_{t,x} = 9.997 \text{ kN} < 29.202 \text{ kN} = V_{sx}$$
$$V_{t,y} = 17.540 \text{ kN} < 29.202 \text{ kN} = V_{sy}$$

which implies that the response spectrum in the current iteration of the model must be upscaled. And the scaling factors are given by

$$M_x := \frac{V_{sx}}{V_{t,x}} = \frac{29.202}{9.997} \approx 2.93$$
$$M_y := \frac{V_{sy}}{V_{t,y}} = \frac{29.202}{17.540} \approx 1.67$$

for x and y directions respectively. Thus, the seismic response scales are revised as follows:

$$\frac{M_x g I_e}{R_x} = 2.93 \cdot 3773.077 = 11055.116$$
$$\frac{M_y g I_e}{R_y} = 1.67 \cdot 3773.077 = 6301.039$$

These new scales are then input to the model, and the model is reanalyzed. The input is shown in figure 18.

Now we check the storey drift of the structure due to the seismic loads. The storey drifts obtained from the model are presented in figure 19. From the model, we obtain as follows:

$$\delta_{0e,x} = 0.0135 \text{ mm}$$
$$\delta_{1e,x} = 0.1418 \text{ mm}$$
$$\delta_{0e,y} = 0.0944 \text{ mm}$$
$$\delta_{1e,y} = 2.1112 \text{ mm}$$

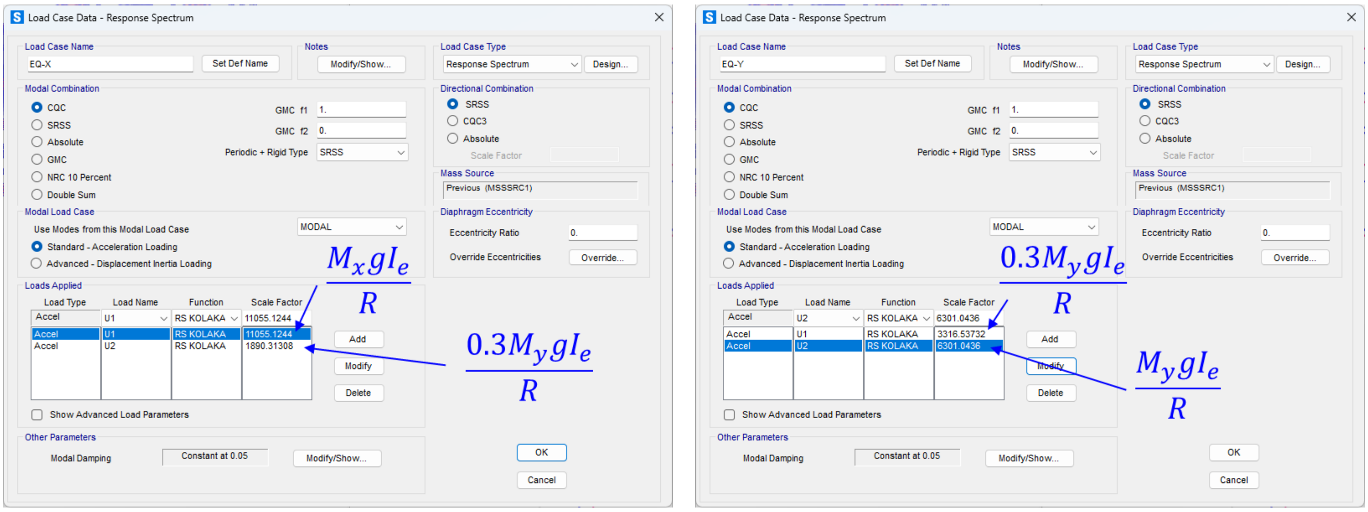


Figure 18: Rescaled response spectrum

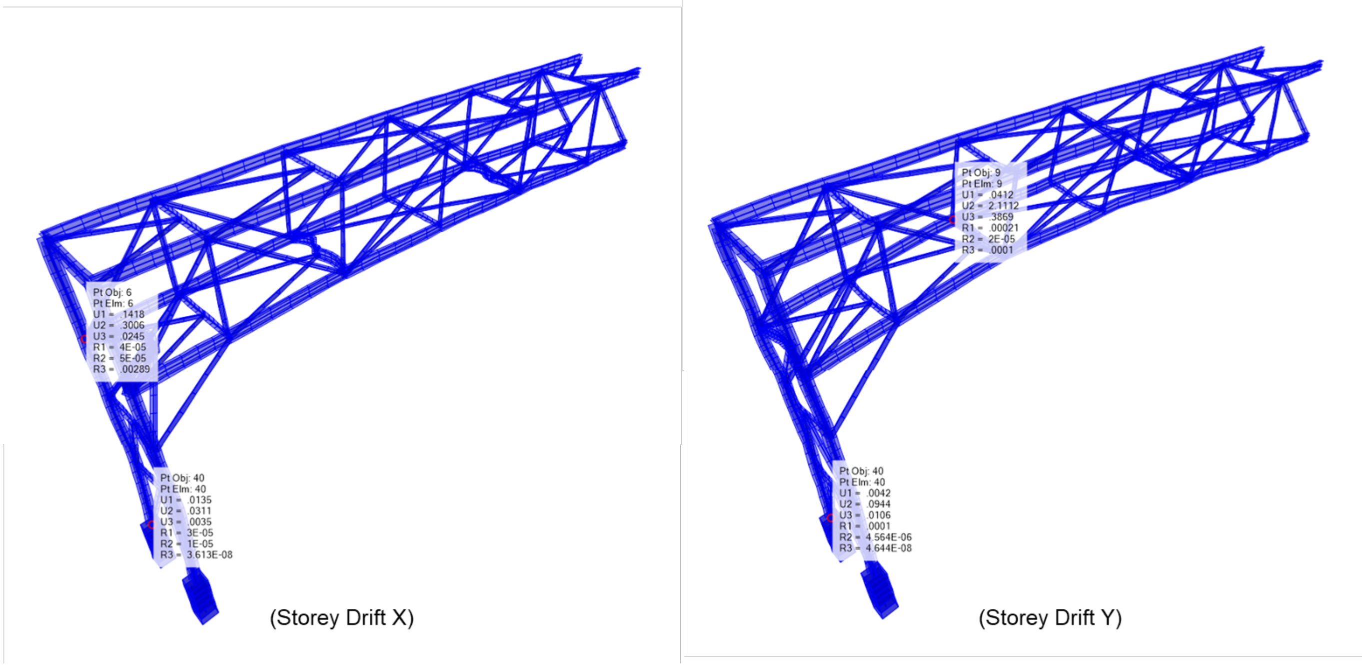


Figure 19: Storey drifts in x (left) and y (right) directions

Then we obtain as follows:

$$\delta_{0,x} = \frac{C_{d,x}\delta_{0e,x}}{I_e} = \frac{3.25 \cdot 0.0135}{1.25} = 0.0351 \text{ mm}$$

$$\delta_{1,x} = \frac{C_{d,x}\delta_{1e,x}}{I_e} = \frac{3.25 \cdot 0.1418}{1.25} = 0.36868 \text{ mm}$$

$$\delta_{0,y} = \frac{C_{d,y}\delta_{0e,y}}{I_e} = \frac{3.25 \cdot 0.0944}{1.25} = 0.24544 \text{ mm}$$

$$\delta_{0,y} = \frac{C_{d,y}\delta_{0e,y}}{I_e} = \frac{3.25 \cdot 2.1112}{1.25} = 5.48912 \text{ mm}$$

And the storey drifts for both x and y directions are given by

$$\Delta_x = \delta_{1,x} - \delta_{0,x} = 0.36868 - 0.0351 = 0.334 \text{ mm},$$
$$\Delta_y = \delta_{1,y} - \delta_{0,y} = 5.48912 - 0.24544 = 5.244 \text{ mm}.$$

Note that the storey height considered in this occasion is the height of the structure in the model, which is $h = 5500$ mm. Then the allowable drift is given by

$$\Delta_a = 0.025h = 0.025 \cdot 5500 = 137.500 \text{ mm}.$$

Then

$$\Delta_x = 0.334 < 137.50 = \Delta_a$$
$$\Delta_y = 5.244 < 137.50 = \Delta_a$$

show that the storey drifts are acceptable.

4.4 Vertical Deflection Control

The deflection of the main longitudinal beam in the structure of pipe rack takes into account all the dead loads, including the self weight of the structure and the pipe operating loads. The deflection from the model is illustrated in figure 20. According to the project design basis, the allowable deflection for pipe rack is given by $L/240$, where L is the span of the pipe rack. Then we obtain

$$\delta = 2.988 \text{ mm} < 66.667 \text{ mm} = \frac{16000}{240} = \frac{L}{240}$$

which shows that the vertical deflection is acceptable.

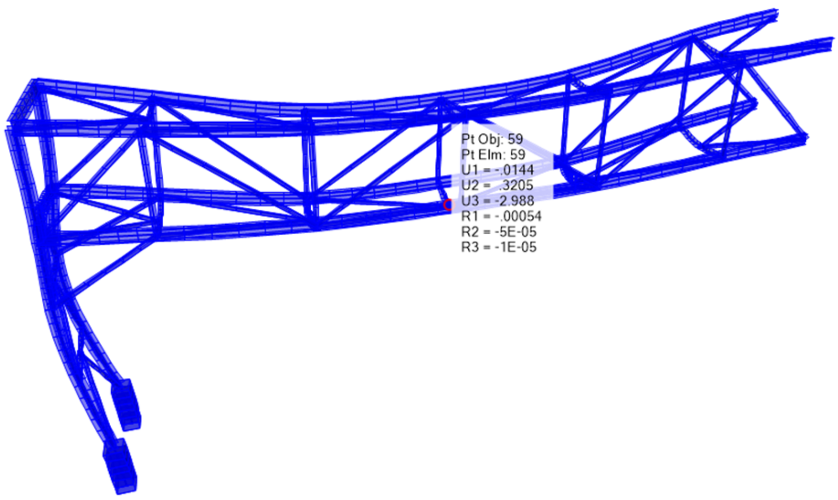


Figure 20: Vertical deflection of pipe rack in SAP2000 model

5 ANALYSIS OF UPPER STRUCTURE

The analysis of the upper structure consists of the steel members capacity evaluation and the concrete structures evaluation for the pedestal and the tie-beam. The analysis uses the dedicated environment in SAP2000 which is based on AISC 360-16 for steel structures and ACI 318-19 for concrete structures which are equivalent to SNI 1727:2020 and SNI 2847:2019 respectively.

5.1 Analysis of Steel Structures

The upper structure use a system of OCBF. The input of the design parameters is illustrated in figure 21.

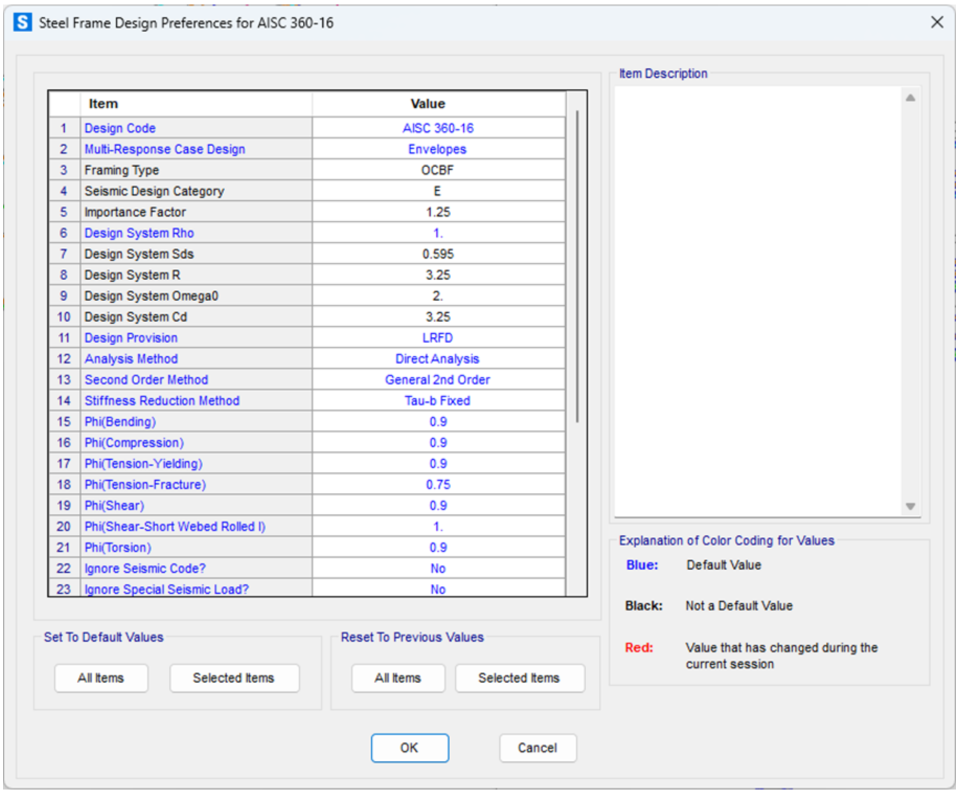


Figure 21: Steel parameters

The result of the analysis is presented in the form of strength ratio, which is presented in figure 22. From the strength ratio result, it can be observed that the maximum strength ratio is less than 0.8, which implies that the design is sufficient.

5.2 Steel Connections

The steel connections in the design will use the designation in the official standard drawings of the project. Hence, no calculations regarding the connections are presented in this document.

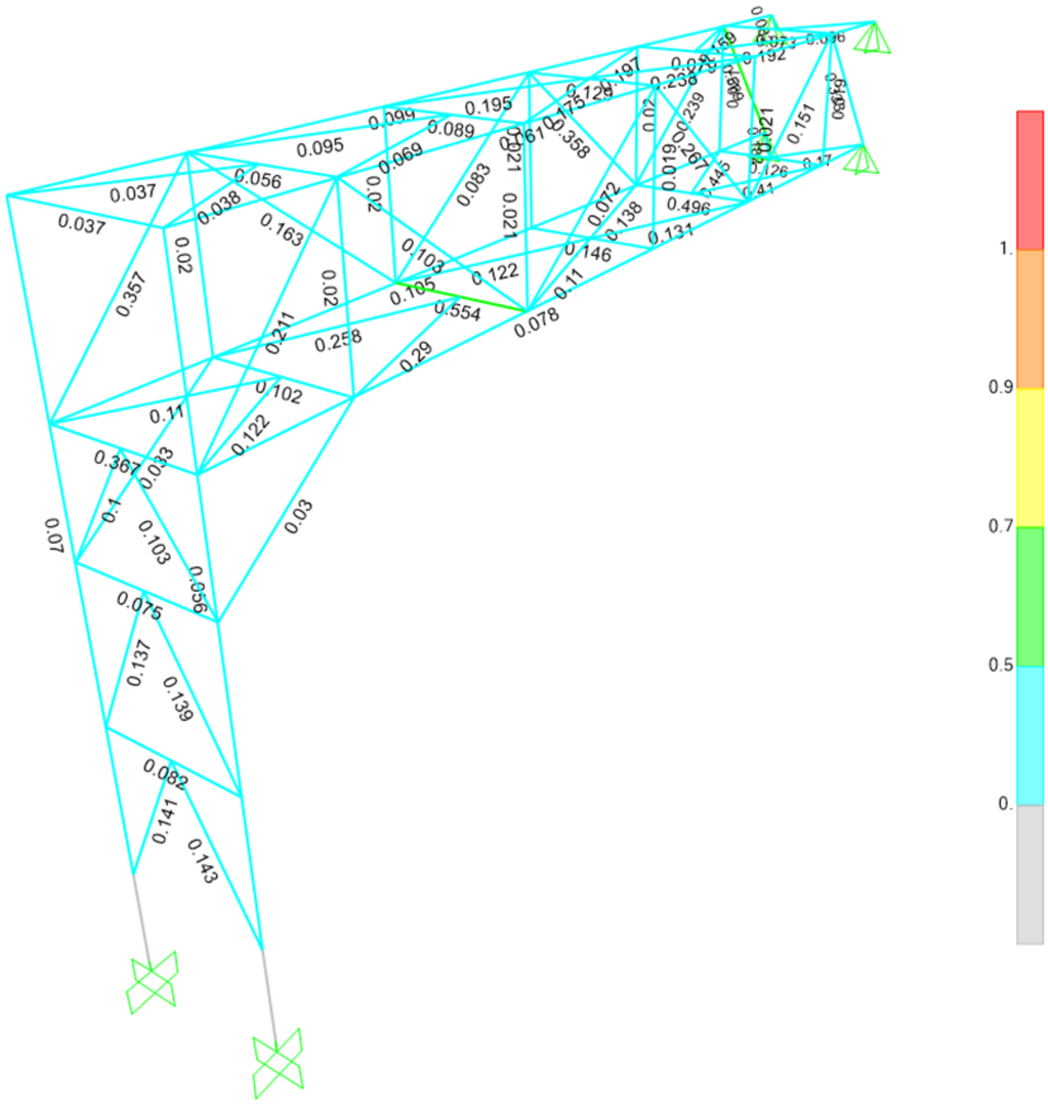


Figure 22: Steel strength ratio

6 ANALYSIS OF FOUNDATION

The analysis of the substructure includes the reinforcement of the pedestal, the tie-beam, the footing foundation, the foundation bearing capacity and the stability of the foundation. The analysis on the stability of the foundation includes the overturning and the sliding stability. The analysis will consider all load combinations in use, as utilized automatically by the software. For the reinforcement analysis, the LRFD load combinations are used. While the foundation stability analysis uses the ASD load combinations.

6.1 Pedestal Reinforcement

We have modelled the pedestals in SAP2000 as columns structure. The configuration in use is as illustrated in figure 23.

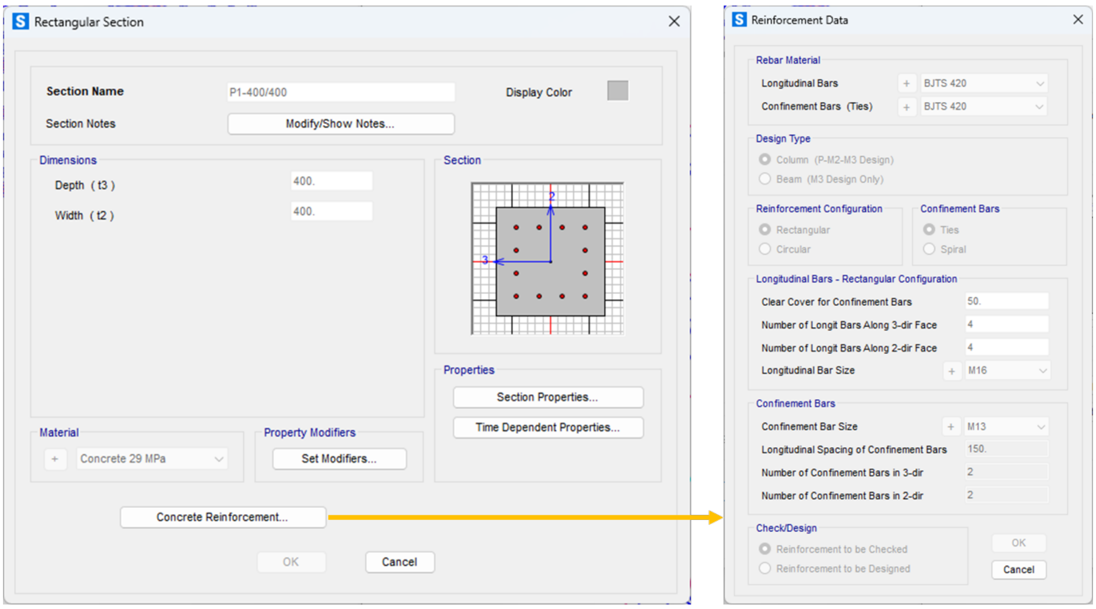


Figure 23: Reinforcement configuration for pedestal P1

The result of the analysis is given in figure 24 which shows that the strength ratio of the pedestal is less than 0.2. If the strength ratio is less than or equal to 1.0, then the capacity is sufficient. Thus, we conclude that the capacity of the pedestal is sufficient since

$$r_c < 0.2 < 1.0\,,$$

where r_c is the strength ratio of the pedestal. One last step, we need to control the reinforcement ratio of the pedestal. SNI 2847:2019 section 16.3.4.1 provides a requirement for the longitudinal rebars as given by

$$A_s \geq 0.005 A_g\,,$$

where A_s is the cross sectional area of the longitudinal rebars. The requirement can equivalently be stated as

$$\frac{A_s}{A_g} \geq 0.005 = 0.5\%\,.$$

From our design, we obtain

$$\frac{A_s}{A_g} = \frac{n_r \cdot A_r}{bh} = \frac{12 \cdot (0.25\pi 16^2)}{400 \cdot 400} = 0.01508 = 1.508\% > 0.5\%$$

which shows that the rebars in use is satisfiable. We summarize the reinforcement configuration of the pedestal in table 11.

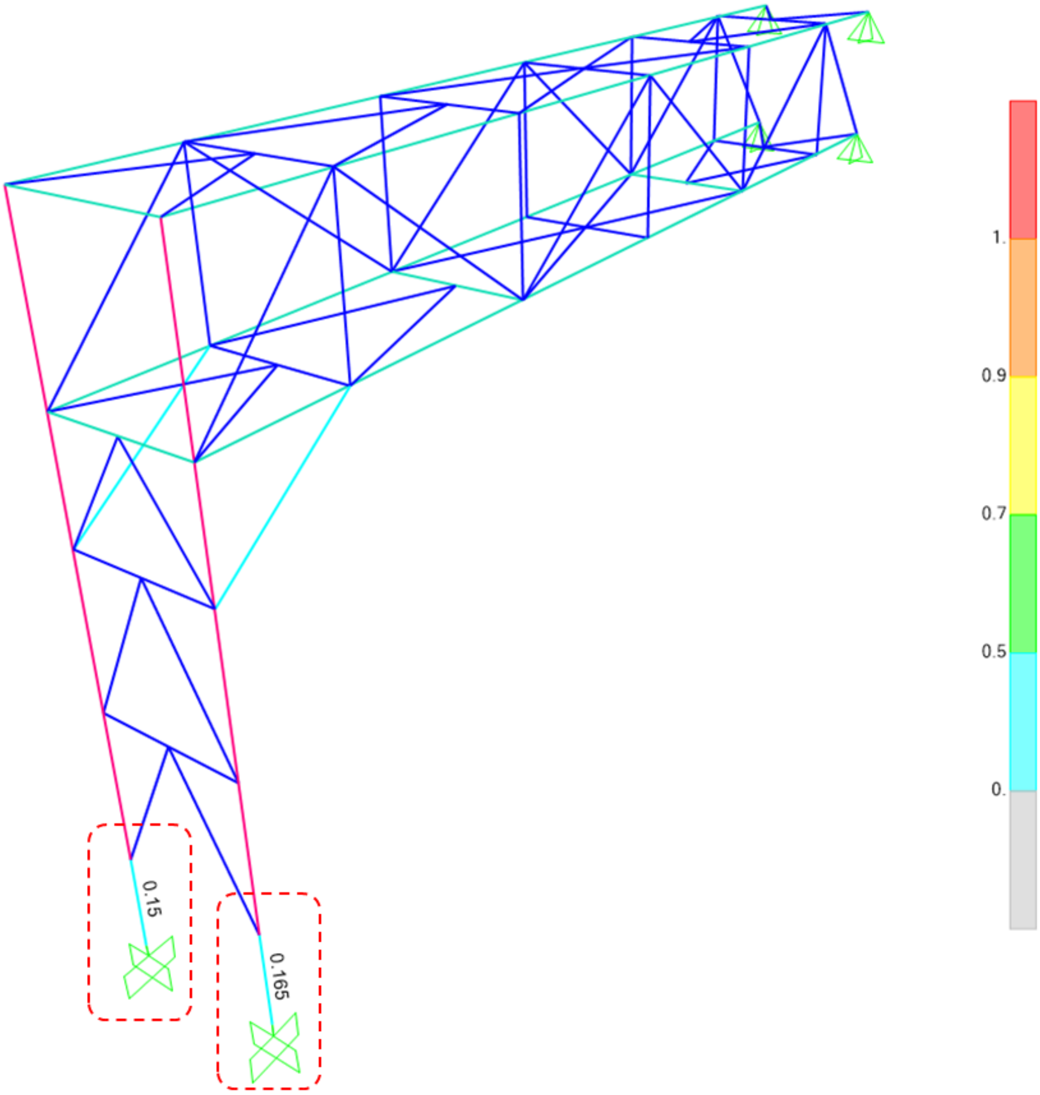


Figure 24: PMM strength ratio of pedestal

Table 11: Pedestal reinforcement summary

Reinforcement Type	Configuration
Longitudinal	12D16
Transversal	2D13-150

6.2 Tie-Beam Reinforcement

We have modelled the time-beam in SAP2000 as a beam structure. The configuration in use is as illustrated in figure 25.

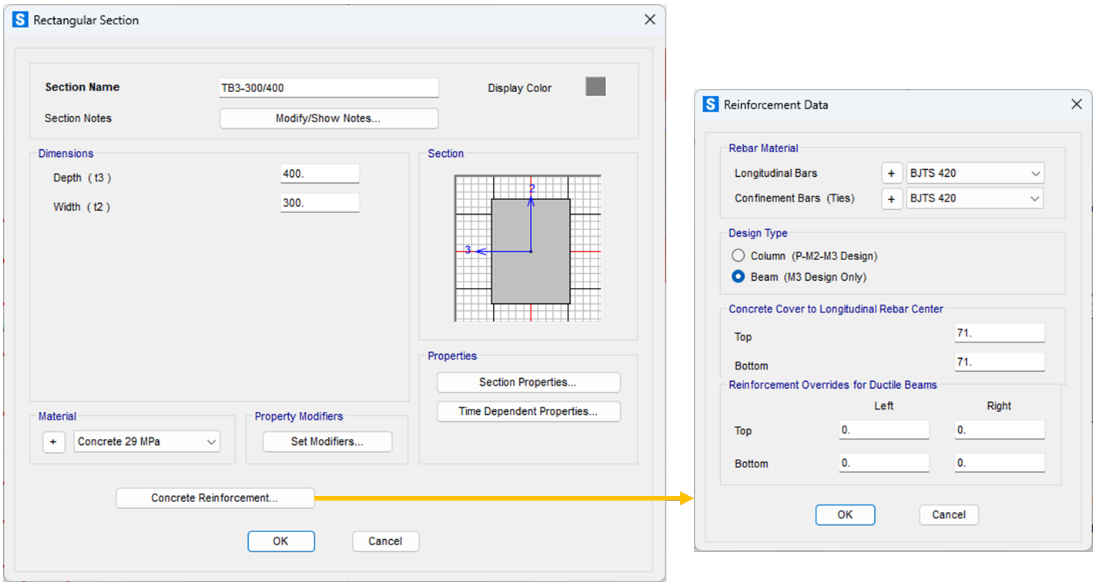


Figure 25: Tie-beam configuration

The result of the analysis is a reinforcement area, i. e., A_s, which is shown in figure 26.

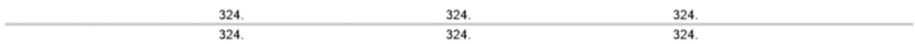


Figure 26: Tie-beam reinforcement result from SAP2000

The detail calculation of the reinforcement configuration on the result provided by the SAP2000 model is given in the attachment. Here we summarize the result in table

Table 12: Reinforcement of tie beam TB1

Type	End Span	Mid Span
Longitudinal	4D16	4D16
Transversal	2D13-200	2D13-200

6.3 Foundation

We conducted a thorough analysis on the stability of the foundation. The detail calculation of the analysis is given in the attachment. Here we present the result from the modelling of the foundation in SAP2000. The distribution of the most critical non-overstrength soil pressure obtained from the model is illustrated in figure 27. And the reinforcement summary of the foundation is given in table 13.

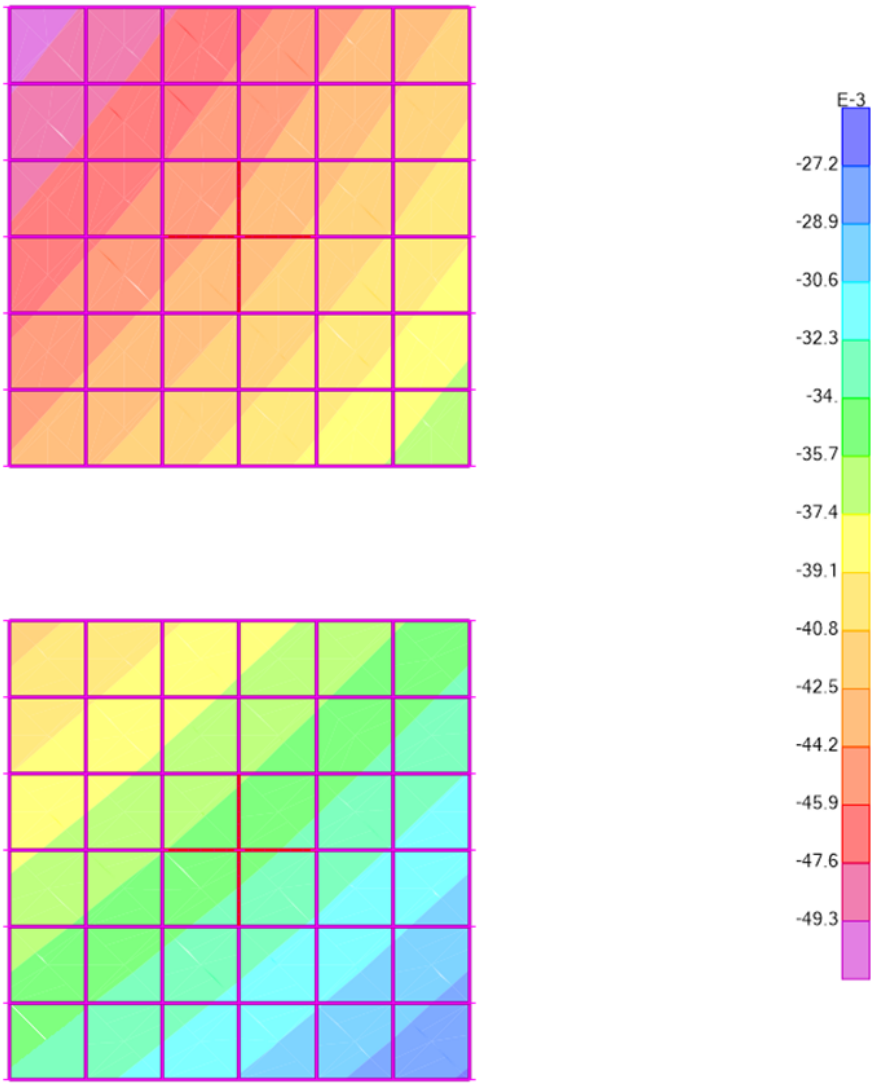


Figure 27: Soil pressure due to load combination 1.0D + 0.7Ev + 0.7Ehx (non-overstrength)

Table 13: Summary of Footing F1 Reinforcement

Direction	Reinforcement
X	D16-200
Y	D16-200

References

Das, B. M. (2011). *Principles of Foundation Engineering*. Cengage Learning.

Roman, S. (2005). *Graduate Text in Mathematics: Advanced Linear Algebra*. Springer.

Rudin, W. (1976). *Principles of Mathematical Analysis, 3rd Edition*. McGraw-Hill.

ATTACHMENT 1

Calculation of Piping Loads

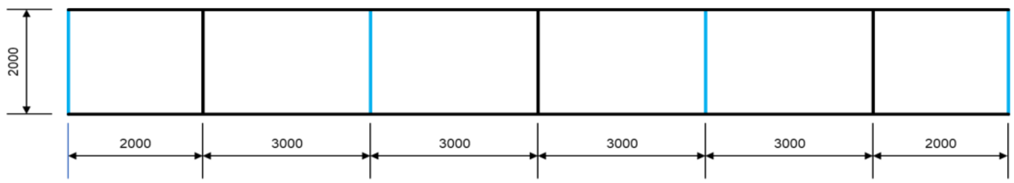
Calculation of Piping Loads for Piperack Cc1~Cc3

A. INTRODUCTION

This document covers the mechanism in calculating the loads of pipe as well as its fluid contents for the input in the model of pipe rack Cc1~Cc3.

B. PIPERACK GEOMETRICAL CONFIGURATION

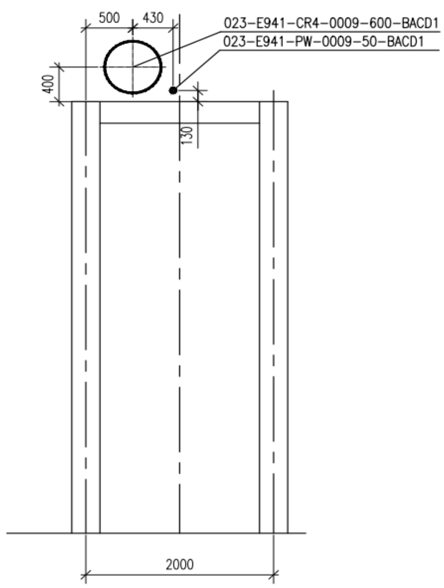
The principle plan view of the piprack is presented in the following diagram.



The line with blue color is the place where the pipe will be rested.

C. PIPING INFORMATION

The piping configuration is determined based on the input from ENFI. The illustration is presented as follows (source: 922A-E941-023-18):



From the data, we can see that there are only 1 pipe in piperack Cc1~Cc3. The detailed information about the piping is presented as follows:

P1. Pipe number: 023-E941-CR4-0009-600-BACD1

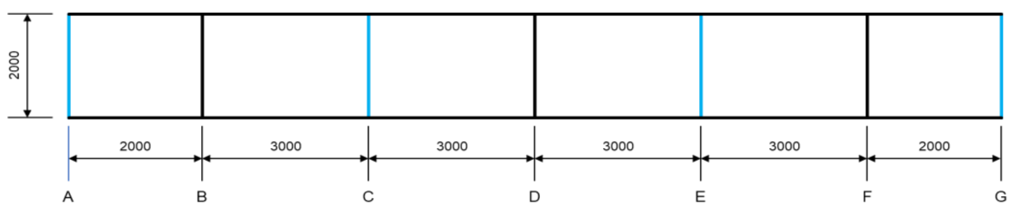
Doc. ref. for the following data: 922A-E941-023

Diameter	DN =	600.00	
Fluid content		Water	
Temperature	T =	65.00	C
Pressure	p =	0.30	MPa

	Empty pipe weight	PE =	153.00	kg/m
	Fluid weight	F_w =	300.00	kg/m
P2.	Pipe number: 023-E941-PW-0009-50-BACD1			
	Doc. ref. for the following data: 922A-E941-023			
	Diameter	DN =	50.00	
	Fluid content		Water	
	Temperature	T =	25.00	C
	Pressure	p =	0.45	MPa
	Empty pipe weight	PE =	5.00	kg/m
	Fluid weight	F_w =	2.00	kg/m

D. CALCULATION OF PIPING LOAD

Let us stick to the following illustration.



As mentioned, there are two pipes in the pipe rack. The pipe with DN = 600 mm will be restrained on grids with blue color, while the pipe with DN = 50 mm will be restrained on all grids. We will refer grids A and G as exterior grids and the others as interior grids. We will develop a systematic algorithmic approach to compute the piping loads. Now suppose the following set:

$$S := \begin{cases} \{A, C, E, G\} & : \text{ for P1} \\ \{A, B, C, D, E, F, G\} & : \text{ for P2} \end{cases}$$

And the map

$$d : S \times S \to \mathbb{R}$$

will describe the distance between a pair of grids. For instance we have

$$d(A, B) = 2000 \text{ mm}.$$

Then suppose a map

$$N : S \to \mathcal{P}(S)$$

such that

$$\forall X \in S : \; N(X) := \left\{ W \in S \,\middle|\, d(W, X) = \inf_{U \in S \setminus \{X\}} d(U, X) \right\}.$$

Let P be the piping load of either PE, F_w or PE + F_w. Then the piping load at a single grid is given by a map

$$Q : S \to \mathbb{R}$$

defined by

$$\forall X \in S : \; Q(X) = P \sum_{U \in N(X)} \frac{d(U, X)}{2}.$$

The calculation is presented as follows:

P1. ***Grids A and G:***

	Empty pipe weight	Q_{PE} =	3,752.33	N
	Fluid weight	Q_{Fw} =	7,357.50	N
	Total weight	Q =	11,109.83	N
	Grids C and E:			
	Empty pipe weight	Q_{PE} =	9,005.58	N
	Fluid weight	Q_{Fw} =	17,658.00	N
	Total weight	Q =	26,663.58	N
P2.	***Grids A and G:***			
	Empty pipe weight	Q_{PE} =	49.05	N
	Fluid weight	Q_{Fw} =	19.62	N
	Total weight	Q =	68.67	N
	Grids B, C, D, E, F:			
	Empty pipe weight	Q_{PE} =	147.15	N
	Fluid weight	Q_{Fw} =	58.86	N
	Total weight	Q =	206.01	N

E. THERMAL EXPANSION-EQUIVALENT LOAD

In different temperatures, the pipe will undergoe thermal expansions which result in the change of its initial length. In other words, the change in temperature deforms the pipe material. Deformations in the pipe will impose some force to the beam at which the pipe is rested. This kind of force can be described using the theory of thermal expansion and the Hooke's Law.

From the theory of thermal expansion, we have

$$\alpha = \frac{1}{L}\frac{\mathrm{d}L}{\mathrm{d}T}$$

where α is the thermal coefficient and L is the length of the material, which, in this case, is assumed to be a function with respect to temperature T, since the length of the material changes with the change of temperature. As commonly used in differential equations, the map

$$L : T \mapsto L(T)$$

is assumed to be differentiable. Supposing the thermal expansion is linear, we may approximate the differential equation above and obtain

$$\frac{\Delta L}{L} = \alpha \Delta T\,.$$

On the other hand, from Hooke's Law, we obtain

$$F = k\Delta L = \frac{EA}{L}\Delta L \quad \models \frac{\Delta L}{L} = \frac{F}{EA}\,.$$

Thus, we obtain

$$F = \alpha \Delta T \cdot EA$$

which is the equivalent force which causes the same deformation to that of the thermal expansion.

All this description only explains the derivation how thermal expansion can be equivalently expressed in a form of force. For the practical purpose of this calculation, we use the method regulated in the design basis (doc. no: DBS-1-000-C-0001), which considers the expansion force as some proportion of the pipeweight multiplied with the friction coefficient, as given by

$$F = c \cdot P \cdot \mu$$

where

$$c = \begin{cases} 10\% & : \text{ there are 4 or more pipes} \\ 30\% & : \text{ there are less than 4 pipes} \end{cases}$$

$$\mu = \begin{cases} 0.30 & : \text{ steel-to-steel} \\ 0.10 & : \text{ steel-to-teflon} \\ 0.08 & : \text{ teflon-to-teflon} \\ 0.40 & : \text{ steel-to-concrete} \end{cases}$$

and P is PE + Fw. And the expansion force will be applied to grids A, C, E, G, similar to the piping loads. Thus, the applied expansion force is given by the map

$$Q_h : S \to \mathbb{R}$$

and is defined by

$$\forall X \in S:\; Q_h(X) = F \sum_{U \in N(X)} \frac{d(U,X)}{2} = c\mu P \sum_{U \in N(X)} \frac{d(U,X)}{2} = c\mu Q(X)\,.$$

The calculation of the expansion force is presented as follows:

	Number of pipes		1.00	
	Proportion factor	c =	0.30	
	Friction coefficient	μ =	0.30	
P1.	***Grids A and G:***			
	Piping load	Q =	11,109.83	N
	Expansion load	Q_h =	999.88	N
	Grids C and E:			
	Piping load	Q =	26,663.58	N
	Expansion load	Q_h =	2,399.72	N
P2.	***Grids A and G:***			
	Piping load	Q =	68.67	N
	Expansion load	Q_h =	6.18	N
	Grids B, C, D, E, F:			
	Piping load	Q =	206.01	N
	Expansion load	Q_h =	18.54	N

ATTACHMENT 2

Configuration of Tie-Beam Reinforcement

TABLE: Concrete Design 2 - Beam Summary Data - ACI 318-19						
Frame	DesignSect	DesignType	Status	Location	FTopCombo	FTopArea
Text	Text	Text	Text	mm	Text	mm2
2	TB3-300/400	Beam	No Messages	200	(PO-LRFD-OS) 0.9D - Ev + Emhy	324.054
2	TB3-300/400	Beam	No Messages	600	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	324.054
2	TB3-300/400	Beam	No Messages	1000	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	324.054
2	TB3-300/400	Beam	No Messages	1400	(PO-LRFD-OS) 0.9D - Ev + Emhy	324.054
2	TB3-300/400	Beam	No Messages	1800	(PO-LRFD-OS) 0.9D - Ev + Emhy	324.054

Frame	DesignSect	DesignType	Status	Location	FBotCombo	FBotArea
Text	Text	Text	Text	mm	Text	mm2
2	TB3-300/400	Beam	No Messages	200	(PO-LRFD-OS) 0.9D - Ev + Emhy	324.054
2	TB3-300/400	Beam	No Messages	600	(PO-LRFD-OS) 0.9D - Ev + Emhy	324.054
2	TB3-300/400	Beam	No Messages	1000	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	324.054
2	TB3-300/400	Beam	No Messages	1400	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	324.054
2	TB3-300/400	Beam	No Messages	1800	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	324.054

Frame	DesignSect	DesignType	Status	Location	VCombo	VRebar
Text	Text	Text	Text	mm	Text	mm2/mm
2	TB3-300/400	Beam	No Messages	200	(PO-LRFD-OS) 1.2D + Ev + Emhy + L (Sp)	0.696
2	TB3-300/400	Beam	No Messages	600	(PO-LRFD-OS) 1.2D + Ev + Emhy + L (Sp)	0.682
2	TB3-300/400	Beam	No Messages	1000	(PO-LRFD-OS) 1.2D + Ev + Emhy + L (Sp)	0.667
2	TB3-300/400	Beam	No Messages	1400	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	0.246
2	TB3-300/400	Beam	No Messages	1800	(PO-LRFD-OS) 0.9D - Ev + Emhy (Sp)	0.246

Frame	DesignSect	DesignType	Status	Location	TLngCombo	TLngArea
Text	Text	Text	Text	mm	Text	mm2
2	TB3-300/400	Beam	No Messages	200	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	600	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	1000	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	1400	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	1800	(PO-LRFD-OS) 0.9D - Ev + Emhy	0

Frame	DesignSect	DesignType	Status	Location	TTrnCombo	TTrnRebar
Text	Text	Text	Text	mm	Text	mm2/mm
2	TB3-300/400	Beam	No Messages	200	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	600	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	1000	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	1400	(PO-LRFD-OS) 0.9D - Ev + Emhy	0
2	TB3-300/400	Beam	No Messages	1800	(PO-LRFD-OS) 0.9D - Ev + Emhy	0

Reinforcement Configuration of Tie Beam TB1

A. INTRODUCTION

This document covers the configuration of beams reinforcement. The analysis of the reinforcement is conducted in SAP2000 with outputs of rebars area for longitudinal rebars and rebars area per length for transfersal rebars.

A.1. Beam Geometric Data

Width of the beam	b =	300	mm
Height of the beam	h =	400	mm
Concrete clear cover	cov =	50	mm
Diameter of longitudinal rebars	D_L =	16	mm
Diameter of transfersal rebars	D_t =	13	mm

B. LONGITUDINAL REINFORCEMENT

Area of a single rebar	A_r =	201.06	mm^2

B.1. Top Fibre

Rebars area at the top fibre (from SAP2000)	$A_{s.top}$ =	324.05	mm^2
Number of required rebars	n =	2.00	pcs
Number for first layer	n_1 =	2.00	pcs
Number for second layer	n_2 =	0.00	pcs
Distance between adjacent rebars	s =	142.00	mm

CONCLUSION: SATISFIED

B.2. Bottom Fibre

Rebars area at the bottom fibre (from SAP2000)	$A_{s.bot}$ =	324.05	mm^2
Number of required rebars	n =	2.00	pcs
Number for first layer	n_1 =	2.00	pcs
Number for second layer	n_2 =	0.00	pcs
Distance between adjacent rebars	s =	142.00	mm

CONCLUSION: SATISFIED

REINFORCEMENT CONFIGURATION IN USE 4D16

C. TRANSVERSAL REINFORCEMENT

Number of legs of the rebar	leg =	2.00	
Area of transversal reinforcement	A_v =	265.46	mm^2
Rebars area to space ratio (from SAP2000)	A_v/s =	0.70	mm^2/mm
Maximum required spacing	s_{req} =	381.41	mm
Spacing in use	s =	200.00	mm

CONCLUSION: SATISFIED

REINFORCEMENT CONFIGURATION IN USE D13-200

ATTACHMENT 3

Analysis of Foundation Stability and Reinforcement

ANALYSIS OF FOUNDATION F1

A. INTRODUCTION

This document covers the design and analysis of a shallow fondation. The aspects to be analysed in the design of shallow foundations include the bearing capacity, overturning and sliding stability. These tree aspects are the common probable failure modes of shallow foundations. All the theory in geotechnics in use are from (Das, 2011).

Geometry of Foundation:

Foundation width in x-direction		B_x =	1.50	m
Foundation width in y-direction		B_y =	1.50	m
Footing thickness		t_p =	0.40	m
Depth of foundation		D_f =	1.00	m

Soil Mechanical Properties:

Borehole reference				
Unit weight		γ =	31.18	kN/m^3
Effective interal friction angle	(data unavailable)	ϕ' =	0.00	deg
Soil cohession	(data unavailable)	c' =	0.00	kPa

B. BEARING CAPACITY

For the bearing capacity, we have computed the soil stress due to bearing loads and the moments from upper structure separately using SAP2000. And the soil stress bearing capacity has already been separately analysed in the soil report. The results are presented as follows.

Non-overstrength soil stress	q =	50.00	kPa
Overstrength soil stress	q_{OS} =	65.00	kPa
Soil bearing capacity (soil report)	q_{all} =	438.00	kPa
Increased bearing capacity due to overstrength	1.2 q_{all} =	525.60	kPa

The increase factor 1.2 is accordance with SNI 1726:2019 section 4.2.3.3.

The bearing capacity is satisfied if the following expressions hold:

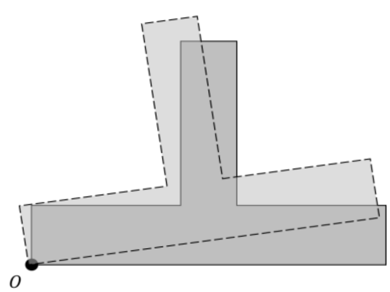
$$r_q := \frac{q}{q_{all}}\ ,\ r_{q.os} := \frac{q_{os}}{1.2 q_{all}} \leq 1$$

Stress ratio	r_q =	0.11
Stress ratio with overstrength	$r_{q.os}$ =	0.12

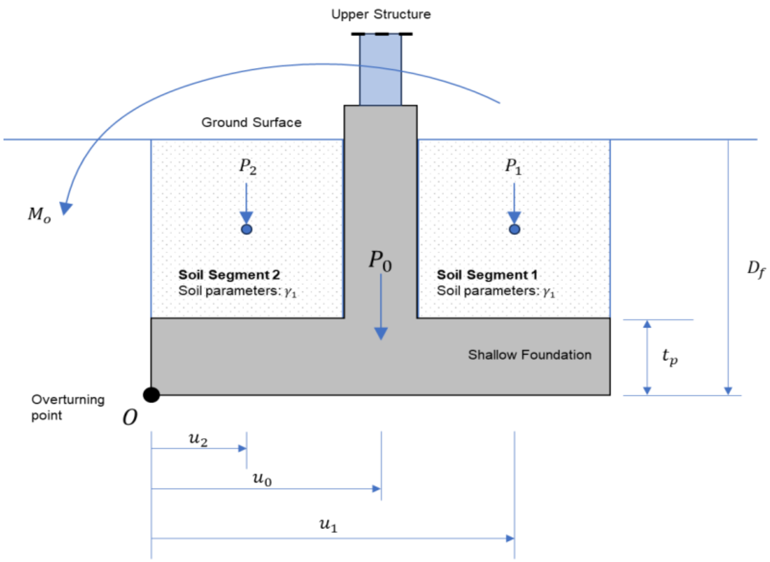
CONCLUSION: SATISFIED

C. OVERTURNING STABILITY

Overturning is another probable phenomenen in a shallow foundation. It is defined as the rotation of the foundation one of its toes (edges). Now let us observe the following illustration.



The measure to be considered in analysing the overturning stability is the ratio of the counter balance to the overturning. Note that overturning is caused by the moment from the upper strucure. Then, there shall be other moments resisting the overturning moment. Then the ratio is the ratio of the magnitudes of the sum of resisting moments to the overturning moment, known as the safety factor of overturning and denoted by FS_o. A more intuitive diagram is presented as follows.



From the diagram above, we can infer that the contributing resisting moments are from the weight of the foundation as well as the force from the upper structure parallel to the gravity, and the weights of soil segments above the footing. And the safety factor of overturning is given in the following expression:

$$\mathrm{FS}_o := \frac{1}{M_o}\sum_{k=0}^{2} u_k P_k \qquad \text{where} \quad \forall k \in \{1,2\}: \; P_k := \gamma_k L(D_f - t_p)$$

And the foundation is deemed stable against overturning if the following expression is satisfied:

$$\mathrm{FS}_o \geq 1.1$$

Note that overturning shall be evaluated in the axis with the worst overtuning.

C.1. Overturning Evaluation (Non-Overstrength)

Soil filling material unit weigth (the value is assumed)	γ_1 =	16.00	kN/m^3
Overturning moment (non-overstrength)	M_o =	4.97	kNm
Components of resisting moment-0:			
Vert. forces from the upper str. as well found. weight	P_0 =	34.26	kN
Overturning distance	u_0 =	0.75	m
Components of resisting moment-1:			
Width of foundation	L =	1.50	m
Depth of foundation	D_f =	1.00	m
Footing thickness	t_p =	0.40	m
Weight of soil segment-1	P_1 =	17.60	kN
Overturning distance	u_1 =	1.13	m
Components of resisting moment-2:			
Width of foundation	L =	1.50	m
Depth of foundation	D_f =	1.00	m
Footing thickness	t_p =	0.40	m
Weight of soil segment-2	P_2 =	17.60	kN
Overturning distance	u_2 =	0.38	m
Total resisting moment:	$\sum^{2} u_k P_k =$	52.09	kNm

$$k=0$$

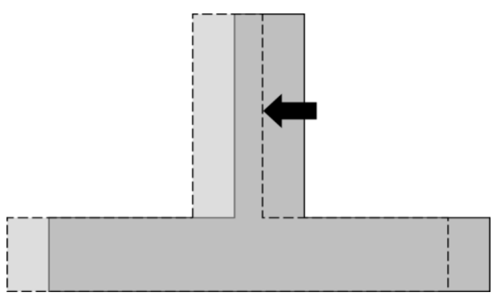
Safety factor of overturning	FS_o =	10.47
CONCLUSION: **SATISFIED**		

C.2. Overturning Evaluation (Overstrength)

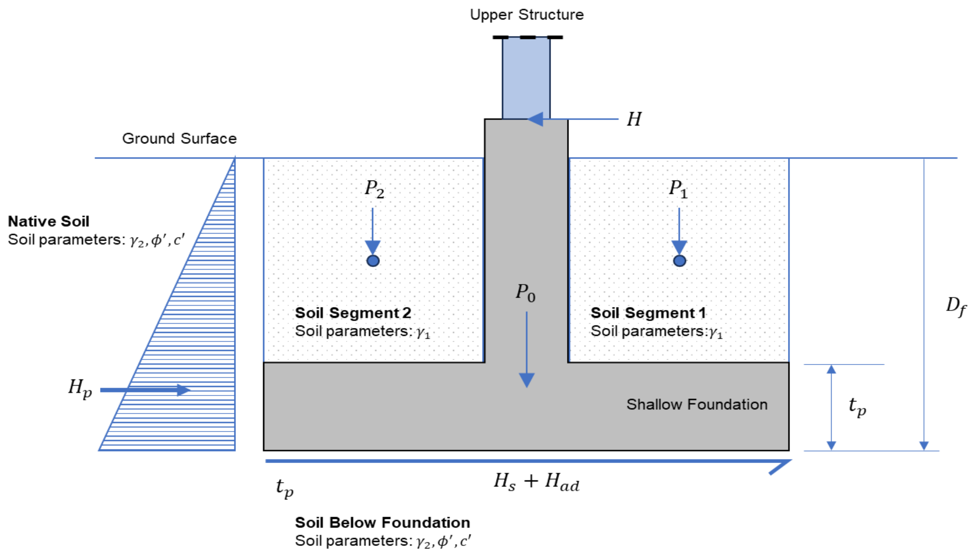
Soil filling material unit weigth (the value is assumed)	γ_1 =	16.00	kN/m^3
Overturning moment (non-overstrength)	M_o =	5.12	kNm
Components of resisting moment-0:			
Vert. forces from the upper str. as well found. weight	P_0 =	92.88	kN
Overturning distance	u_0 =	0.75	m
Components of resisting moment-1:			
Width of foundation	L =	1.50	m
Depth of foundation	D_f =	1.00	m
Footing thickness	t_p =	0.40	m
Weight of soil segment-1	P_1 =	17.60	kN
Overturning distance	u_1 =	1.13	m
Components of resisting moment-2:			
Width of foundation	L =	1.50	m
Depth of foundation	D_f =	1.00	m
Footing thickness	t_p =	0.40	m
Weight of soil segment-2	P_2 =	17.60	kN
Overturning distance	u_2 =	0.38	m
Total resisting moment:	$\sum_{k=0}^{2} u_k P_k =$	96.06	kNm
Safety factor of overturning	FS_o =	22.52	
CONCLUSION: **SATISFIED**			

D. SLIDING STABILITY

The phenomena of sliding in shallow foundations is described by the following illustration.



The foundation resistance against sliding takes forms of soil passive pressure, soil lateral resistance below the footing and the adhesive friction between the soil and the footing surface below the footing. A measure to identify the stability of foundations against sliding is the ratio of the sliding resistance to the sliding force, called safety factor of sliding and denoted by FS_s. Now let us observe the following diagram.



The components of sliding stability is presented as follows:

$$H_r = H_p + H_s + H_{ad}$$

where

$$H_p = \left(\frac{\gamma D_f^2 K_p}{2} + 2c' D_f \sqrt{K_p} \right) L \,,$$

$$H_s = \tan \delta' \sum_{k=0}^{2} P_k \,,$$

$$H_{ad} = B L c_a$$

and

$$K_p = \tan^2 \left(45 + \frac{\phi'}{2} \right) ,$$

$$\delta' = k_1 \phi' \,,$$

$$c_a = k_2 c' \,.$$

Note that ϕ' is the effective soil friction angle and k_1, k_2 are some factors satisfying $k_1, k_2 \in \left[\frac{1}{2}, \frac{2}{3}\right]$. And the foundation is deemed stable if the following expression is satified:

$$\mathrm{FS}_s := \frac{H_r}{H} = \frac{H_p + H_s + H_{ad}}{H} \geq 1.1$$

D.1. Sliding Evaluation (Non-Overstrength)

Half of internal friction angle + 45 degrees	0.5 ϕ' + 45 =	0.79	rad
Rankine coefficient of passive force	K_p =	1.00	
Soil lateral factor	k_1 =	0.50	
Soil adhesion factor	k_2 =	0.50	
Lateral friction angle	δ' =	0.00	rad
Soil-slab adhesion	c_a =	0.00	kPa
Passive lateral earth force	H_p =	23.39	kN
Soil lateral force below the footing	H_s =	0.00	kN
Adhesive force	H_{ad} =	0.00	kN

Resisting horizontal force	H_r =	23.39	kN
Sliding force (non-overstrength)	H =	4.55	kN
Safety factor of sliding (non-overstrength)	FS_s =	5.14	
CONCLUSION: SATISFIED			

D.2. Sliding Evaluation (Overstrength)

Half of internal friction angle + 45 degrees	$0.5\ \phi' + 45$ =	0.79	rad
Rankine coefficient of passive force	K_p =	1.00	
Soil lateral factor	k_1 =	0.50	
Soil adhesion factor	k_2 =	0.50	
Lateral friction angle	δ' =	0.00	rad
Soil-slab adhesion	c_a =	0.00	kPa
Passive lateral earth force	H_p =	23.39	kN
Soil lateral force below the footing	H_s =	0.00	kN
Adhesive force	H_{ad} =	0.00	kN
Resisting horizontal force	H_r =	23.39	kN
Sliding force (non-overstrength)	H =	8.92	kN
Safety factor of sliding (non-overstrength)	FS_s =	2.62	
CONCLUSION: SATISFIED			

E. REFERENCES

Das, Braja M. (2011). *Principles of Foundation Engineering* . Stamford, US: Cengage Learning.

Footing Reinforcement of F1 of Pipe Rack Cc1~Cc3

A. INTRODUCTION

This report covers the analysis of the reinforcement of shallow foundations.

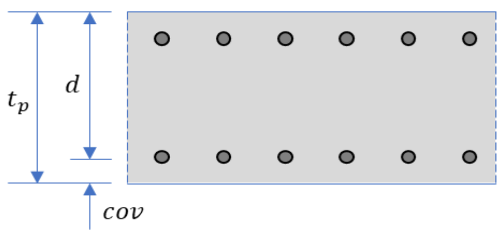
B. MATERIAL PROPERTIES

Rebar yield strength	f_y =	420.00	MPa
Concrete compressive strength	f_c' =	29.00	MPa
Whitney factor	β_1 =	0.84	

Whitney factor describes the rectangular stress block equivalent of concrete structure. The value of the factor is given in accordance with SNI 2847:2019 section 22.2.2.4.3 as follows:

$$\beta_1 = \begin{cases} 0.85 & : 17\ \mathrm{MPa} \le f_c' \le 28\ \mathrm{MPa} \\ 0.85 - \frac{0.05(f_c'-28)}{7} & : 28\ \mathrm{MPa} < f_c' < 55 \\ 0.65 & : f_c' \ge 55\ \mathrm{MPa} \end{cases}$$

C. MECHANICAL AND GEOMETRICAL PROPERTIES



Footing thickness	t_p =	400.00	mm
Footing width	b =	1500.00	mm
Rebar diamater (in x-direction)	D_x =	16.00	mm
Rebar diamater (in y-direction)	D_y =	16.00	mm
Clear cover	cov =	50.00	mm
Dist. from compressive fibre to centre of tensile rebar (x)	d_x =	326.00	mm
Dist. from compressive fibre to centre of tensile rebar (y)	d_y =	342.00	mm
Distance between adjacent rebars in use:			
For x-direction	s_x =	200.00	mm
For y-direction	s_y =	200.00	mm

D. DUCTILITY REQUIREMENT OF THE REINFORCEMENT

SNI 2847:2019 section 8.6.1.1 asserts that the minimum reinforcement of a concrete slab shall be of the following:

$$A_{s,\min} = \begin{cases} 0.0020A_g & : f_y \le 420\ \mathrm{MPa} \\ \max\left\{\frac{0.0018\cdot 420}{f_y}A_g\,, 0.0014A_g\right\} & : f_y \ge 420\ \mathrm{MPa} \end{cases}$$

And the reinforcement also cannot exceed the following requriement:

$$A_{s,\max} = 0.75\rho_b A_g$$

$$\rho_b = \frac{0.85 f'_c \beta_1}{f_y} \frac{600}{f_y + 600}$$

The ductility of the Reinforcement in use is satisfiable if

$$A_{s,\min} \le A_s \le A_{s,\max}$$

holds.

D.1. Ductility in X-Direction

Gross sectional area	A_g =	489000.00	mm^2
Reinforcement sectional area	A_s =	1507.96	mm^2
Required minimum rebars	$A_{s,min}$ =	870.42	mm^2
Required maximum rebars	$A_{s,max}$ =	10671.93	mm^2

CONCLUSION: SATISFIABLE

D.2. Ductility in Y-Direction

Gross sectional area	A_g =	513000.00	mm^2
Reinforcement sectional area	A_s =	1507.96	mm^2
Required minimum rebars	$A_{s,min}$ =	913.14	mm^2
Required maximum rebars	$A_{s,max}$ =	11195.70	mm^2

CONCLUSION: SATISFIABLE

E. FLEXURAL BENDING CAPACITY OF REINFORCEMENTS

The reinforcement in use shall have a sufficient capacity against the ultimate bending moment in the footing foundation. The nominal bending capacity of the reinforced concrete structure is given by

$$M_n = f_y A_s \left(d - \frac{a}{2}\right)$$

where

$$a = \frac{f_y A_s}{0.85 f'_c b} .$$

The bending capacity is satisfiable if the expression

$$r_b = \frac{M_u}{\phi M_n}$$

is satisfied, where Mu is the ultimate bending moment and ϕ = 0.9.

E.1. Bending Capacity in X-Direction

Ultimate bending moment (overstrength considered)	M_u =	53868440.00	Nmm/m
Equivalent thickness of concrete compressive fibre	a =	17.13	mm
Nominal moment	M_n =	201046209.21	Nmm/m
Strength ratio	r_b =	0.30	

CONCLUSION: SATISFIABLE

E.2. Bending Capacity in Y-Direction

Ultimate bending moment (overstrength considered)	M_u =	54785650.00	Nmm/m
Equivalent thickness of concrete compressive fibre	a =	17.13	mm
Nominal moment	M_n =	211179730.47	Nmm/m
Strength ratio	r_b =	0.29	

CONCLUSION: SATISFIABLE

F. SUMMARY

The result of the reinforcement design is presented in the following table:

Direction	Reinforcement
X	D16-200
Y	D16-200