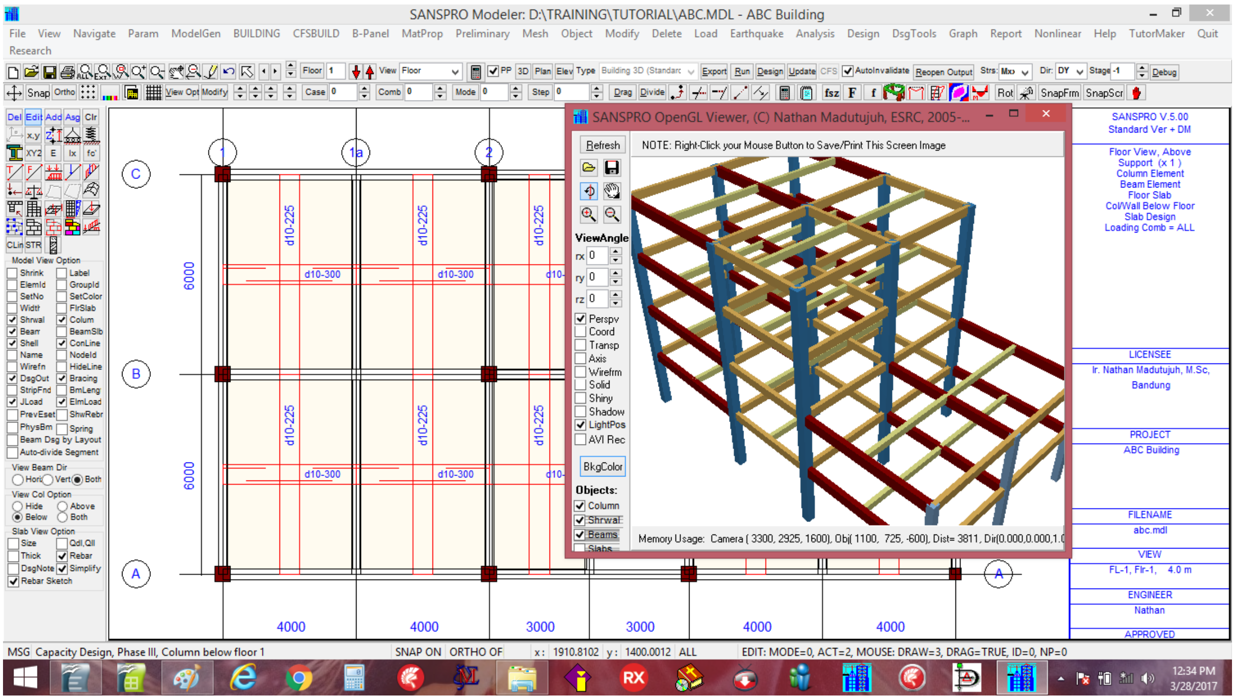
Engineering Software Research Center

Jl. Situ Aksan 29, Bandung 40221, INDONESIA
Tel/Fax: 022-603595, 6041685, 6003596
E-mail: esrc.nathan@gmail.com, Web: www.esrcen.com

SANSPRO V.5.10 Tutorial

by
Nathan Madutujuh



March 2017

SANSPRO V.5.10 TUTORIAL
Table of Contents

1. Installation

Step by step to install the SANSPRO program:

1. Insert the provided CD
2. Run SETUP.EXE
3. Follow the instruction, install to a new directory (for example : c:\Program Files\SANSPRO510)
4. Copy the user license file SANSPRO.UIF from CD (inside your name's directory)
5. Right-Click the installed directory (SANSPRO510), change properties : **[ ] Read Only**

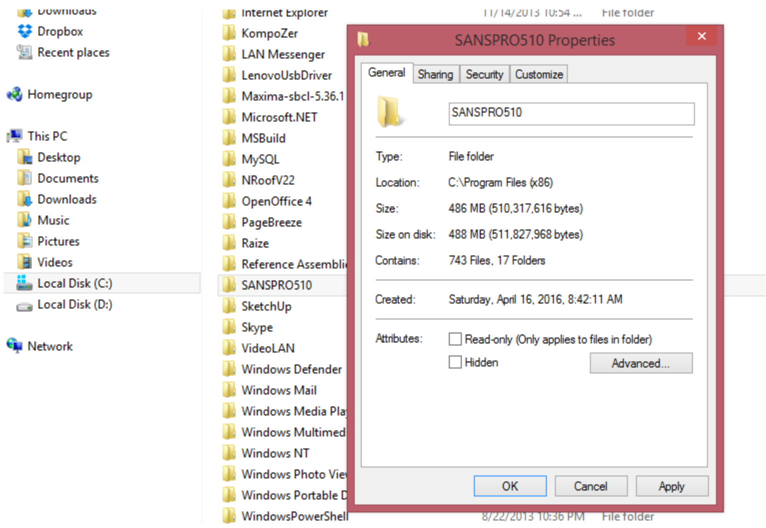


Fig.1. Set Directory for NOT Read Only

6. Select SANSPRO.EXE, Right-click, set **[x] RUN AS ADMINISTRATOR**

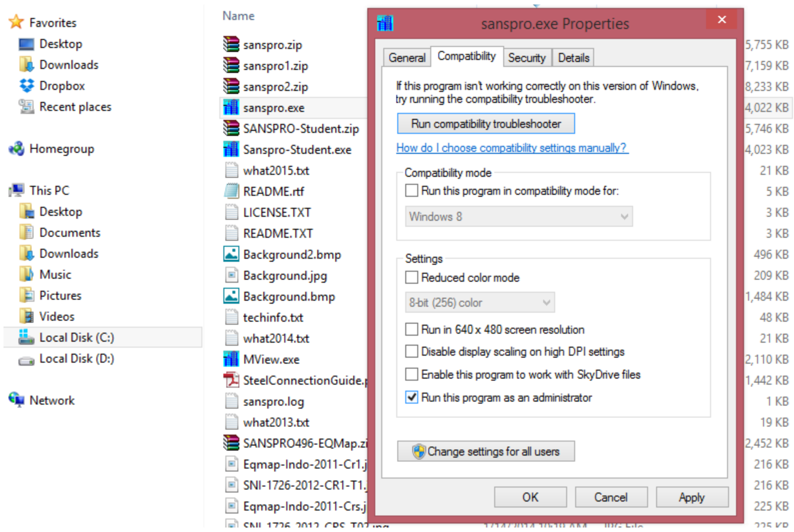


Fig.2. Set SANSPRO to "Run this program as an administrator"

7. Select SANSPRO.EXE, Right-click, select SEND TO DESKTOP to create Short-cut if needed
8. SANSPRO will be appear on desktop and ready to run
9. User can selected 7 supported languages from **Param – User preferences** menu

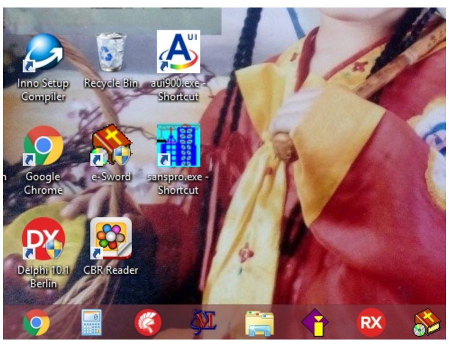


Fig. 3. Short-cut for SANSPRO.EXE on desktop

2. About ESRC

ESRC is a research center founded by Dr. Nathan Madutujuh in 1989 in Bandung, INDONESIA. ESRC is focusing on development of advanced engineering software using latest numerical methods, software development tools, and available hardware technologies. ESRC provides the engineering software at affordable cost to equip more engineers with better tools to create better environment for the world.

Our Services

We provides our engineering software and also special purpose software designed for specific industry needs. ESRC can also collaborate with other academic or research institutes to develop specific software for special needs. With our vast knowledge and million lines of available source code and library, both numerical and 2D/3D graphics, we can deliver new software in short time.

Our Products:

Building Design:

SANSPRO - Integrated Building Design
DSGWIN - Concrete Section Designers
BEAMCOL - Beam Column Joint Design
CONBRIDGE - Concrete Bridge Design
CAMBER - Bridge Camber Analysis

Steel Structures:

STEELCON - Steel Connection Design
TOWERWIN - Steel Lattice Tower Design
PURLIN - Steel Purlin Design
NROOF - Light Steel Roof Design
NBRIDGE - Steel Composite Bridge Design
GABLE - Preliminary Gable Frame Design
MATPLAN - Steel Material Planning

Foundation Design:
RETWALL - Retaining Wall Design Program
GRAVWALL - Gravity Wall Design Program
BASEMENT - Concrete Basement Design
PILECAP - Concrete Pilecap Design
TOWERPAD - Tower Foundation Design
FOOT - Spread Footing Program

Geotechnics:
SOILAB - Soil Laboratory Data Processing
BORLOG - Boring Log Program
NSLOPE - Slope Stability Analysis Program
NPILE - Static Pile Capacity Program
REESE - Pile Lateral Capacity Design

Transportation:
WinCANDE - Nonlinear Box Culvert Design
BRASDAT - BRASS Post-Processing
SapBRIDGE - SAP Bridge Post-Processing
INSPECT - Bridge Inspection Program

Retrofitting and Evaluation:
TDS302 - Data Acquisition System
SFRS - Fiber Wrap Strengthening Design

CONTACT ADDRESSES:

ESRC Office:
Jl. Situ Aksan 29
Bandung 40221
INDONESIA
Web: www.esrcen.com
E-mail: esrc.nathan@gmail.com

ESRC Training Centre:
Jl. Situ Aksan 29
Bandung 40221
INDONESIA

E-mail: esrc.reg@gmail.com

3. Introduction to SANSPRO Program

SANSPRO is an Integrated Software for Structural Modeling, Finite Element Static & Dynamic Analysis, Steel and Concrete Design, Foundation Design, Volume & Cost, and Detail Drawing Generator. Using integrated approach, user needs only to enter a single comprehensive model, then other process such as analysis, design and reporting will be done using the same model.

SANSPRO can be used for 2D or 3D structures and has a very comprehensive interactive building modeler using layout oriented approach. It has capabilities to do static and dynamic spectrum response analysis, and also Direct Integration (Time History) Analysis. For building with stage construction, a sequential load analysis can be done to reflect more accurately the distribution of gravity load during construction, especially for transfer beam and bracings.

Truss elements, Frame elements and shell finite elements area available to model various beam, column, floor slab, shearwall and shell structures. For shell element formulation, there are several options available : Hybrid formulation, advanced ANDES and Discrete Kirchoff element with Drilling DOF.

The program also provide various load table and load generator for Self-weight, Floor slab (DL,LL), Static Equivalent earthquake load, Response spectrum load, mass calculation, and also wind load.

After analysis, one can choice to run a steel / concrete design, or a concrete capacity design using the latest building design code. Design results, including rebar number, floor slab rebar sketch, shearwall rebar detail can be shown and checked visually. Optionally, user can generate a detail drawing for building floor layout, column rebar, and beam rebar.

Simplified and also more detailed Capacity Foundation Design methods are also provided along with Foundation Detail Drawing Generator. Using this approach, designing a pile foundation for whole building can be done in few seconds.

For building model, user can calculate the total volume and cost per floor using the user provided unit price. The program will provide concrete volume and weight, steel weight, and formwork area. The final result will be concrete volume to area ratio, steel weight to area or concrete volume ratio, and total cost per floor area.

A comprehensive building model, analysis and design report can also be generated to a PDF file format.

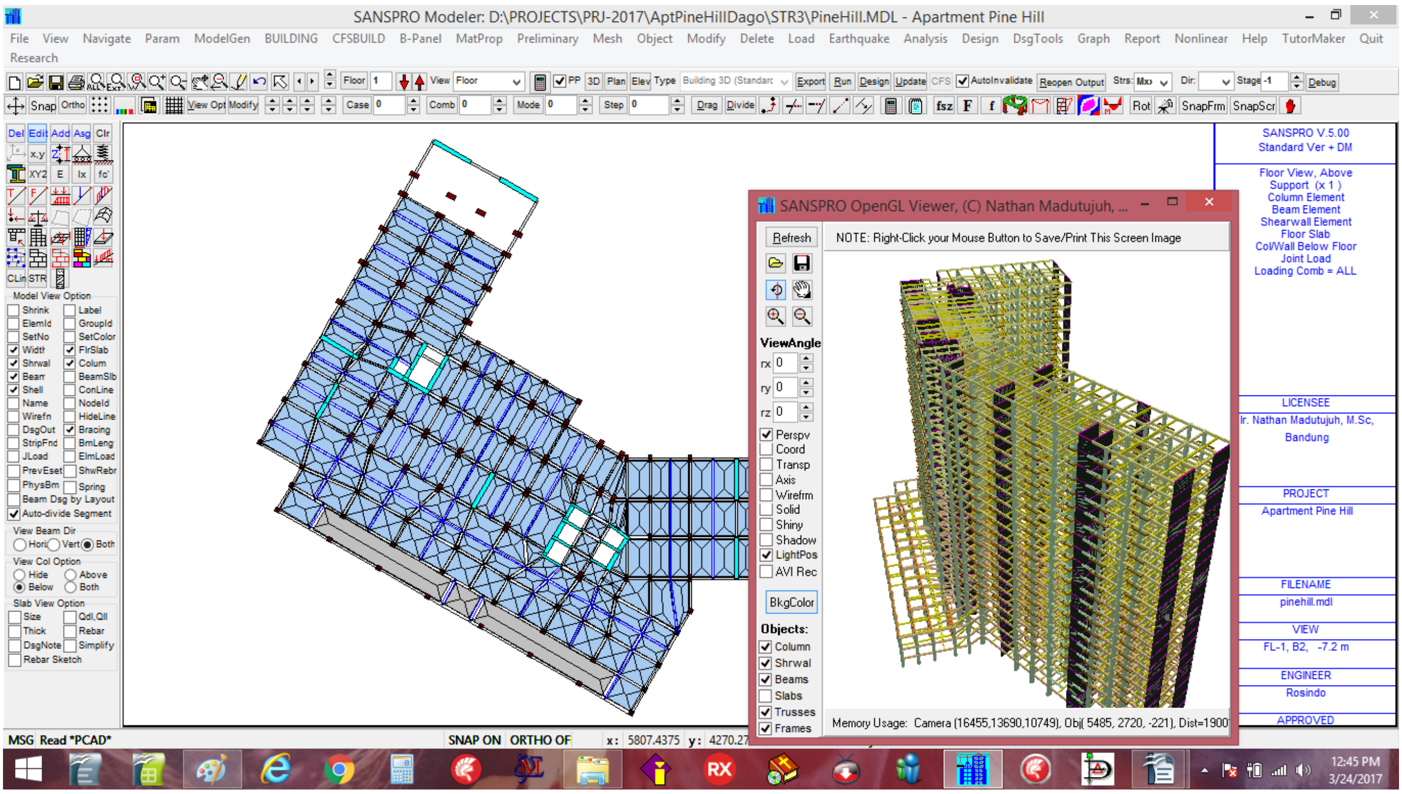
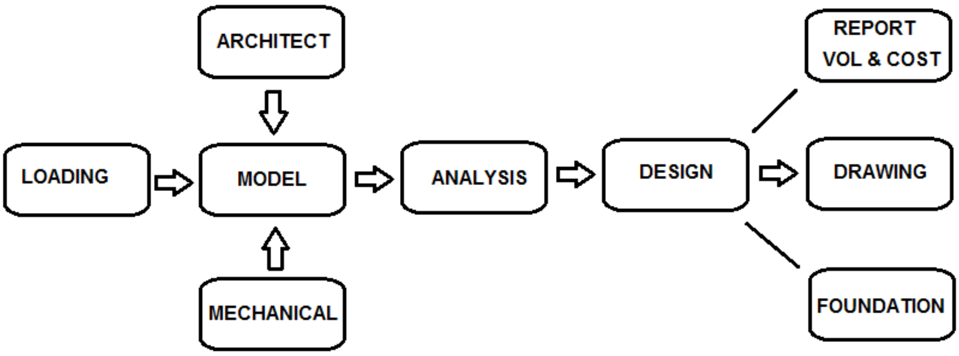


Fig. 4. Main menu of SANSPRO V.5.10 Program

4. Program Layout



5. Design Code supported

SANSPRO support latest available design codes, among them are:

Earthquake Load Generator	: Indonesian Code 1726-2012, UBC-94, UBC-97, IBC-2003, IBC-2009, ASCE-7-10 Mexico-1993
Concrete Design Code	: Indonesian Code 2847-2013, ACI-2015
Steel Design Code	: Indonesian Code 1729-2013, AISC-2010
Foundation Design Code	: Indonesian Code 2847-2013

Other design code can be supported by changing the appropriate design parameters:

Load factors
Material strength factors
Earthquake Zone factor (Ss, S1)

6. Design Procedures

SANSPRO is a computer program that can be used for designing various type of structures: Continuous beams, Portal frames, Plane Truss, Factory Building, and Highrise Buildings. It has the following capabilities:

a. Structural modelling
b. Structural analysis
c. Concrete and steel design
d. Steel connection design for truss and frames
e. Concrete slab design
f. Foundation design
g. Detail Drawing generator
g. Volume and Cost calculation
h. Report generator

Procedure for using SANSPRO program is as follows:

1. **Model creation (File *.MDL)**

   a. Using **Model Generator (to generate simple model for exercise)**

      1. Continuous beams
      2. Plane truss
      3. Portal + bracing
      4. Gable frame
      5. Simple Building
      6. General Building

   b. Using **Axis lines**
   c. Using **Import Mesh** from AutoCad DXF Files
   d. Using **Import Construction Lines** (help lines) from AutoCad DXF Files
   e. Using **Interactive method** (one by one)

2. **Data Checking (Export Data, File *.DAT)**

   a. Using **Model Advisor**
   b. Preparing for Analysis (Export Data)

3. **Structural Analysis (Output file *.OUT)**

   a. Static Analysis (to find displacement, reactions, element forces)
   *.OUT - Displacement, element forces, reactions
   *.SUP - Support reactions summary

   b. Eigen Value Analysis (to find vibration mode only)
   *.RSP - eigen values file

   c. Dynamic Spectrum Response Analysis (Dynamic analysis)
   *.BSH - Dynamic Spectrum response output, time period, base shear
   *.ROT - Floor eccentricity
   *.DRF - Story Drift and Building separation
   *.SSH - Story Shear and P-Delta Checking

   d. Direct Integration Analysis (Time History/Impact Load)
   *.DVA - Displacement, Velocity, Acceleration for each time step

4. **Structural Design**

   a. Concrete Frame/Building Design (ACI/PBI, Output *.D02)
   b. Steel Frame/Building Design (ASD/LRFD, Output : *.D02)
   c. Prestressed Concrete Design (Output : *.D02)
   d. Steel Truss Design (Output : *.D01)

e. Concrete Building Capacity Design (Output: *.C01, *.C02, *.C03, *.C04)

5. Foundation Design

a. Concrete Footing Slab Foundation Design
b. Concrete Mini Pile Foundation Design
c. Concrete Bored Pile Foundation Design

6. Detail Drawing Generator

a. Structural Layout
b. Floor slab rebar layout
c. Beam Layout and Detail
d. Column Layout and Detail
e. Shearwall rebar layout
f. Foundation layout and detail
g. Other structural detail

7. Volume and Cost Report

a. Installed unit price
b. Formwork area
c. Concrete Volume
d. Steel and Rebar weight
e. Concrete volume / area ratio
f. Steel weight per concrete volume ratio
g. Steel weight per area ratio
h. Total Volume, Weight, Ratio and Cost per floor

8. Report Generator

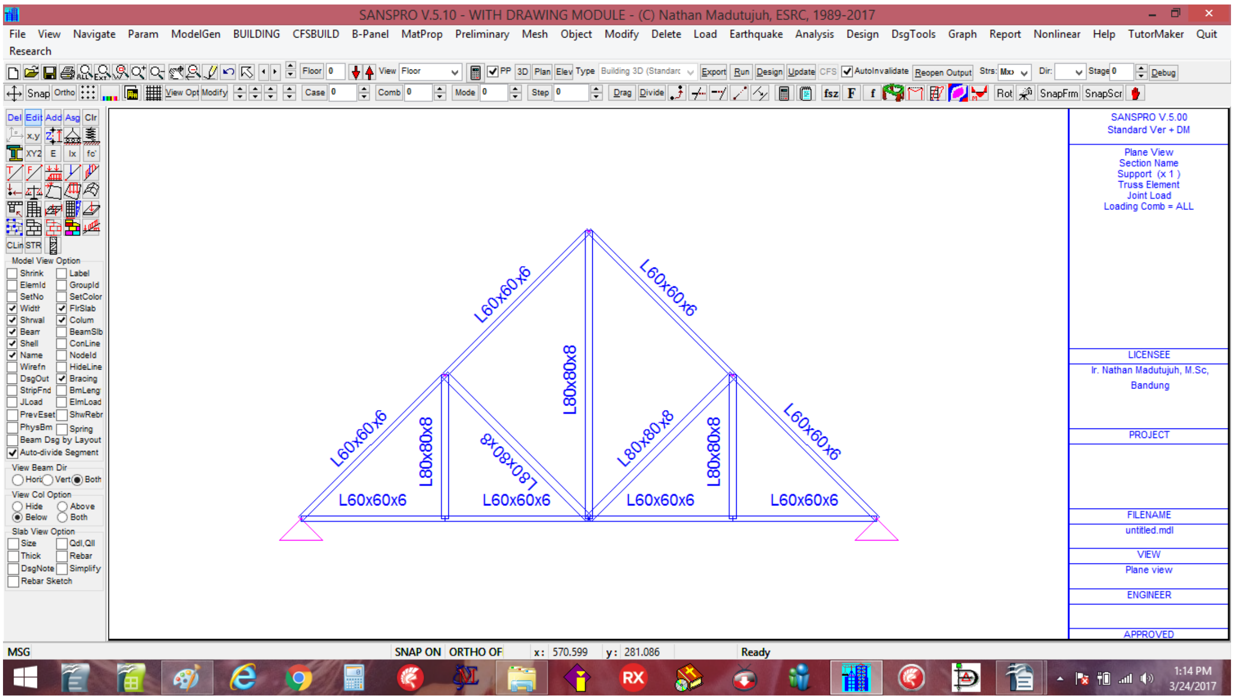
a. Structural Data Summary
b. Comprehensive Building Report (File *.BTR)
c. Dynamic Analysis Result Checking
d. Comprehensive Report Generator (File *.PDF)

7. Basic Example:

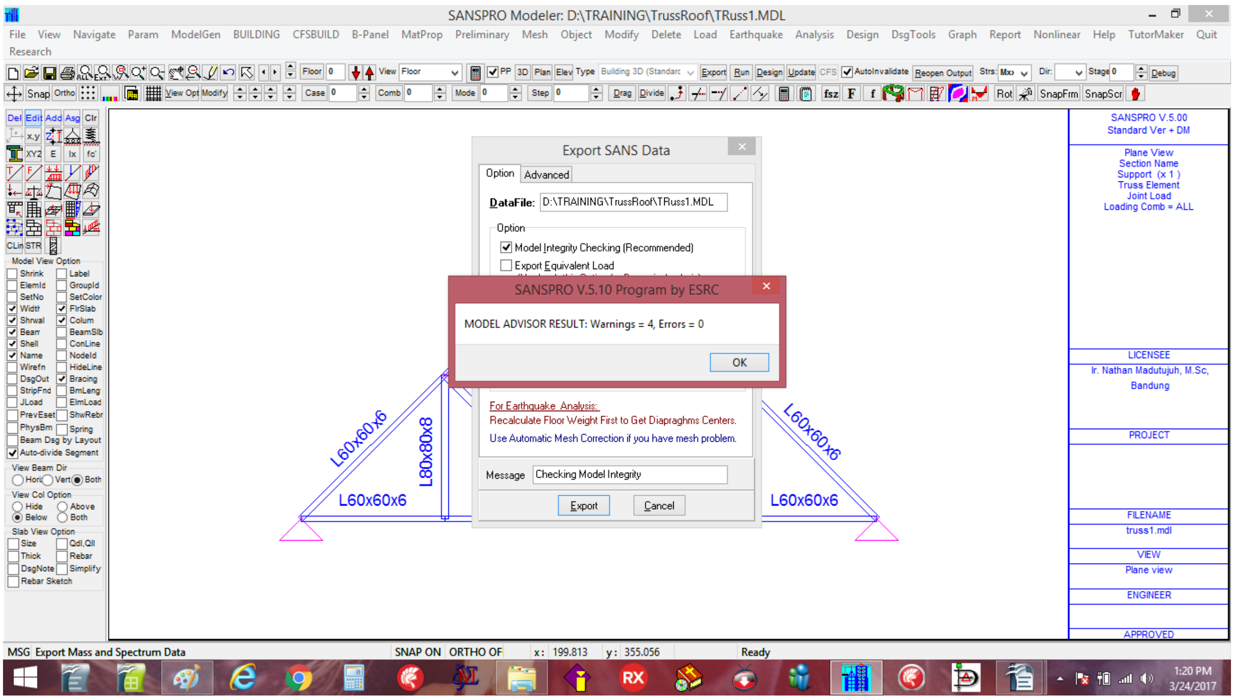
1. Example 1: Simple Truss Design

A Roof Truss usually made from steel angle connected by bolt to conenctions plates.
Load from DL, LL, Wind load will be considered.

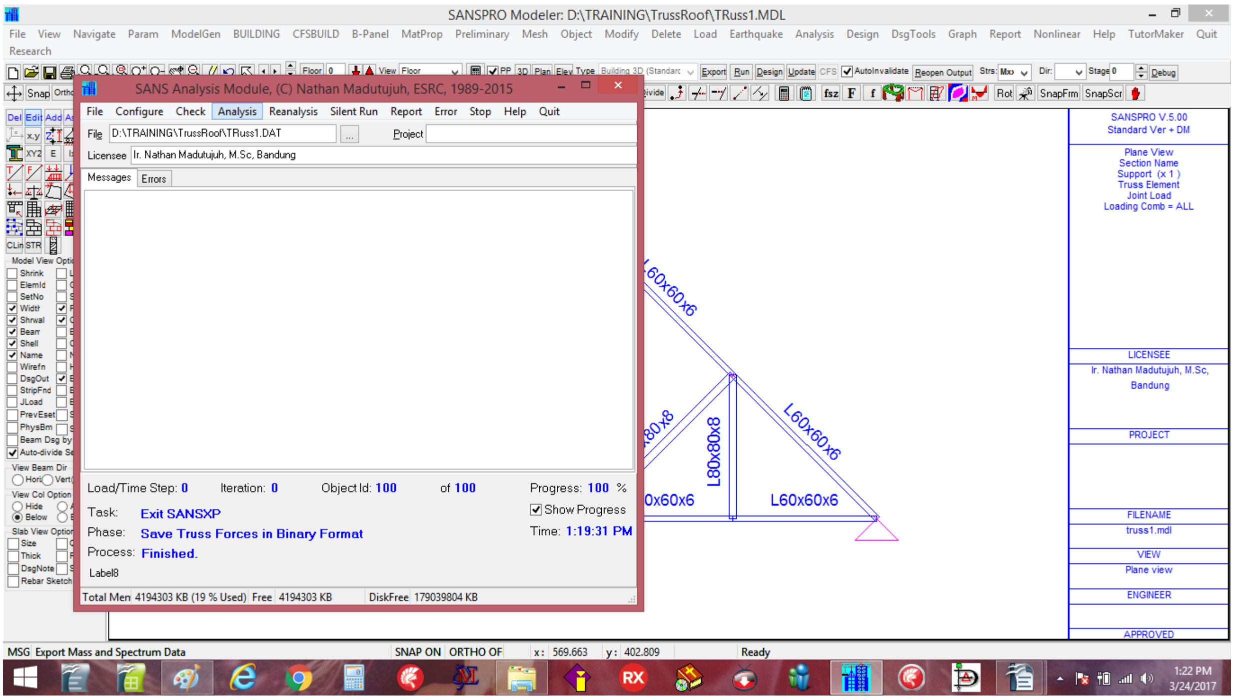
1. Open Modeler
2. Click menu : **ModelGen – Roof Truss (Simple)**
   Click : **Y**
   Click : **OK** (4 times, change data if necessary)
3. A simple truss model will be appeared, click icons **[ ] Name** to display section name



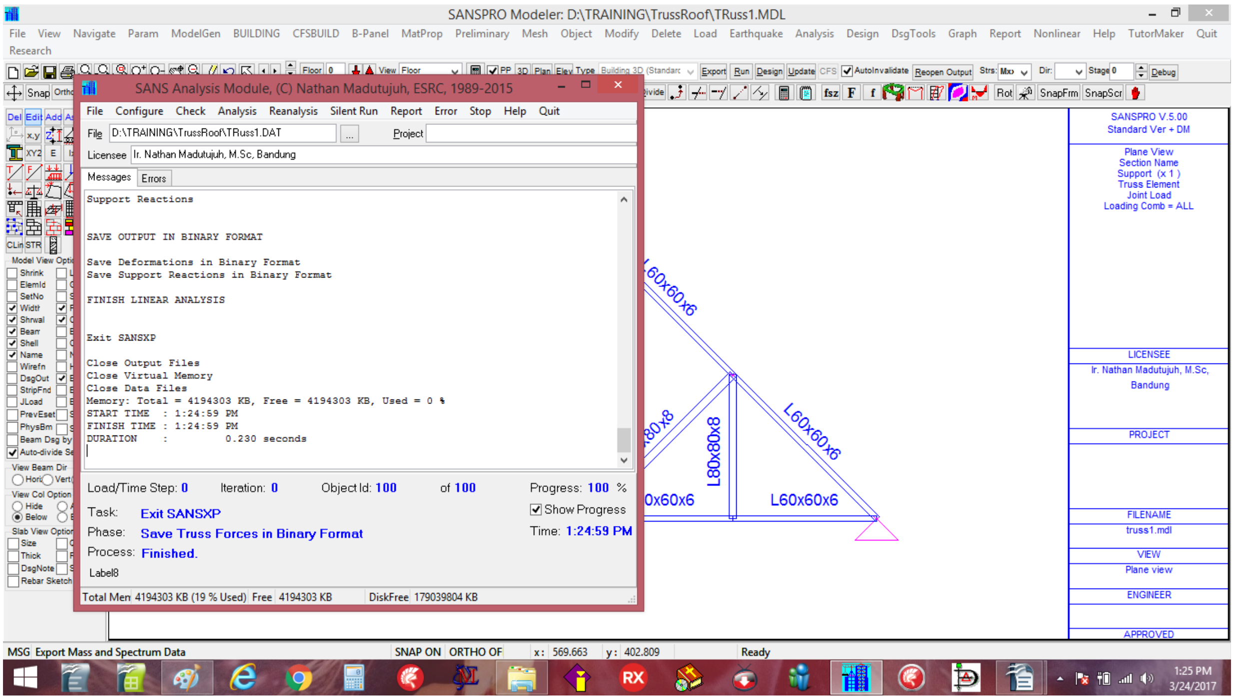
4. Click : **File – Save** (or Press F2) → Name it as Truss1.MDL
5. Click : **File - Export** (or Press F4) → Click **[Export]** – Click **[OK]** - Click **[Continue]** – Click **[OK]**



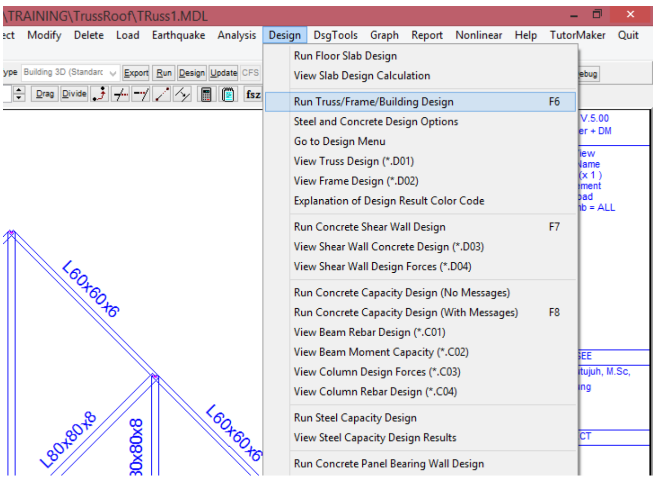
6. To run analysis: Click menu **Analysis**, Click **Analysis Menu**, a window will be appear



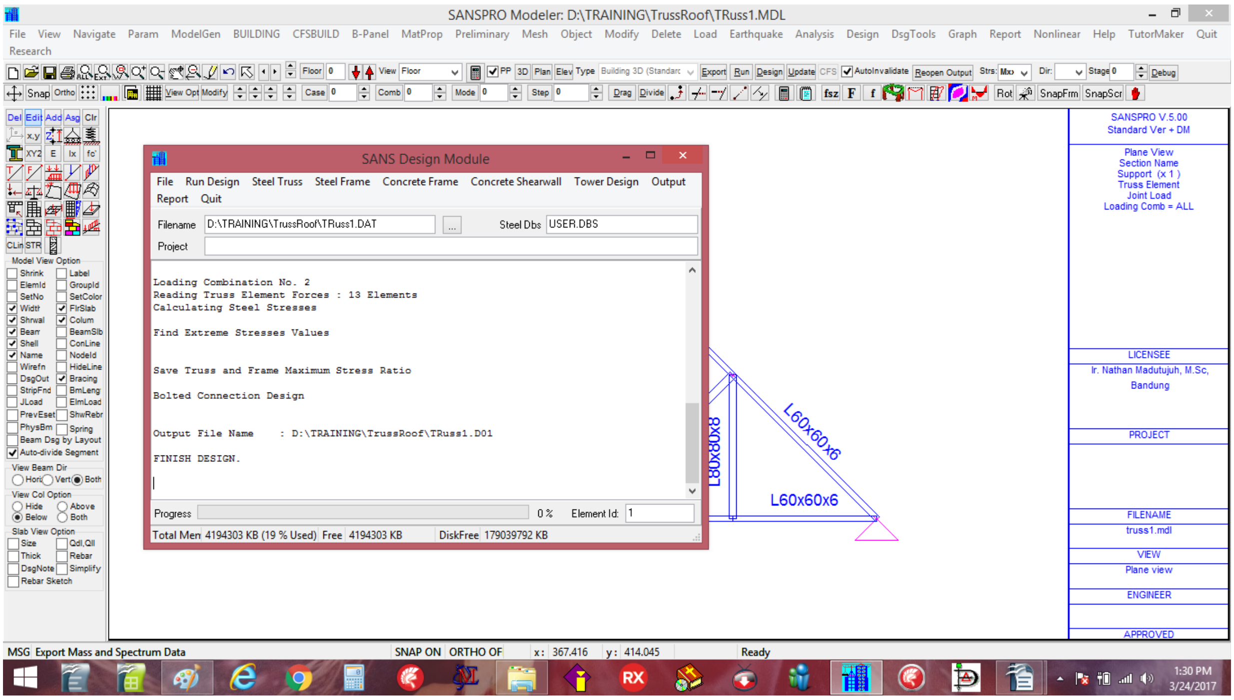
7. Click **Analysis** to start analysis procedure, then click **Quit**



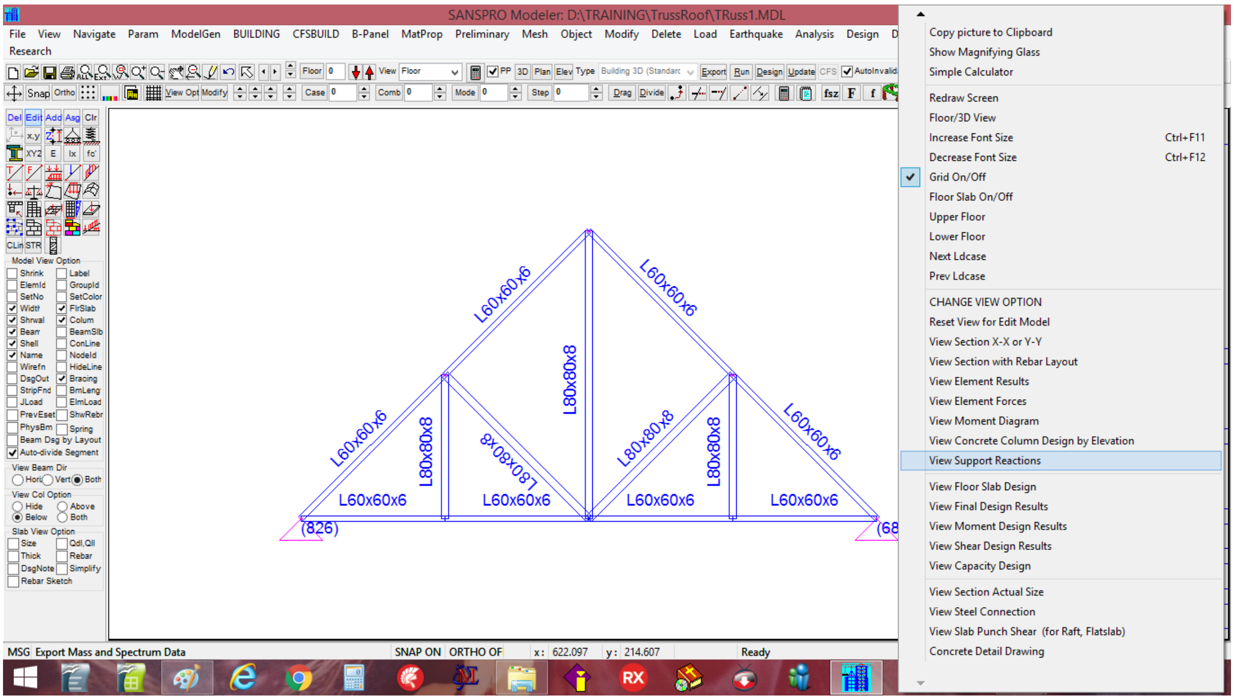
8. To run a steel truss design, Click **Design – Run Truss/Frame/Building Design,** a window will appear



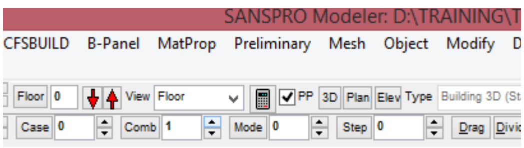
9. Click **Truss Design** – Click **[Start Design]** – Click **[OK] –** Click **Quit**

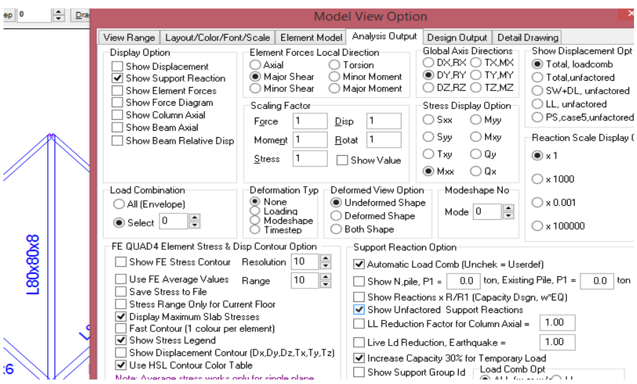


10. To View Truss Reactions : **Right-Click**, Select **View Support Reactions**

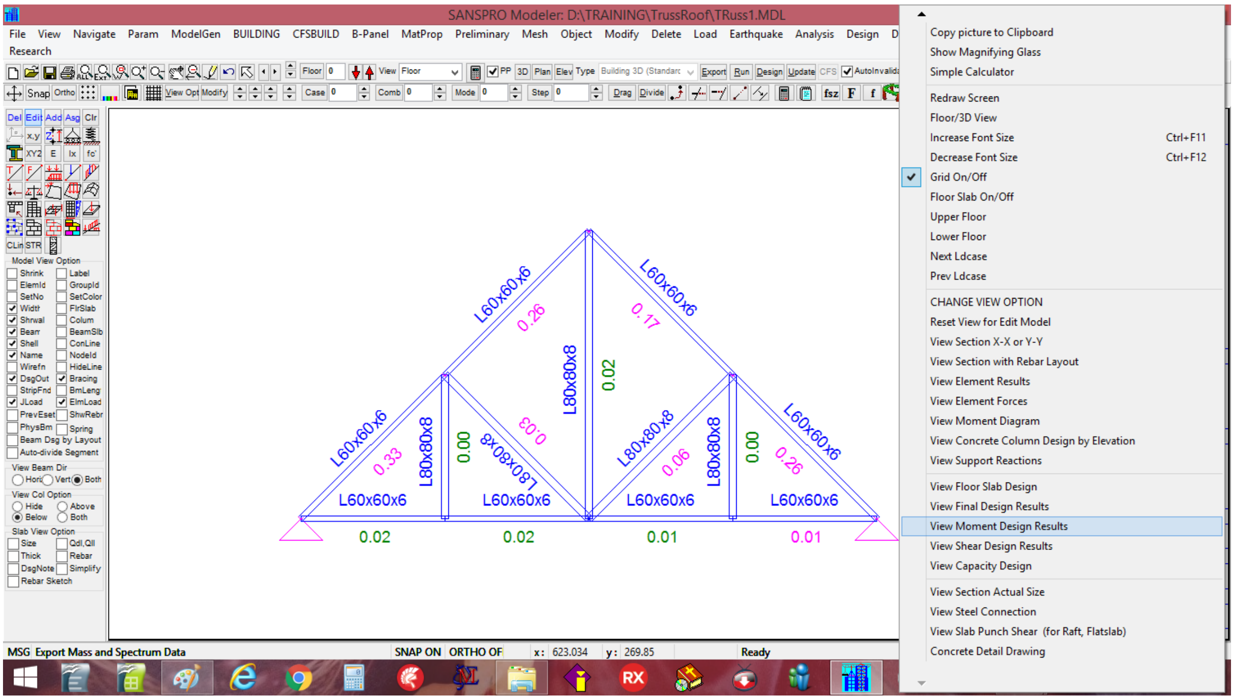


Support reactions (factored) from all load combinations will be appear.
To see just support reaction from a certain load combination, select the right Load **Comb** number.
To see unfactored support reactions, **Right-Click**, select **Change View Option**, **[ ] Show unfactored support**



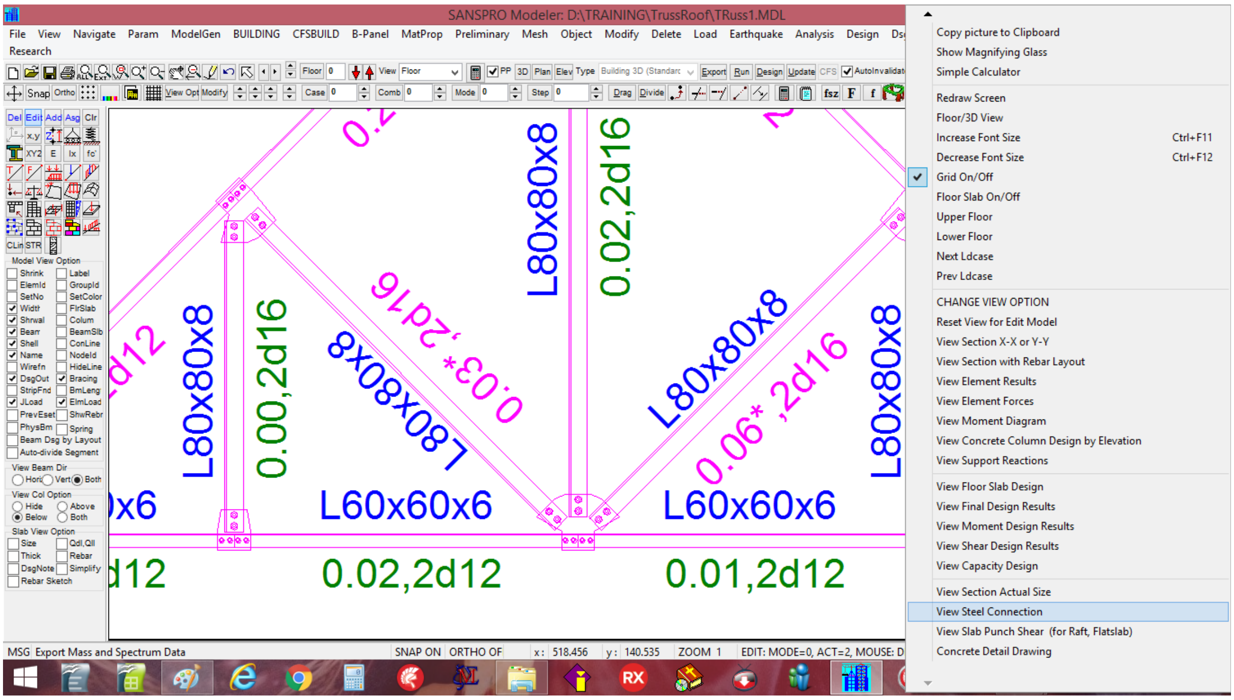


11. To see truss design results, **Right-Click**, select **View Moment Design Results**



Steel truss stress ratio will be appear. Design is OK if the ratio is fr <= 0.85 for truss with eccentric connection or single L diagonals and fr <= 1.0 for symmetric connection.

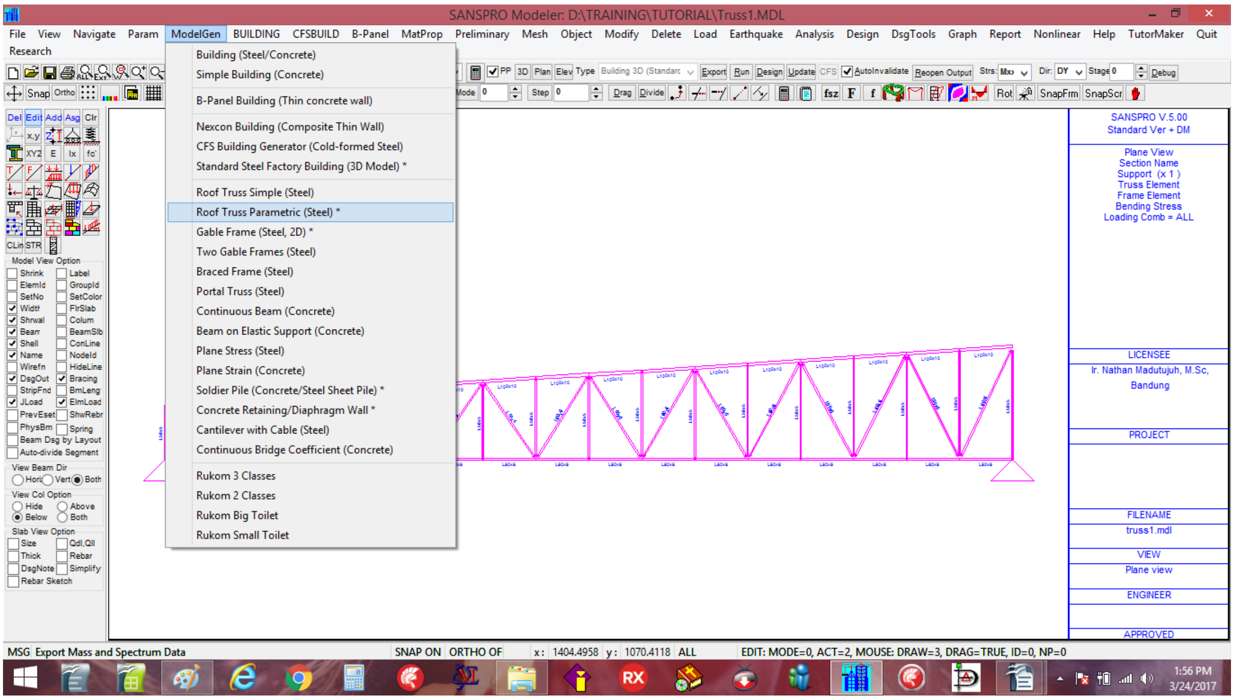
12. To see steel truss bolted connection, **Right-Click**, select **View Steel Connection**



NOTE:

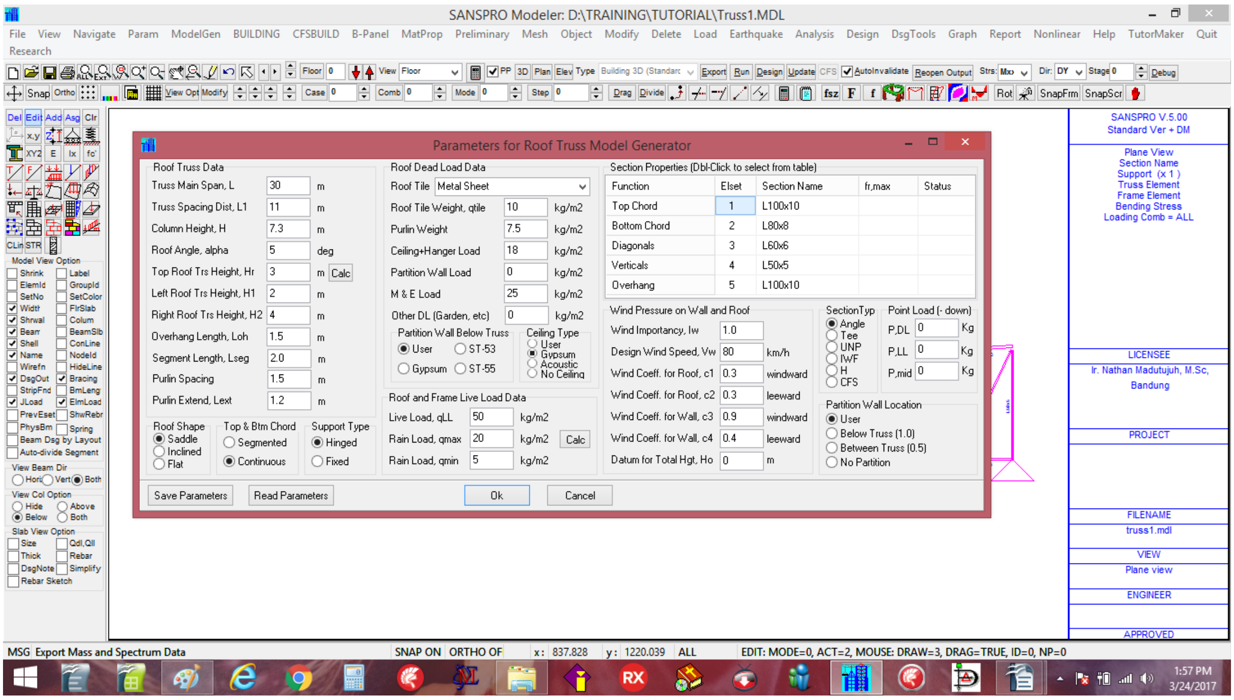
- The procedure for **Save, Export, Analysis**, and **View Support Reactions** are same for all models, so it will not be repeated again in this tutorial
- The program will automatically save a backup file every t minutes (if asked) or after every SAVE command
- **Design Code** and **Load factors** can be changed by user before analysis
- After a model has been created, user can repeat analysis and design anytime
- Various more complex truss system can be generated by selecting the second menu : Roof Truss Parametric

Generating Roof Truss by Parametric Method



User can select various parameters suitable for the model required:

> Truss span, section, roof shape, support type, roof tile type, ceiling type, live load, wind load, point load, section type, etc.

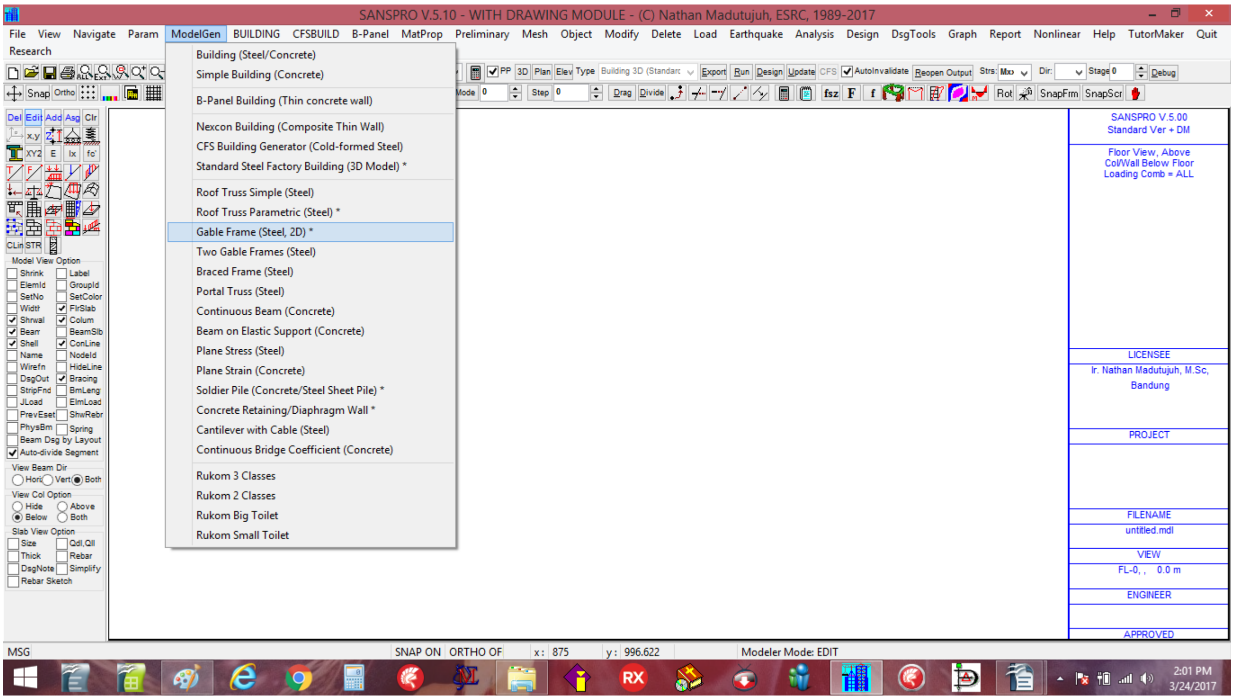


After a truss model generated, the procedure for analysis, design, visual design checking are the same as previous.

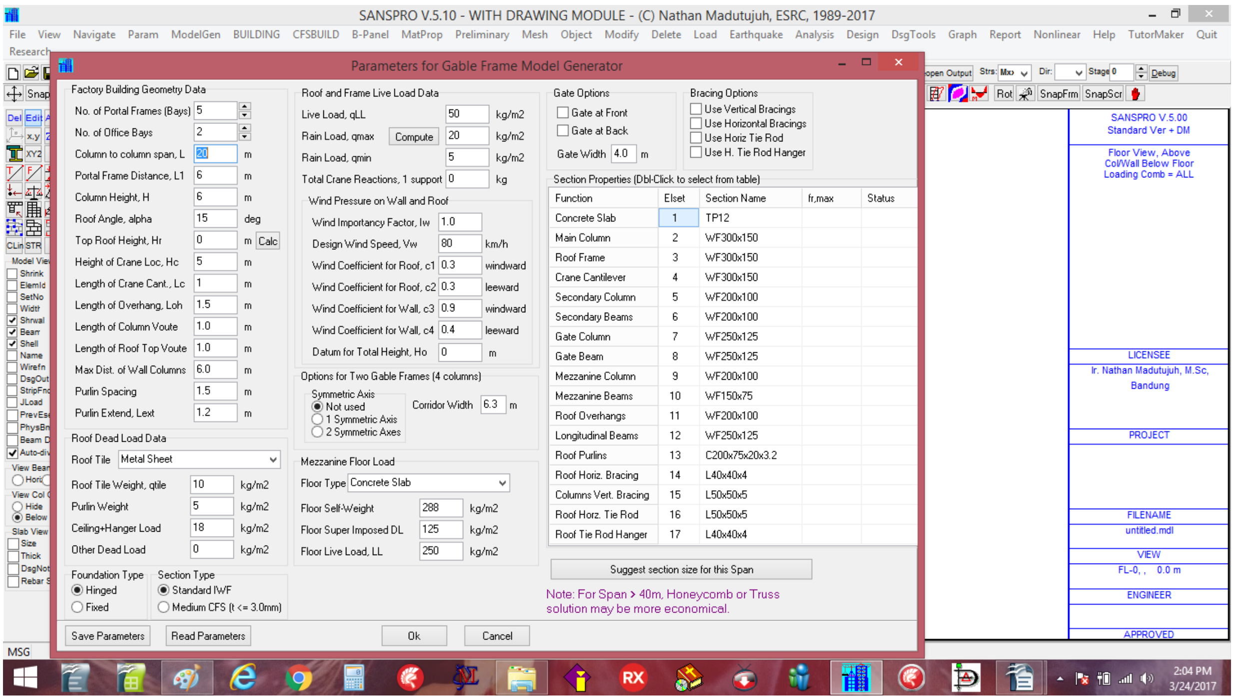
2. Example 2: Gable Frame Design

A Gable frame is usually found at factory building. It may have a corbel to hold rail for crane.
Load to be considered: DL, LL, Wind Load, Crane Load, Earthquake Load (for heavy Concrete frame)

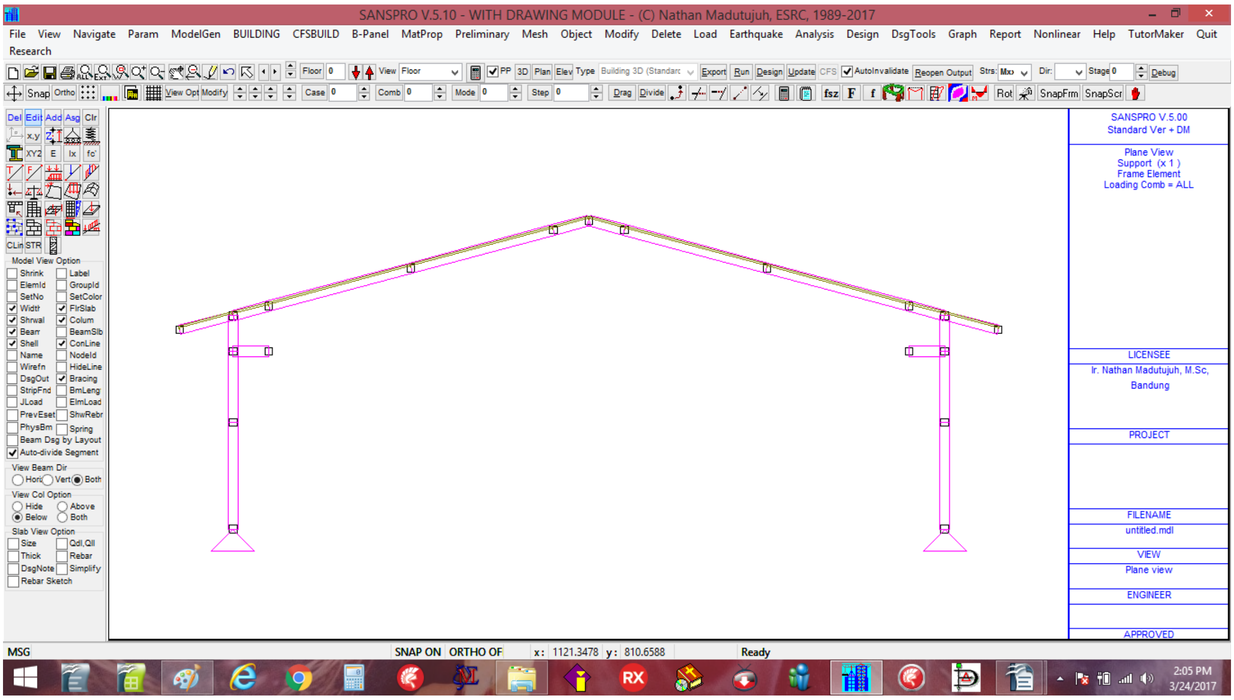
1. Open Modeler
2. Click menu : **ModelGen – Gable Frame (Steel, 2D)**
   Click : **Y**



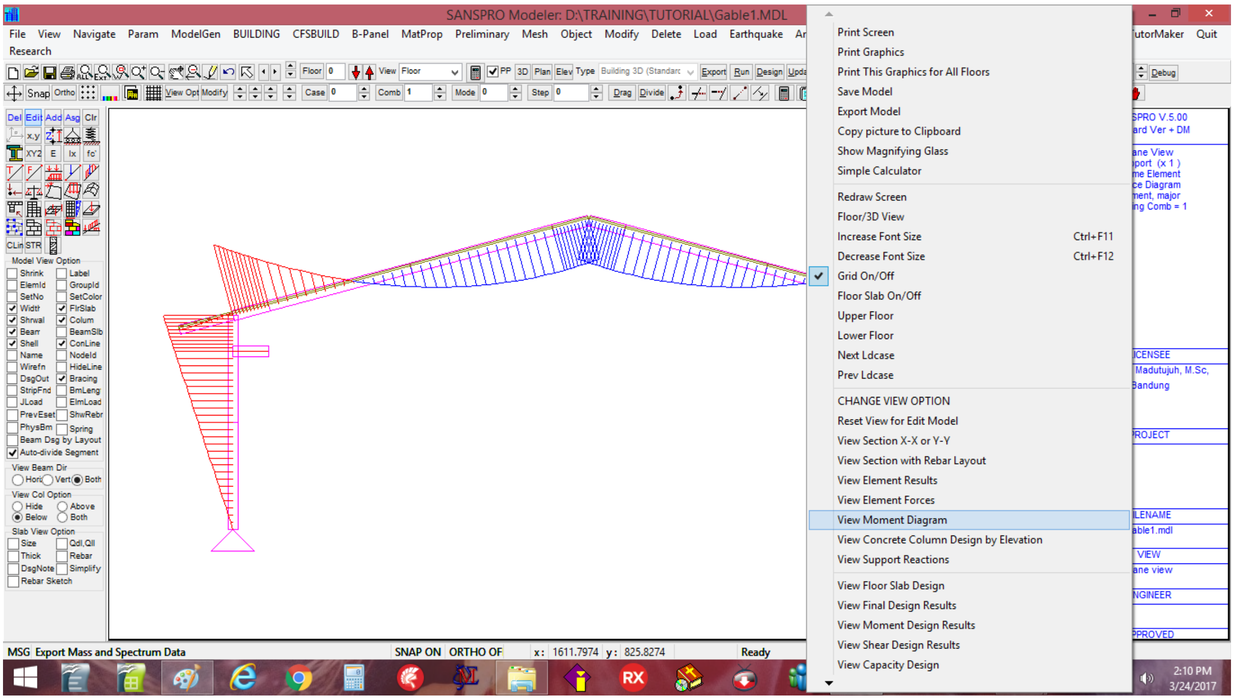
3. A **Parametric Menu** will be appeared. User can select main span, section, roof shape, support type, frame distance, roof tile type, ceiling type, live load, wind load, point load, section type, etc.



Clik **[OK]** 4 times to get:



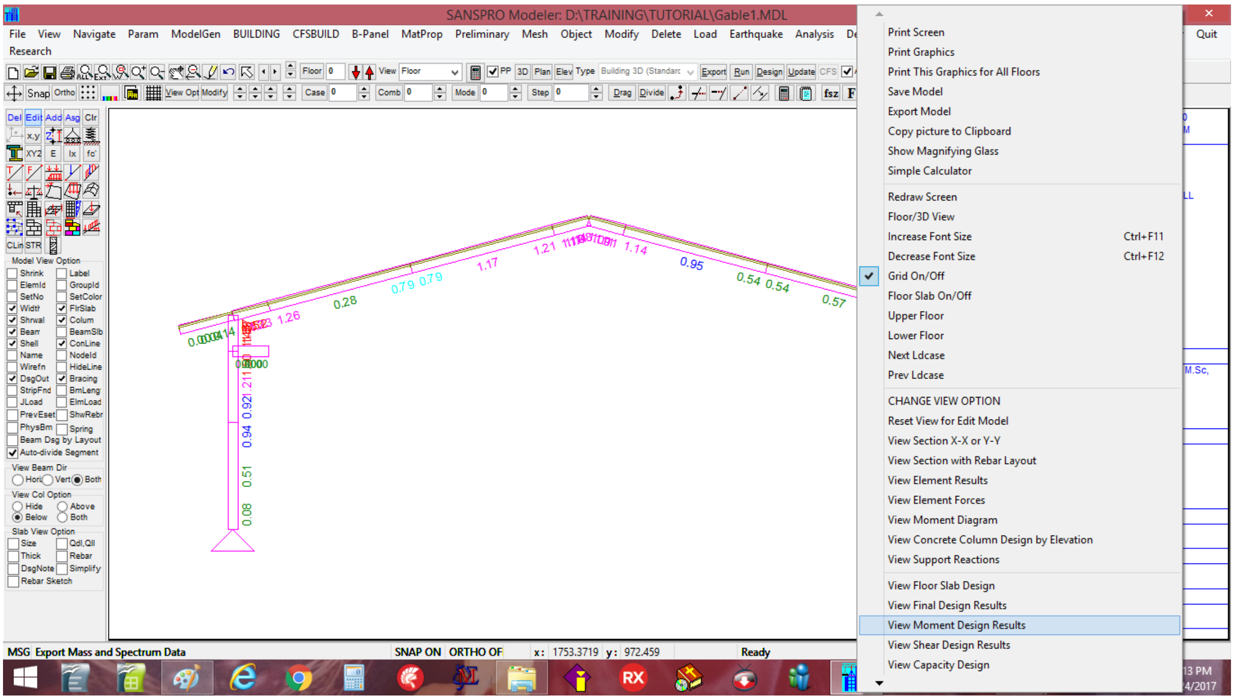
4. Click **F2** (save) and give new name : Gable1.MDL
5. Click **F4** (export) and Run Analysis
6. Click **Design - Steel/Frame/Building Design**, Select **Steel Frame**
7. View Moment Diagram by **Right-Click**, **View Moment Diagram**



NOTE:

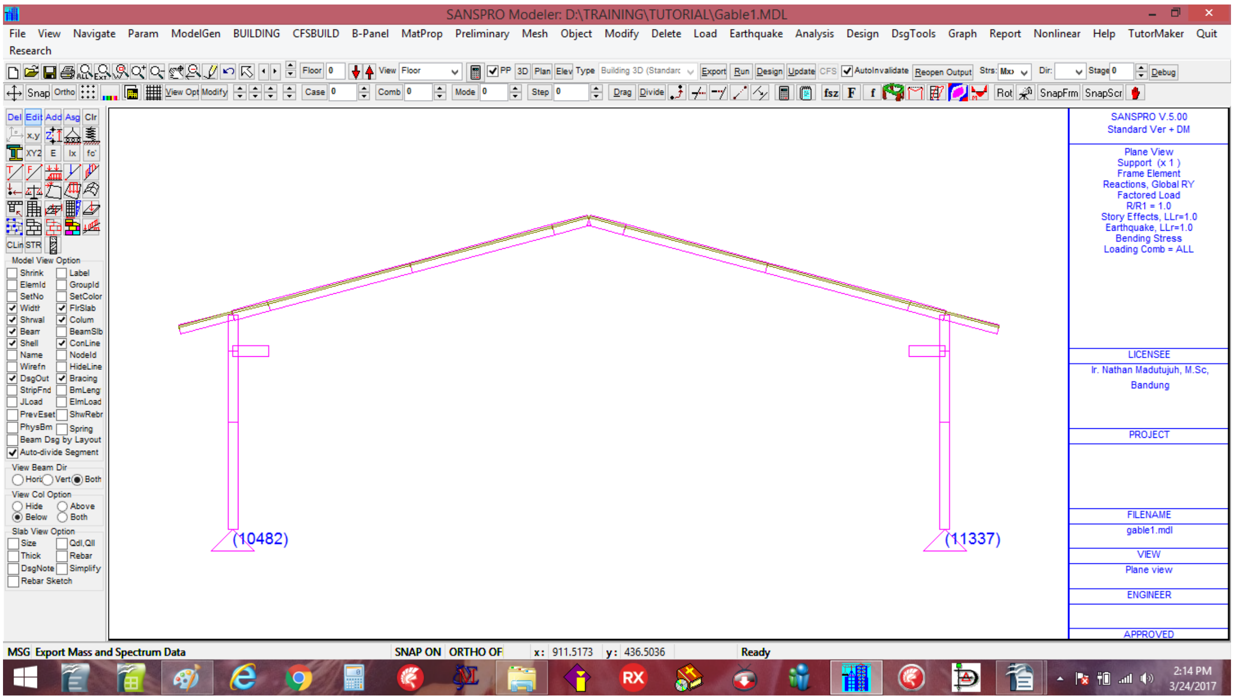
- If **load comb = 0** selected, an envelope of moment diagram will be displayed.

8. View Steel Design Results by **Right-Click**, **View Moment Design Results**



Steel frame stress ratio will be appear. Design is OK if the ratio is **fr <= 1.0** (recommended <= 0.8-0.9).

9. View Support Reactions by **Right-Click**, Select **View Support Reactions**



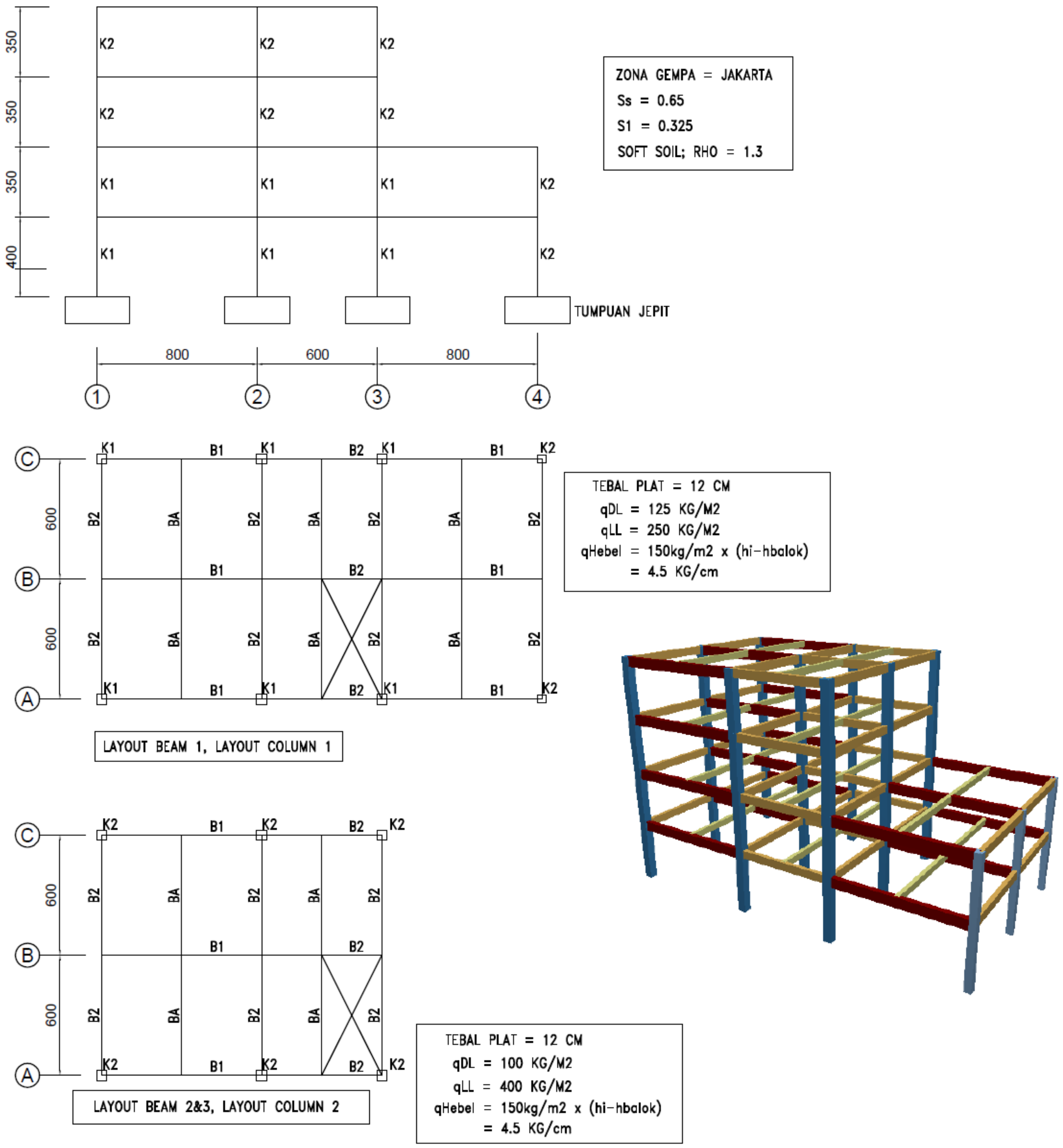
NOTE:

- The procedure for **Save, Export, Analysis**, and **View Support Reactions** are same for all models, so it will not be repeated again in this tutorial
- The program will automatically save a backup file every t minutes (if asked) or after every SAVE command
- **Design Code** and **Load factors** can be changed by user before analysis
- After a model has been created, user can repeat analysis and design anytime
- Various more complex truss system can be generated by selecting menu : **Roof Truss Parametric**

8. Basic Building Design:

1. Example 3: Building Design (Gravity and Static Equivalent Load Analysis)

In this very simple building design example, we will use a very simple 4 story building model, with two floor layouts, two type of slab thicknesses, two column layouts and two beam layouts.



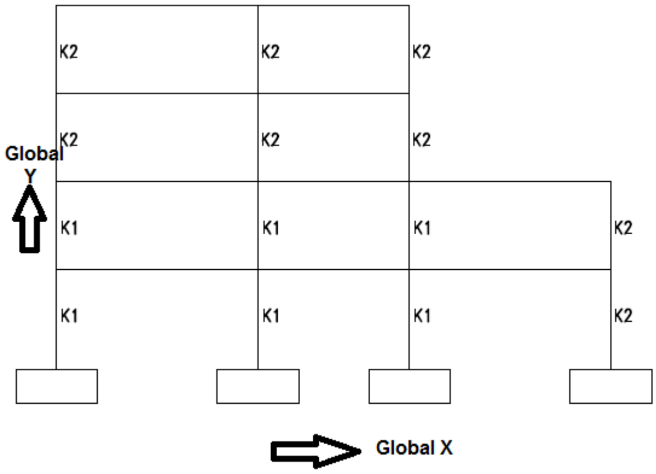
KETERANGAN :		
MUTU BETON = K-300 / fc'=25MPa	REDUKSI LIVE LOAD = 0.25	UNIT COST :
MUTU BAJA = U-39	MASS FACTOR = 0.25	CONCRETE = RP 800.000 /M3
B1 = 30/60	PONDASI = PILE 25/25, L = 18 M	STEEL = RP 7.000 /KG
B2 = 25/50	AXIAL CAPACITY = 40 TON	FORMWORK = RP 100.000 /M2
BA = 25/40	TENSION CAPACITY = 20 TON	
K1 = 50/50	LATERAL CAPACITY = 2 TON	
K2 = 40/40	PILE SPACING = 3D	

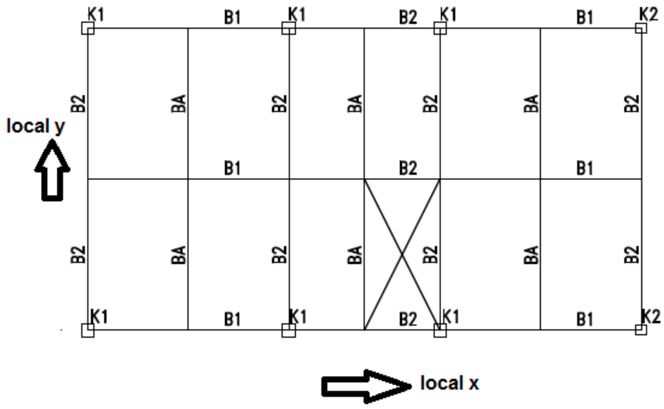
NOTE : Units are in kg, cm unless stated different.

Introduction to Building Layout Oriented Modelling Concepts

SANSPRO uses a unique approach for modelling a building, called **"Building Layout Oriented Modelling"**. Here a typical building floors is divided into several layout, analogue to building layout drawing. Floors with same layout will share same floor layout model.

SANSPRO uses terms : Beams and slab layout, Column layout, and Shearwall layout to allow combination of several layout in a floor. A Master table called Building Story Data Table will store all layout used in all floors. A layout can have more than one type and size of section. Slab layout will follow automatically a beam layout.





For sample building that we will use for exercise in this chapter, we can find out that there are at least 2 column layout needed, and also 2 beam layout needed. If the loading at roof level is different than at typical floor than 3 or 4 beam layouts will be needed.

Building Story Data

Floor	Column Layout	Beam Layout	Shearwall Layout
0	0	0	0
1	1	1	0
2	1	1	0
3	2	2	0
4	2	2	0

Slab Type	qDL	qLL
0	0	0
1	125	250
1	125	250
1	125	250
2	100	400
	kg/m2	kg/m2

Floor	Column Layout	Column Size
0	0	0
1	1	K1, K2
2	1	K1, K2
3	2	K2
4	2	K2

Floor	Beam Layout	Beam Size
0	0	0
1	1	B1, B2, BA
2	1	B1, B2, BA
3	2	B1, B2, BA
4	2	B1, B2, BA

Visible Column

User will work on top view drawing of a floor layout. A column shown on the drawing can be selected as a column below this floor level, or on/above this floor level, or both below and above columns shown.

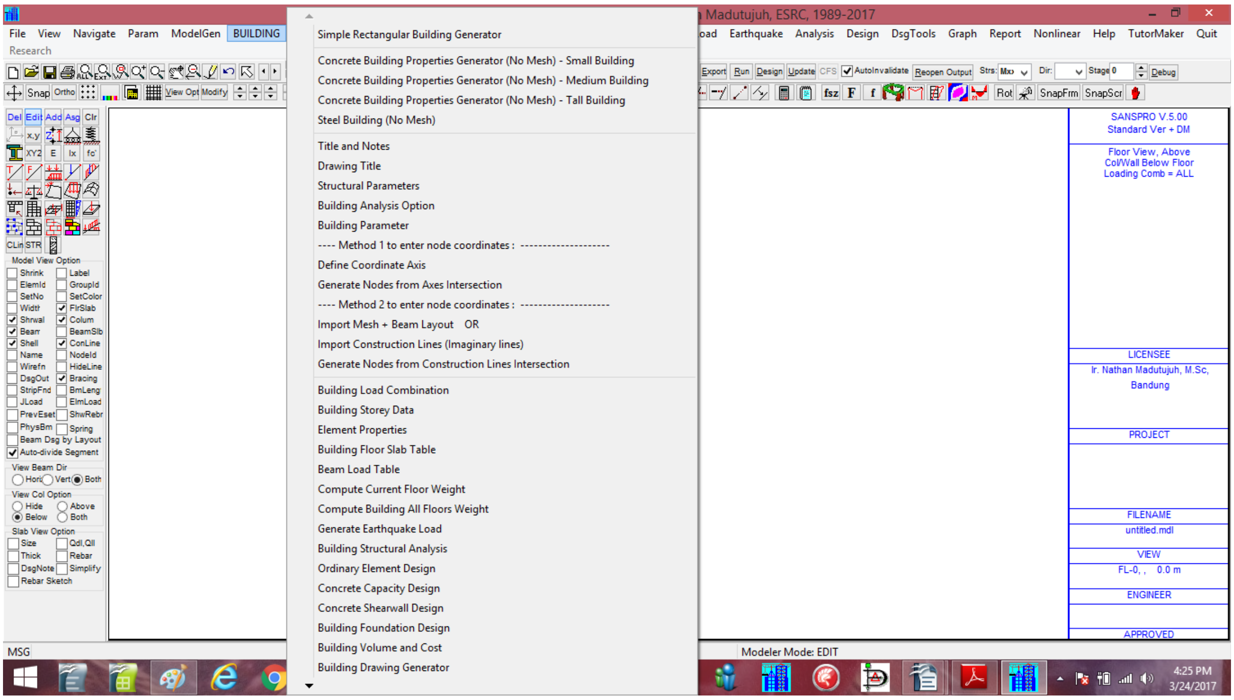
Node Coordinates and Axis

A Floor layout will need a set of node coordinates (2 dimensional), defined on the floor top view, as local x,y coordinates. SANSPRO program will use this floor coordinates and floor height/level to generate 3D node coordinates in global direction (X,Y,Z), where : X = x, Y = floor level, Z = -y.

Procedure for Creating a new Building Model:

1. Generate Building Material and Properties Wizard
2. Generate Project Data, Drawing Title
3. Generate Structural Parameters
4. Generate Analysis Option
5. Generate Building Parameters
6. Generate Node Coordinates (using Coordinate Axis, Construction Lines, or one by one)
7. Generate Load Combinations
8. Generate Story Data
9. Generate Element Properties
10. Generate Floor Slab Table
11. Generate Beam Load Table
12. Compute Floor Weight
13. Generate Earthquake Load
14. Continue with Analysis and Design

All steps above are already provided at a certain menu called **BUILDING**. This menu collect all necessary commands needed to model and design a building from other SANSPRO menu.



After a building model created, user can do the following:

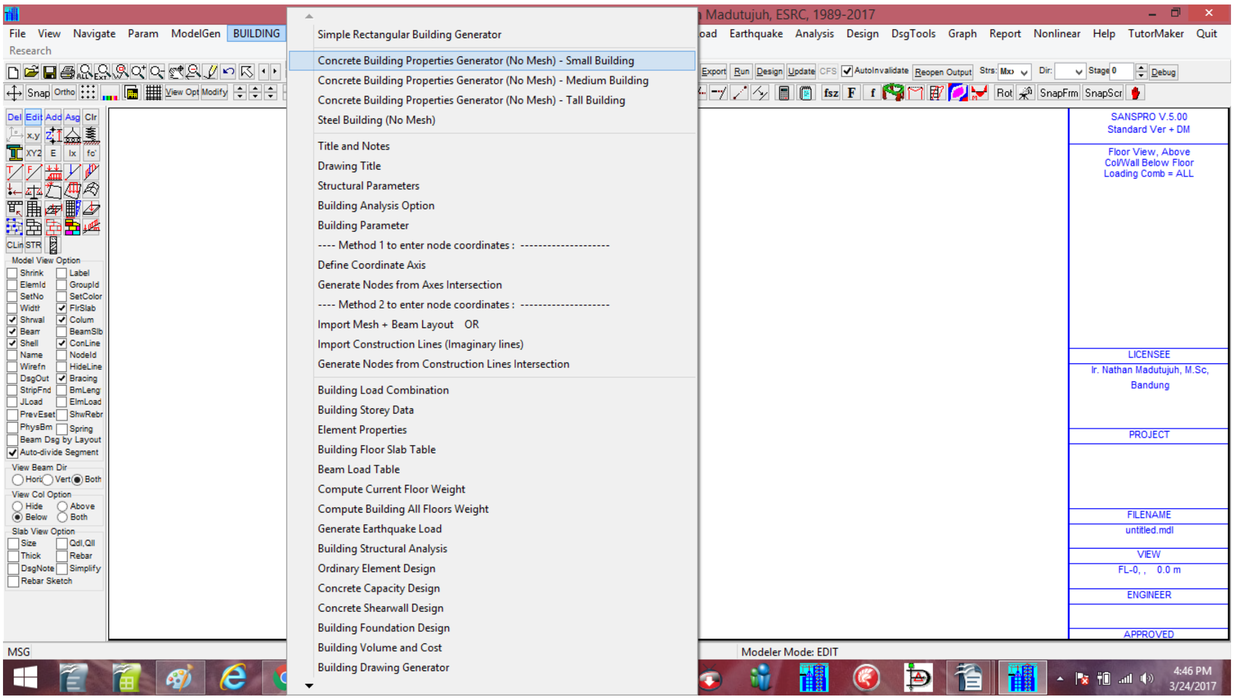
1. Run analysis
2. Run Section Design
3. Run Shearwall Design
4. Run Foundation Design
5. Run Volume and Cost Calculation
6. Generate Detail Drawing

The detail procedure for creating a building model is as follows.

Note that we will use this same building model for example 4 to 9 later.

1. Generate Building Material and Properties Wizard

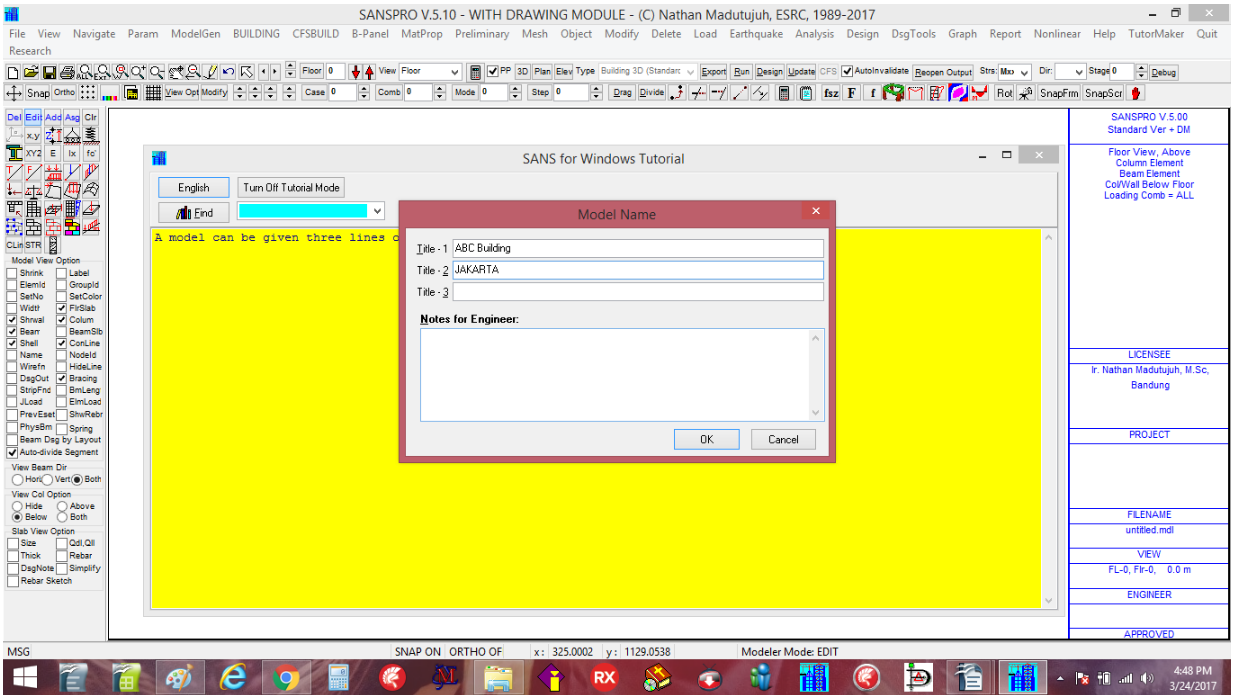
Click **Building – Concrete Building Properties Generator (No Mesh) – Small Building**



This command will generate several material properties and section properties useful for small buildings.

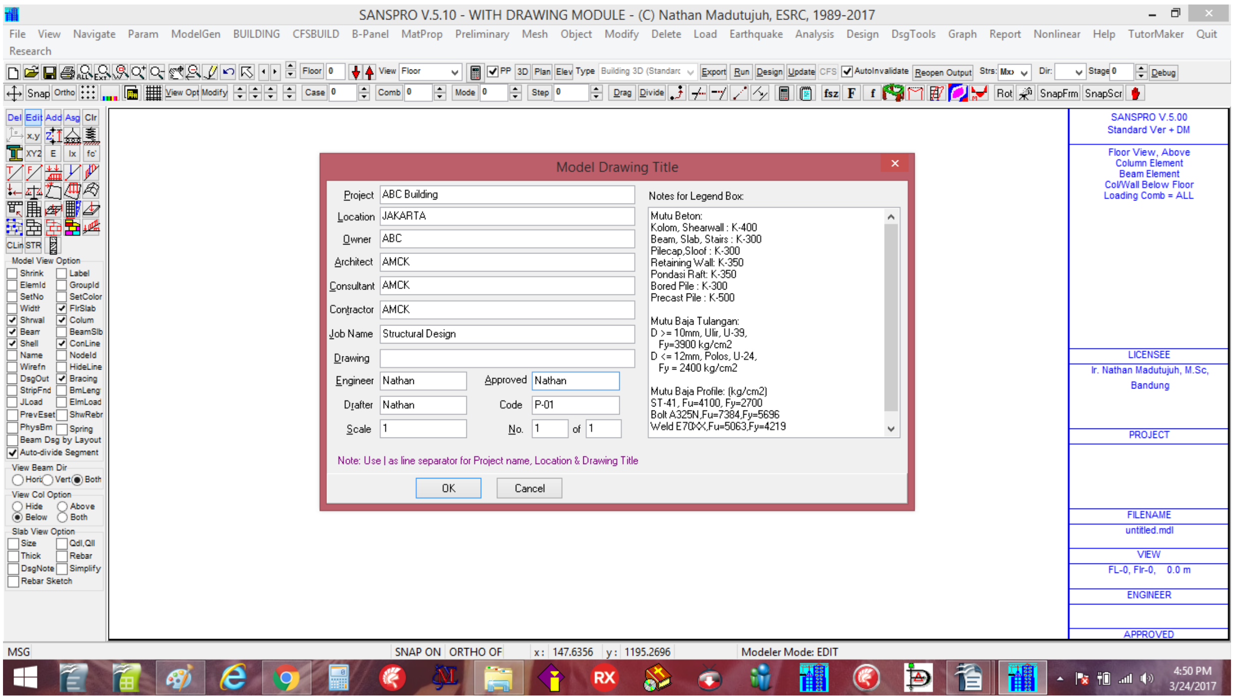
Click **[Yes]** 2x, Enter Total Floor number : **NST = 4**

2. Click **Building – Title and Notes**, enter the following data



Then Click **Building – Drawing Title**, enter data for drawing text

Please replace my name with your name !

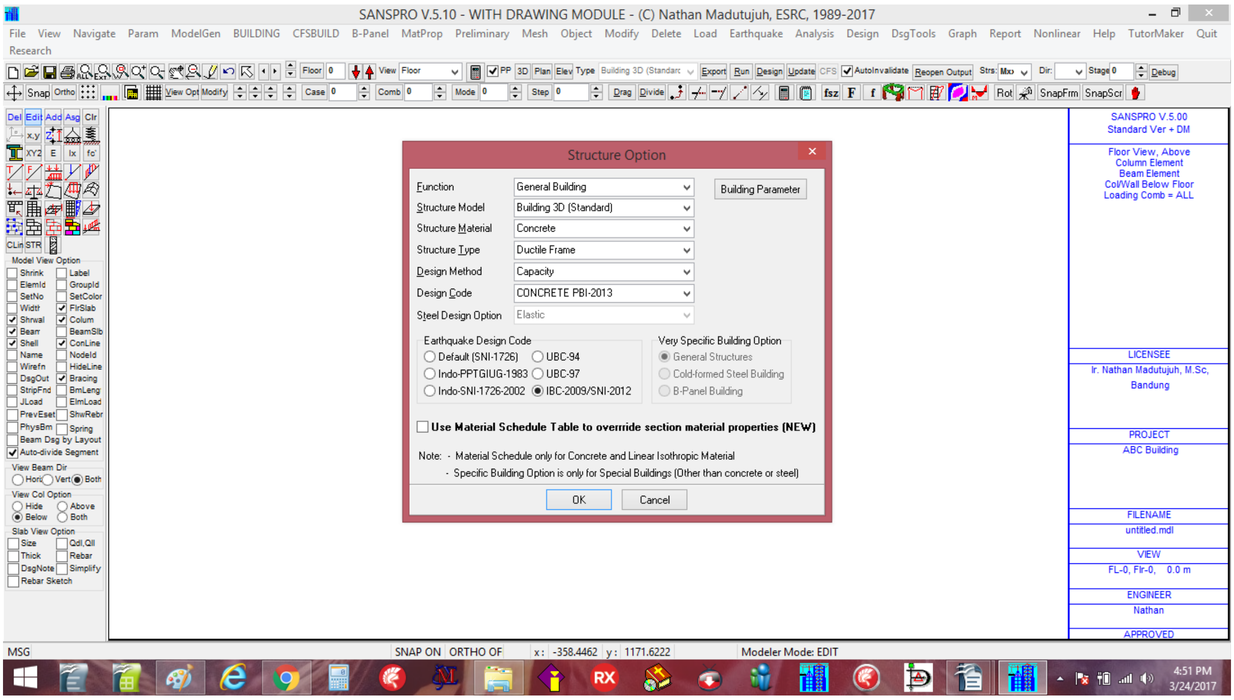


3. Generate Structural Parameters

Click **Building – Structural Parameters**

Select Design Code : CONCRETE PBI-2013

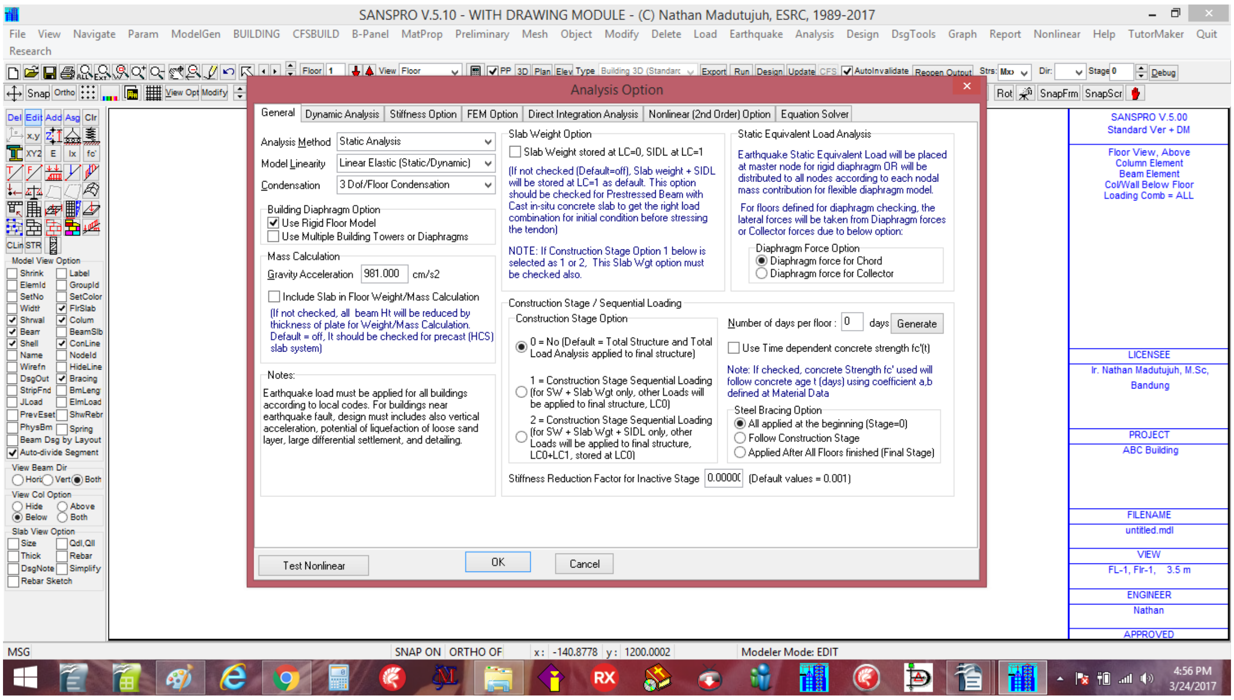
Select Earthquake Design Code: IBC-2009/SNI-2012



NOTE:

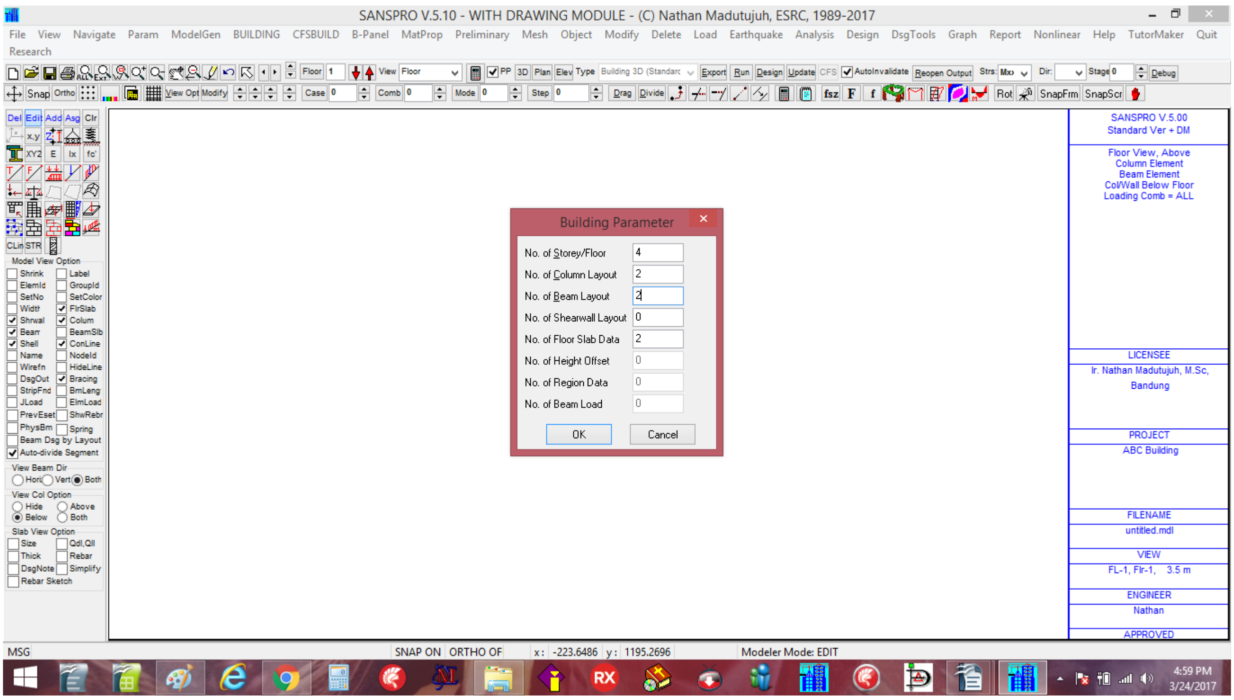
Option [ ] Use Material Schedule Table will be useful for highrise building if the concrete strength varies due to floor height.

4. Click **Building - Analysis Option**



For this exercise using Static Analysis, nothing changes, just use default settings, then click **[OK]**

5. Click Building - Building Parameters



See "**Introduction to Building Layout Oriented Modelling Concepts" above:**

Enter No. of Column Layout = 2
Enter No. of Beam Layout = 2
Click **[OK]**

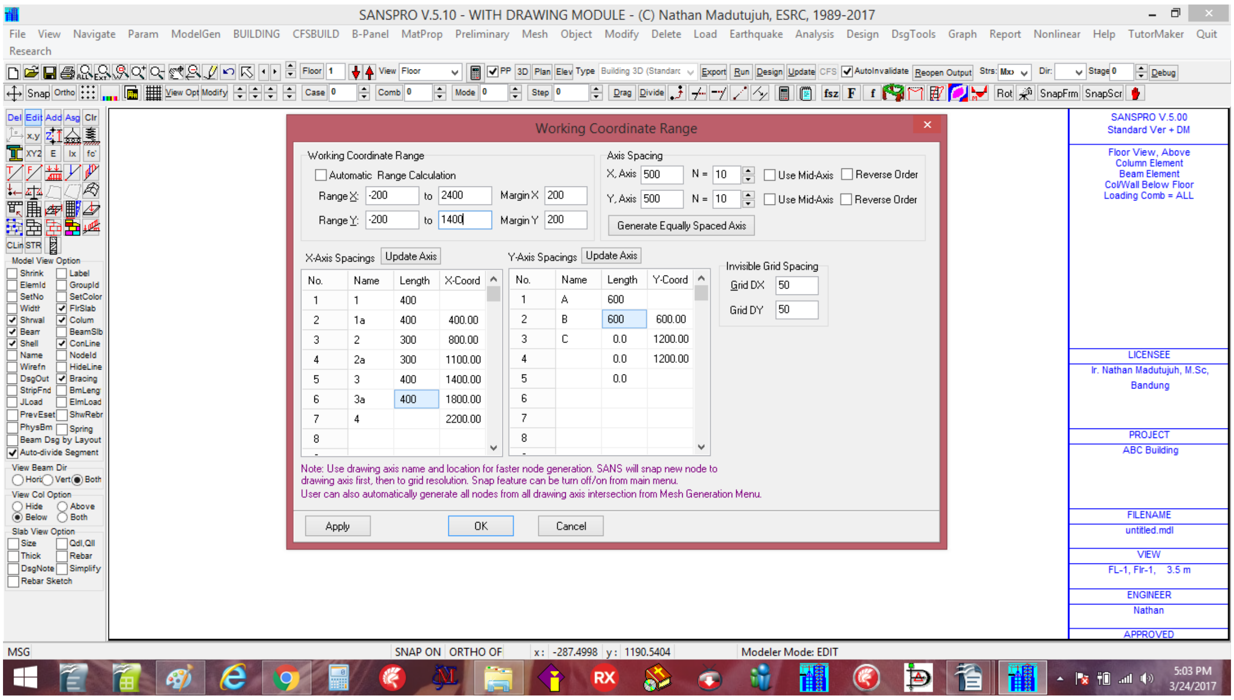
6. Generate Node Coordinates (using Coordinate Axis, Construction Lines, or one by one)

Here we will use coordinate axis approach.

x-axis : 1, 1a, 2, 2a, 3, 3a, 4 with spacing : 400,400,300,300,400,400 cm
y-axis : A, B, C with spacing : 600,600 cm

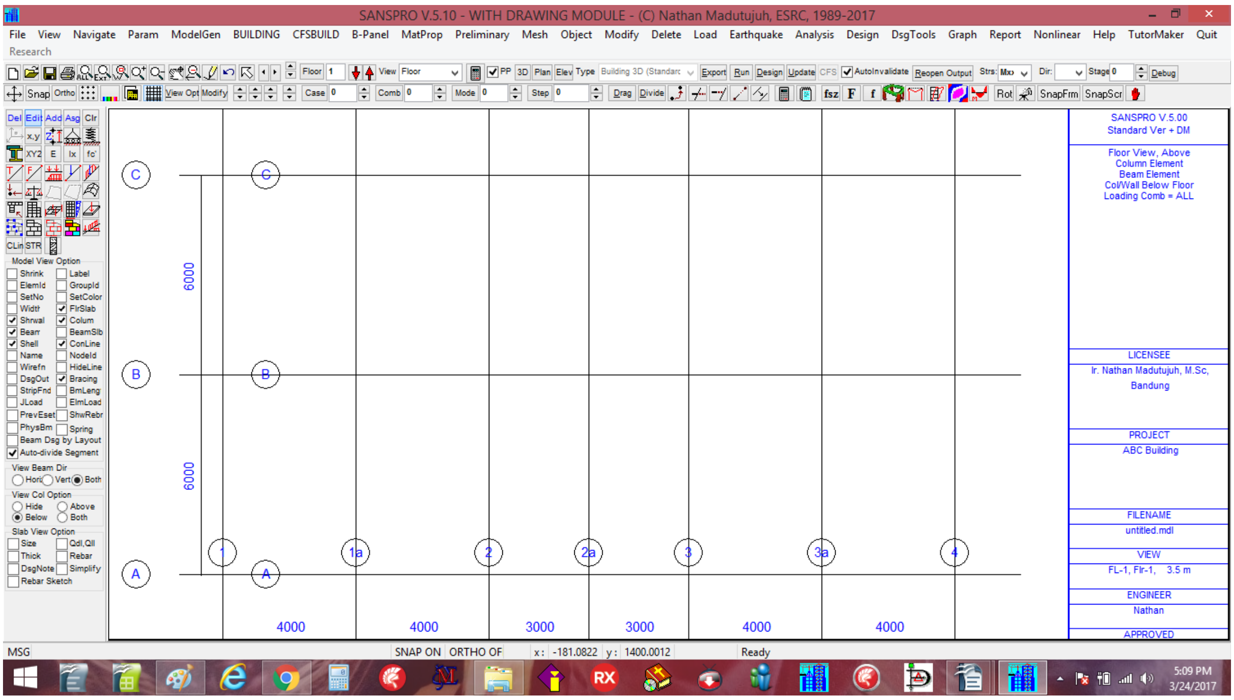
Click **Building – Define Coordinate Axis, enter** Name and Length as follows:
(NOTE : **X-coord** and **Y-coord data** will be generated automatically, no need to enter it manually)

Remember to change **Range - X** to 2400 cm and **Range – Y** to 1400 cm (from Xmax + 200, Ymax + 200) to get a nice working space.



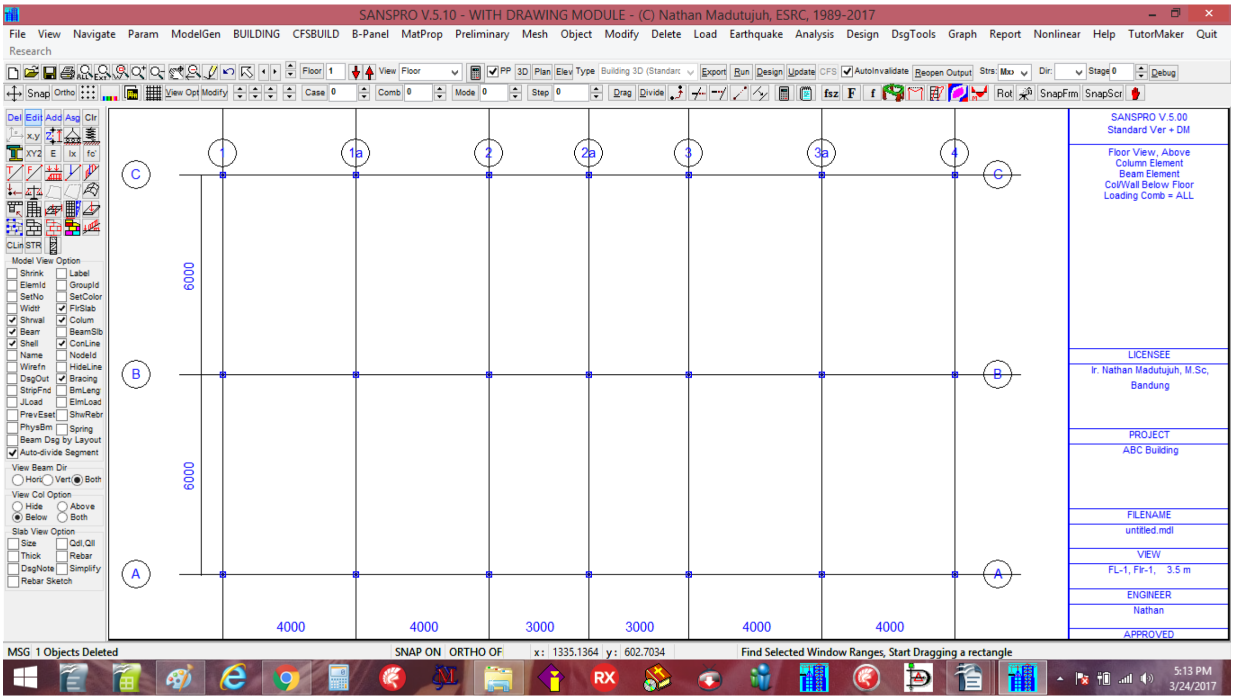
Click **[Apply]** and **[OK]** : Coordinate Axis will be generated automatically

Click **icon [||||]** to Show Axis lines:



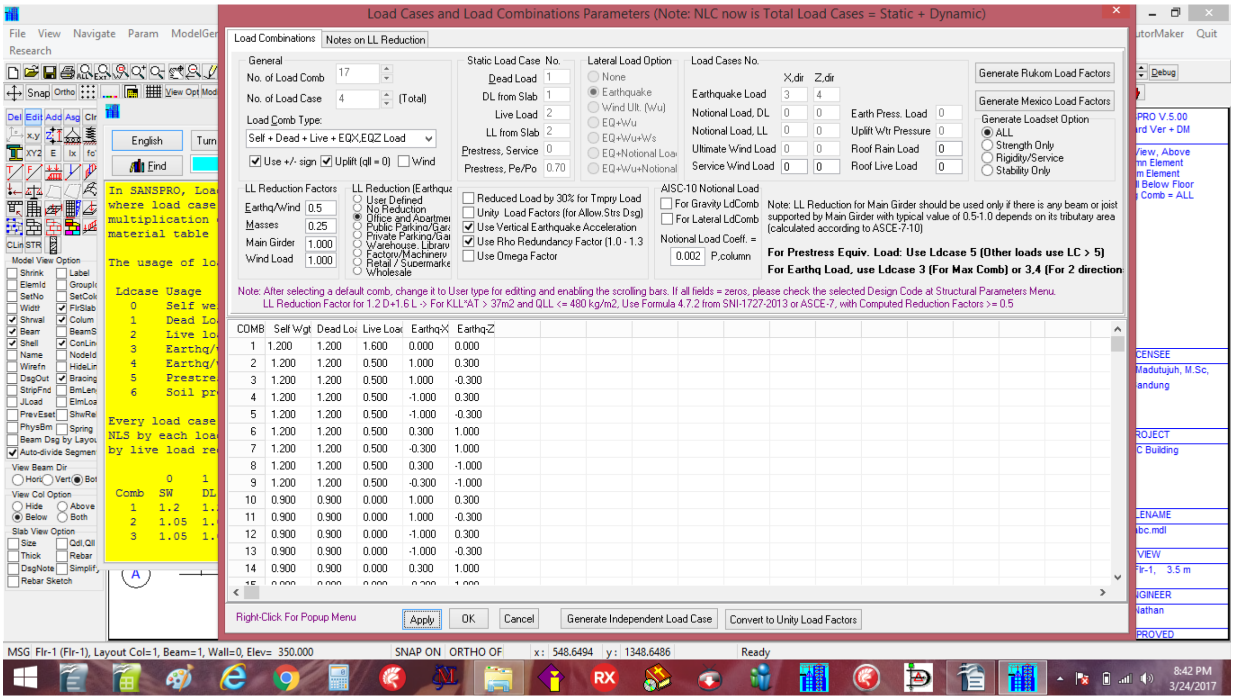
Node coordinates can be generated by clicking : **Building – Generate Nodes from Axis Intersection**
Click **[Yes]**

Node coordinates will be generated from Axis Intersection as follows:



7. Generate Load Combinations

Several Pre-defined Load Combinations are available. After selecting a predefined load comb, user can change it later by changing the type to user-defined load comb.



Building Function : Office

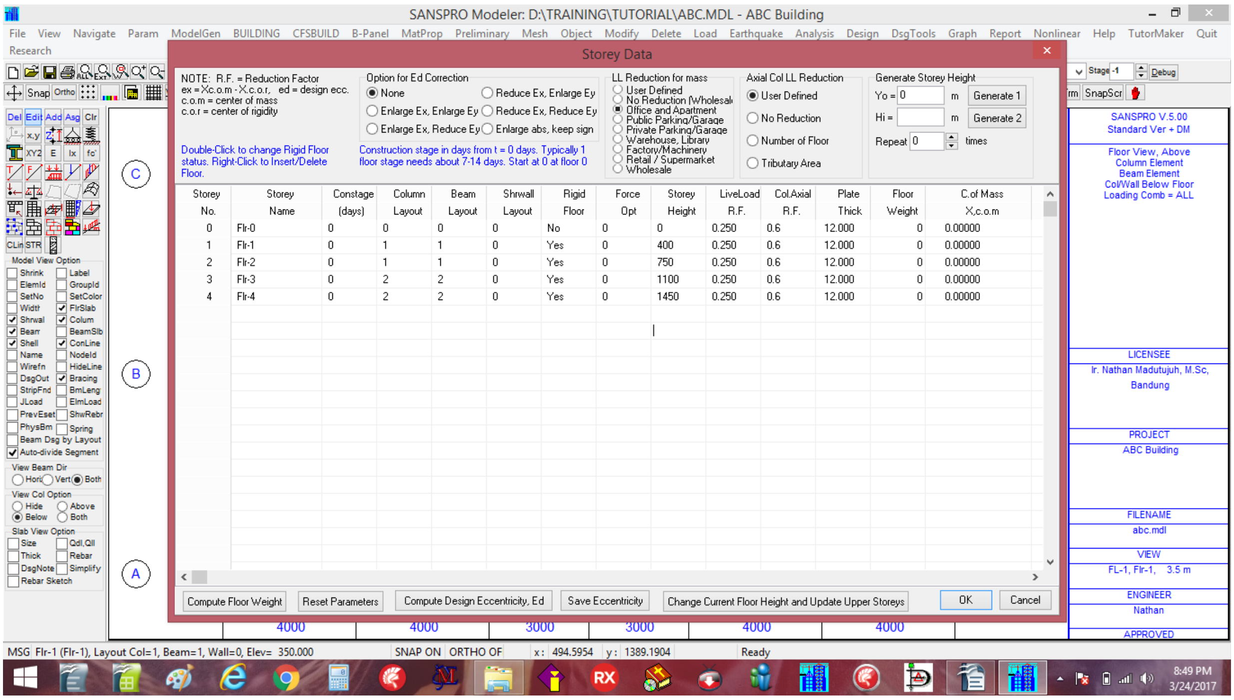
Select Load Comb Type : Self + Dead + Live + EQX + EQZ Load

Live Load Reduction Factor for Mass calculation : 0.25
Live Load Reduction Factor for EQ Load Comb : 0.50

Live Load Reduction Factor due to Area Contribution : 0.60 – 1.0 (depends on elements, distance)

8. Generate Story Data

We will now define Building Story data from the given building layout and section as follows :



Set LL R.F. (Reduction Factor) = 0.25 (Click Office and Apartment)
Set Column Axial R.F. = 0.60 (from calculation of KLL, see attachment)

Calculation of LL R.F. :

$$L = L_o\left(0{,}25 + \frac{4{,}57}{\sqrt{K_{LL} A_T}}\right)$$

<u>Where:</u>

L = Reduced Live Load
Lo = Unreduced Live Load
At = Tributary Area in m2
kLL = Coefficient (see table below)

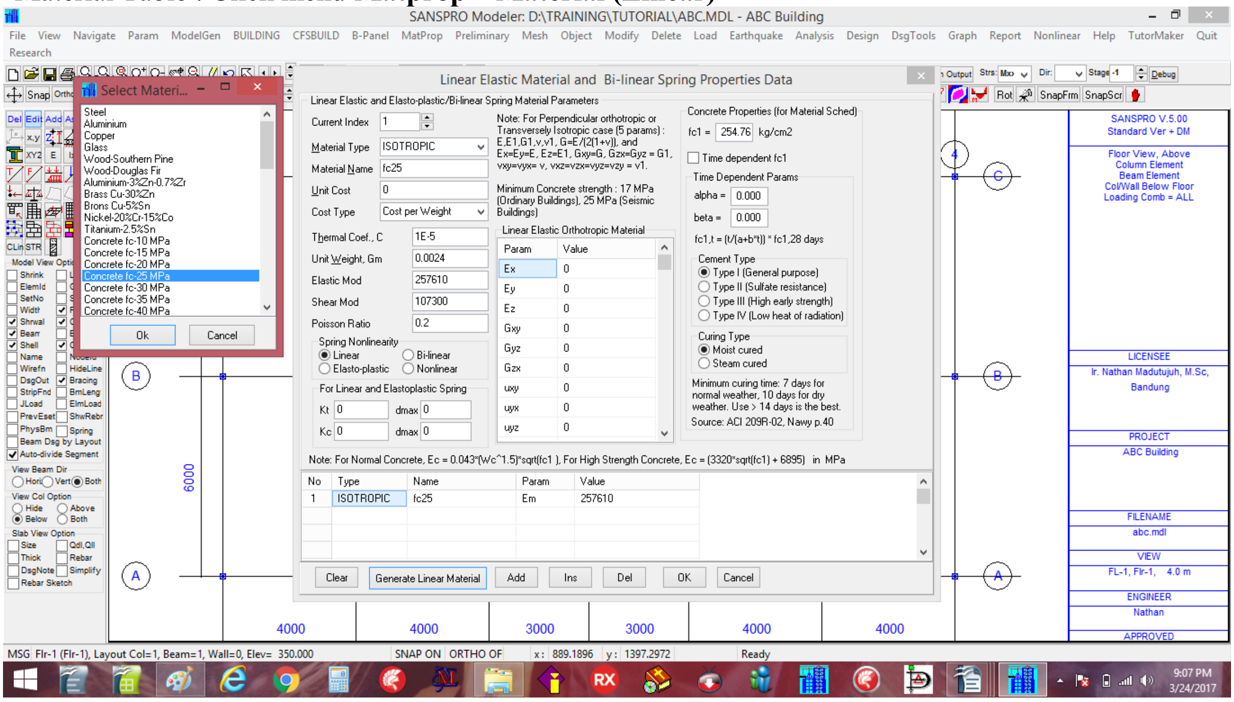
Location	kLL
Interior column	4
Edge column	3
Corner column	2
Edge, interior Beam	2
Cantilever beam	1
Floor Slab	1

9. Generate Element Properties

Element data set properties are defined into 4 tables as follows:

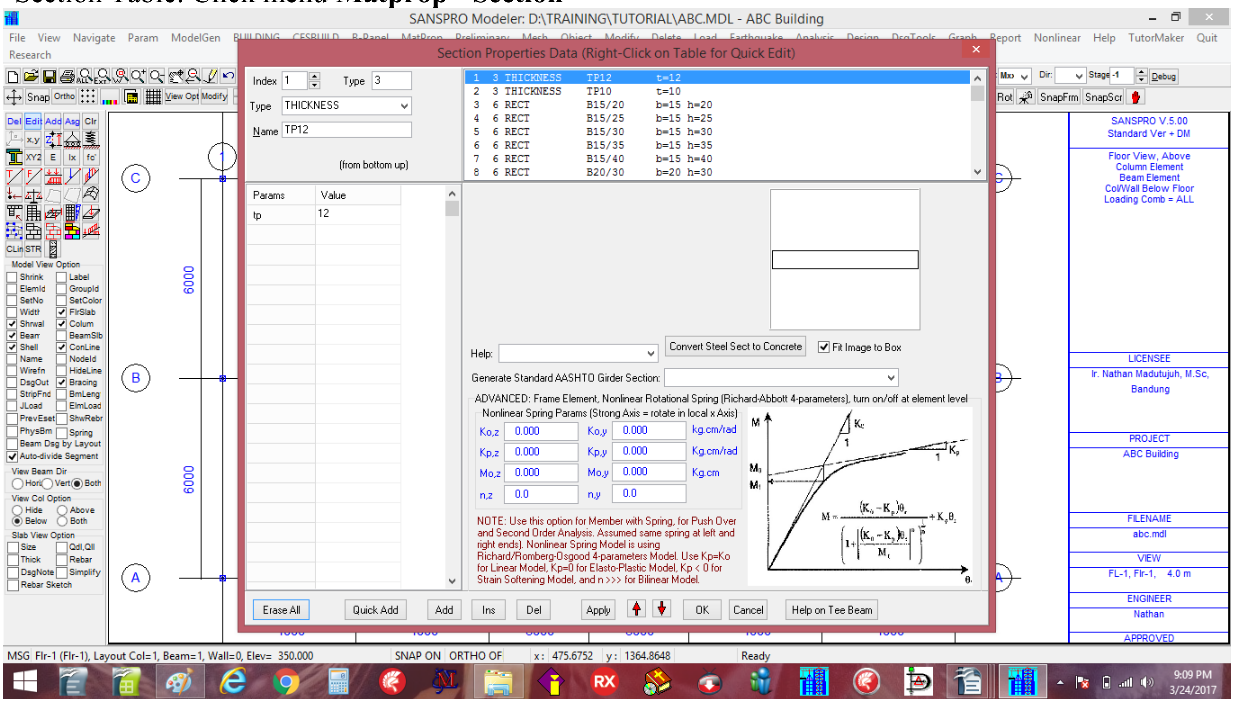
Material Table : Contains material data, E, unit weight, poissor ratio, etc.
Section Table : Contains section size, b, h, etc.
Design Table : Contains fc', fy, db, dbv, etc.
ELSET Table : Combining the above tables into one ELSET table

Material Table : Click menu **Matprop – Material (Linear)**



Click **[Generate Linear Material]** – Select **Concrete fc' 25 Mpa**

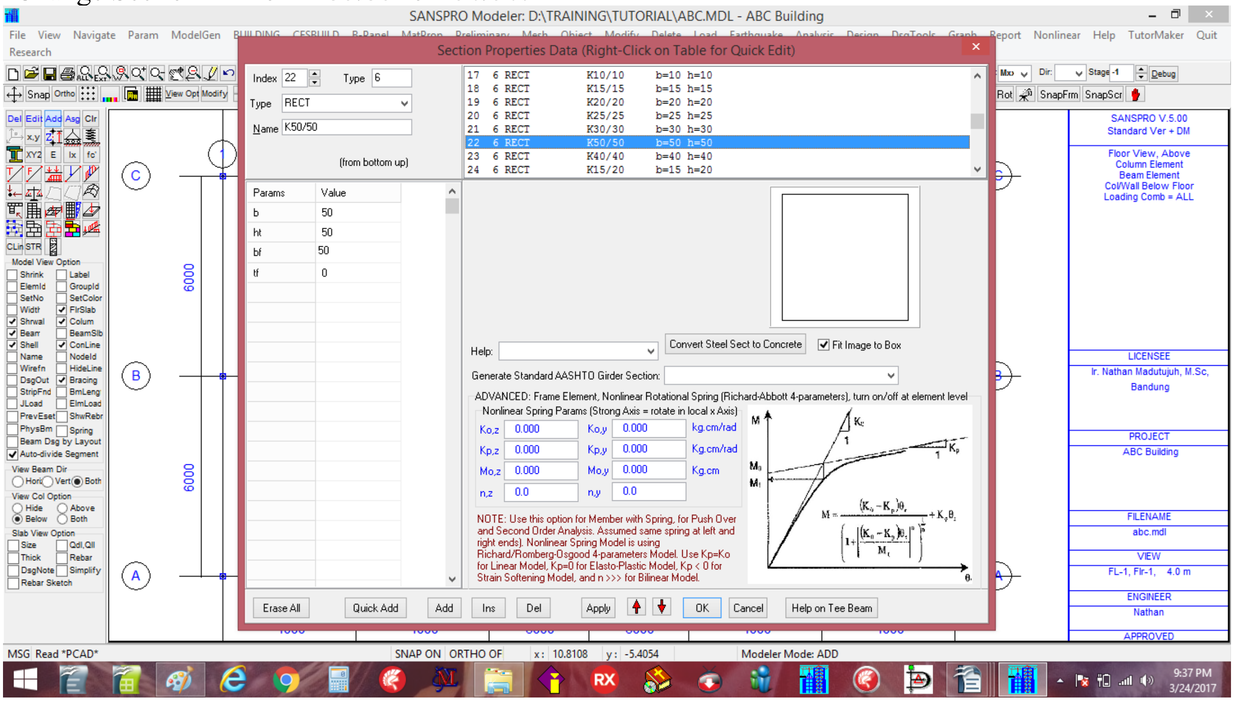
Section Table: Click menu **Matprop - Section**



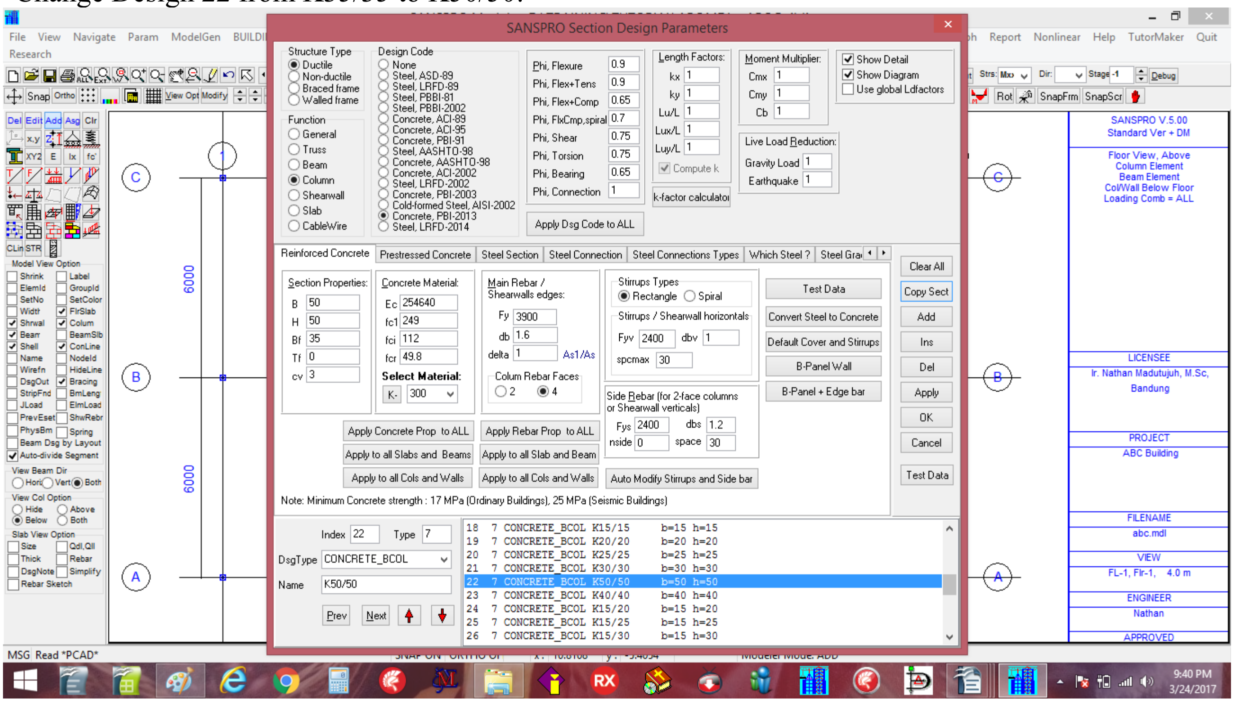
Section already generated by Wizard. Nothing to do.
This table can be added or deleted if necessary, but ELSET table must be generated afterward.

We need K1 = K50/50, but at the generated section table there is no K50/50, the maximum size is K40/40.
Please change Section Number 22 : from K35/35 to K50/50 for next step.

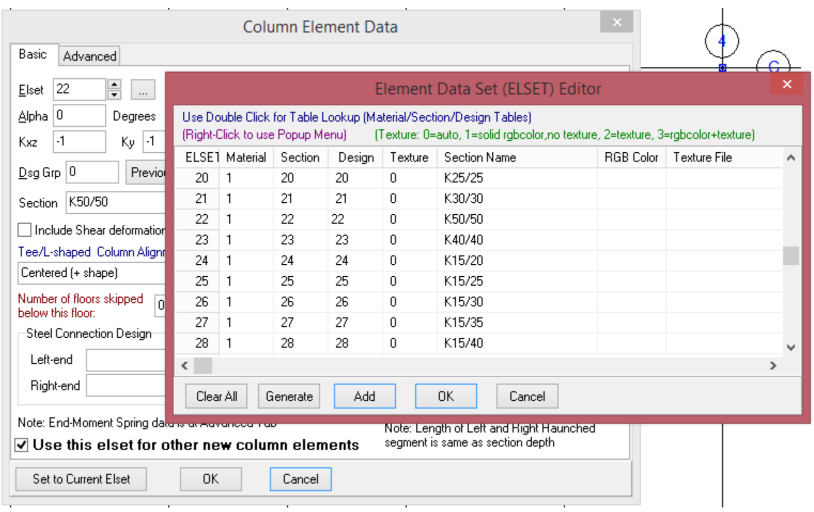
Change Section 22 from K35/35 to K50/50:



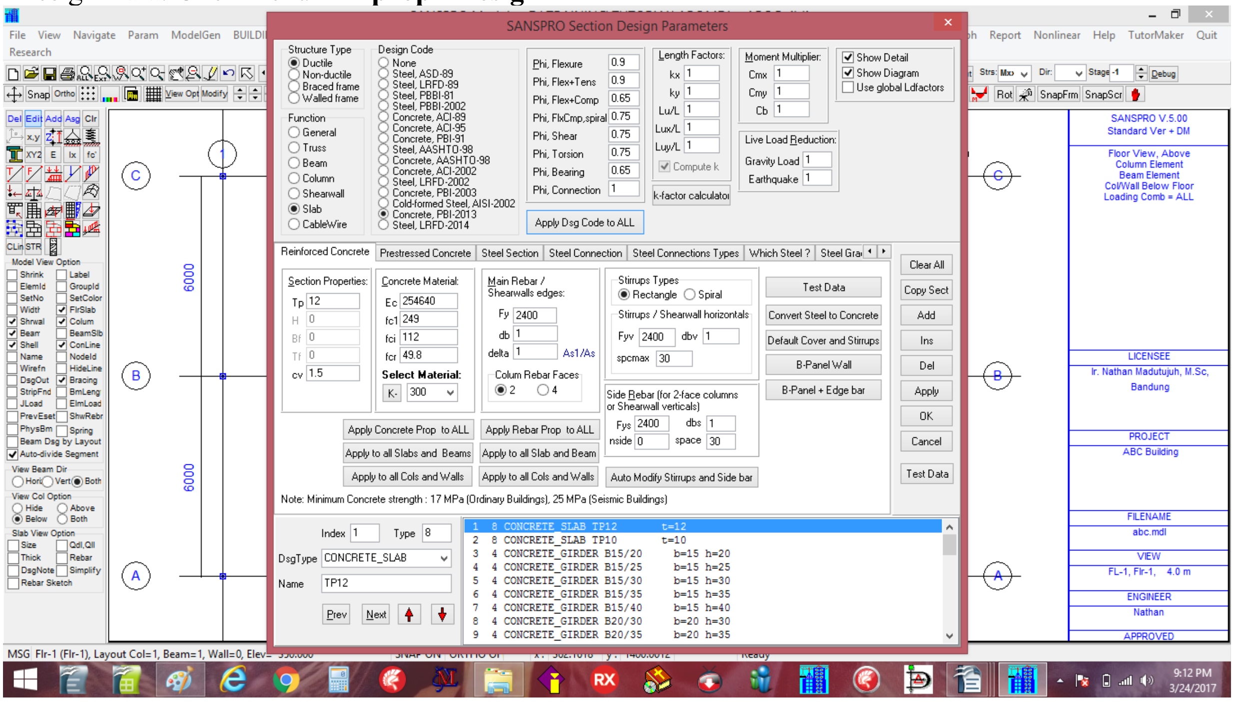
Change Design 22 from K35/35 to K50/50:



Elset table will automatically reflects this change:



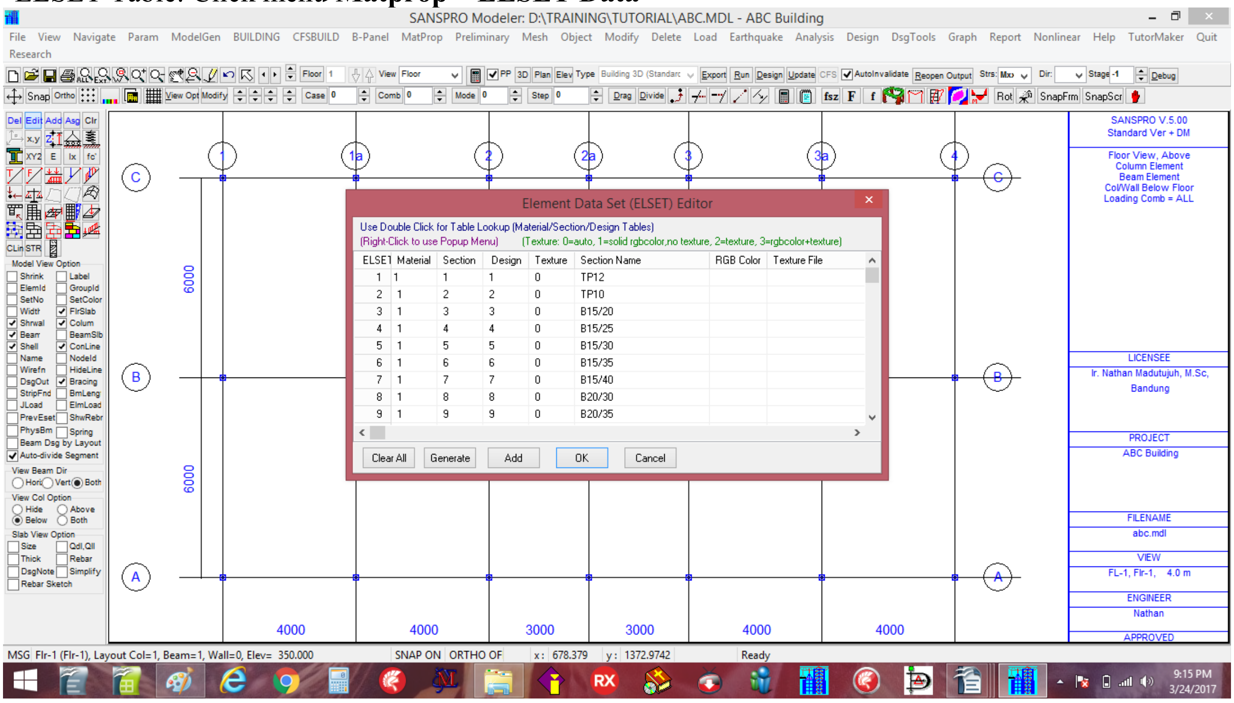
Design Data: Click menu **Matprop - Design**



Select Design Code : **Concrete PBI-2013**, Click **[Apply Dsg Code to ALL]**
Select Material : **K-300** (fc'25), Click **[Apply Concrete Properties to ALL]**

We need K1 = K50/50, but at the generated Design table there is no K50/50, the maximum size is K40/40. Please change Design Number 22 : from K35/35 to K50/50 for next step.

ELSET Table: Click menu **Matprop – ELSET Data**

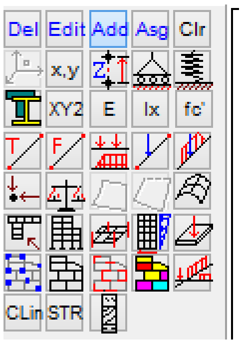


From this table it can be seen that all material index is 1 (using same material fc'25), while section and design index is from 1..41

This ELSET table will be used for selecting element data properties when we add any element.

10. Generate Column elements

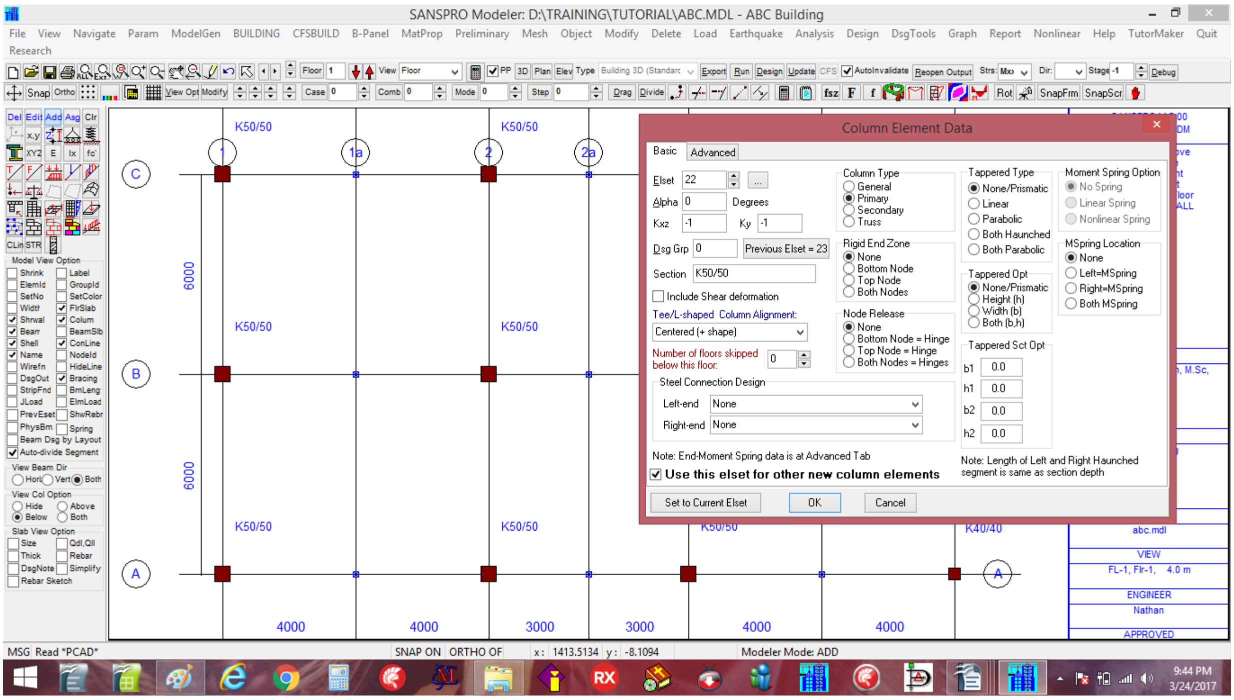
After we define all element properties and load combination, now we can define the structural elements. Starting from column elements as follows:



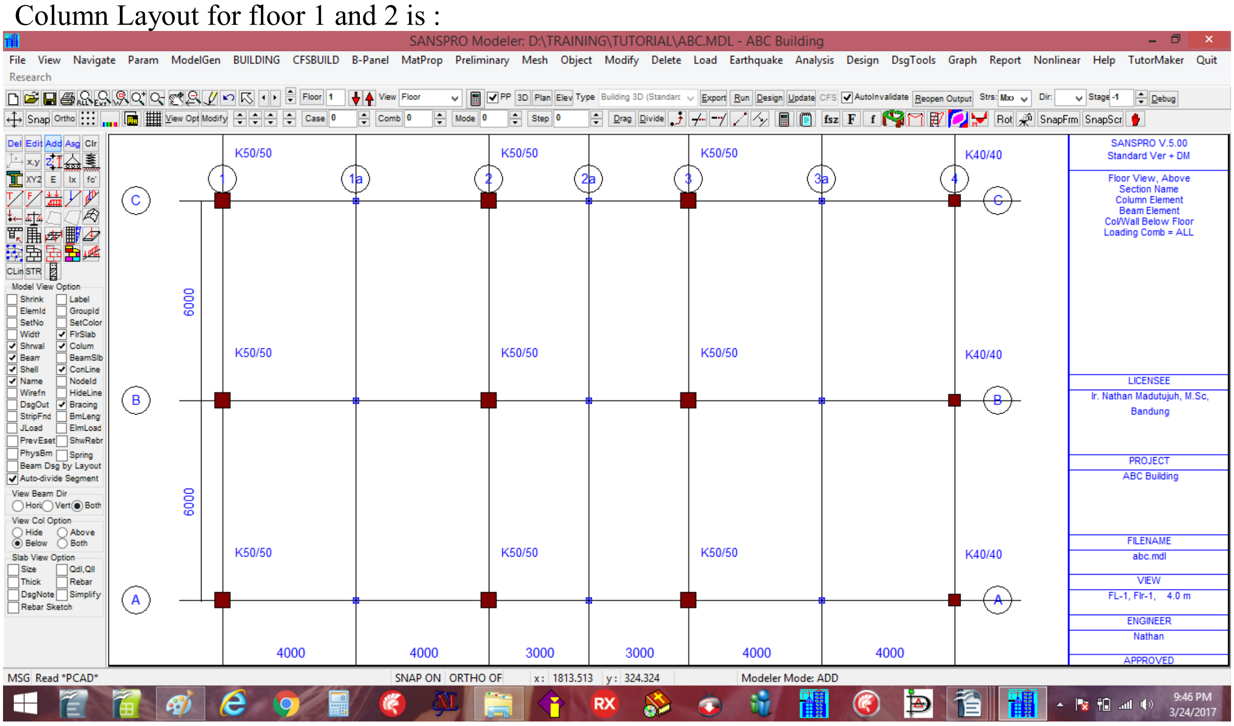
Select Floor No. 1 (column below this floor will be defined)
Click icon Click icon Add
Click any column position with K1
Select Elset 22 = K50/50, Click **[OK]** *(If you still have K35/35, see above)*
Click for other column position with K1 size

For Column K2:

Click icon Click icon Add
Click any column position with K2
Select Elset 23 = K40/40, Click **[OK]**
Click for other column position with K2 size

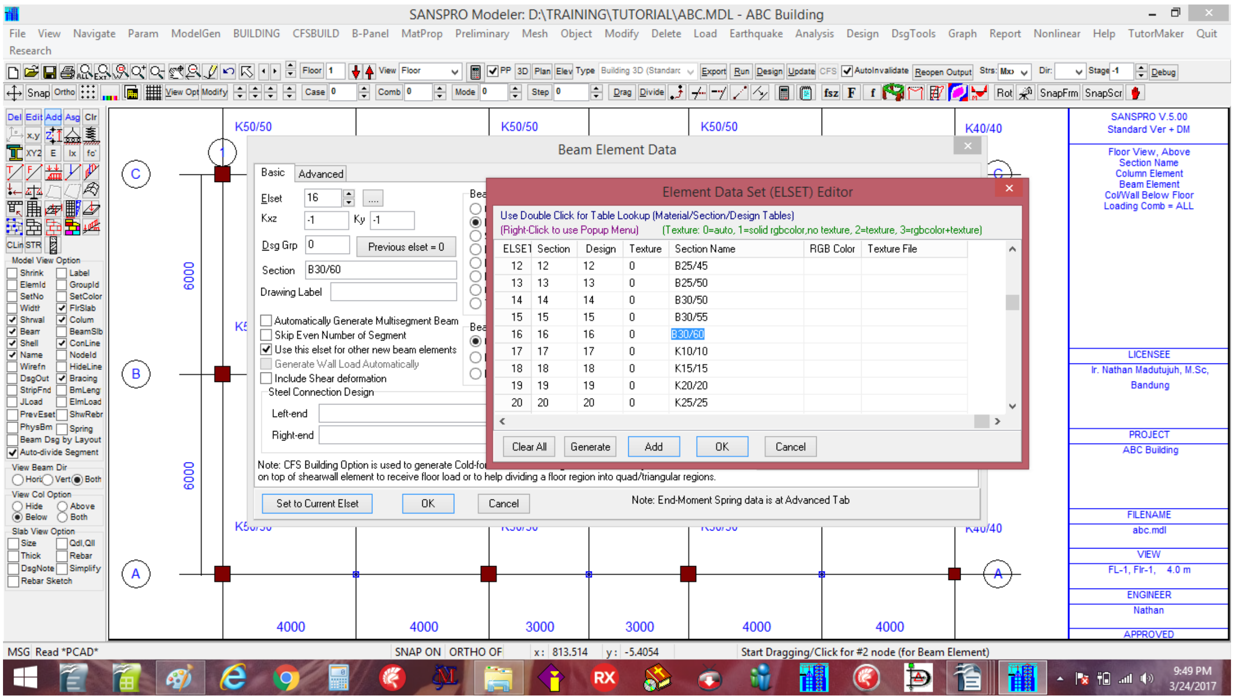


Column Layout for floor 1 and 2 is :

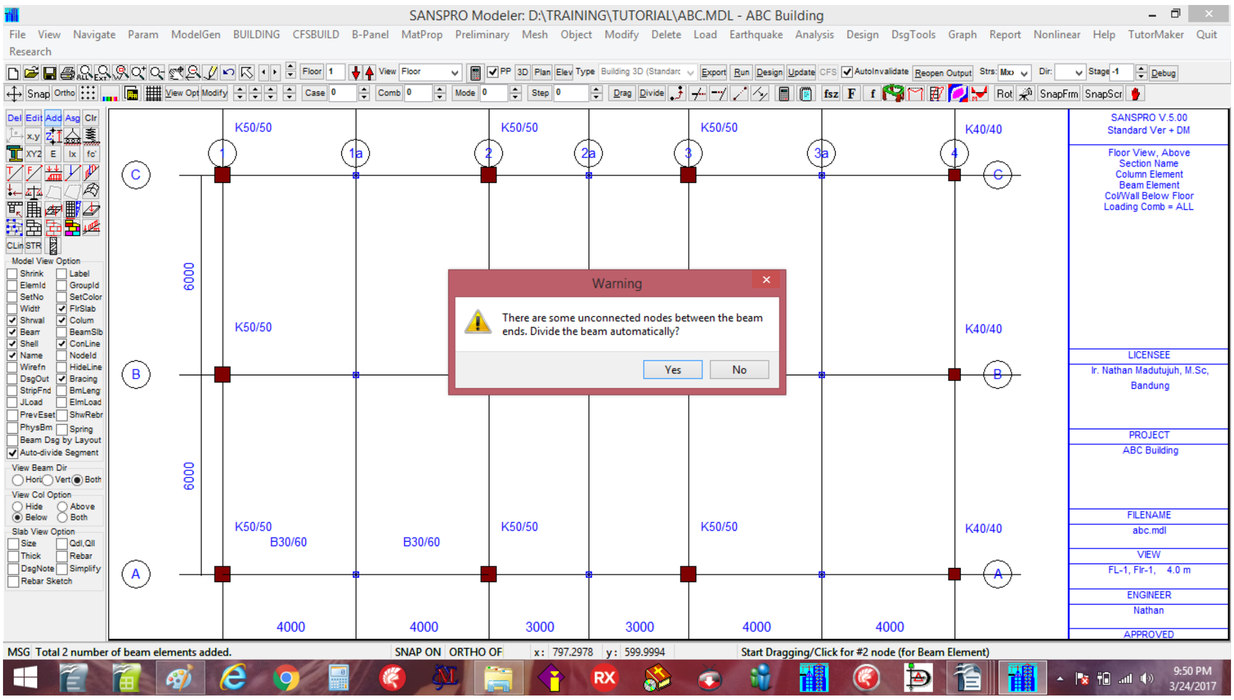


11. Generate Beam elements

Click icon Click icon Add
Click first point of any beam with size **B1 = 30/60**
Drag to second point of the beam along the line with same beam size
Select Elset 16 = B30/60, Click **[OK]**



(If the dragged line passing some points, answer with **[Yes]** for Auto-Divide beams)



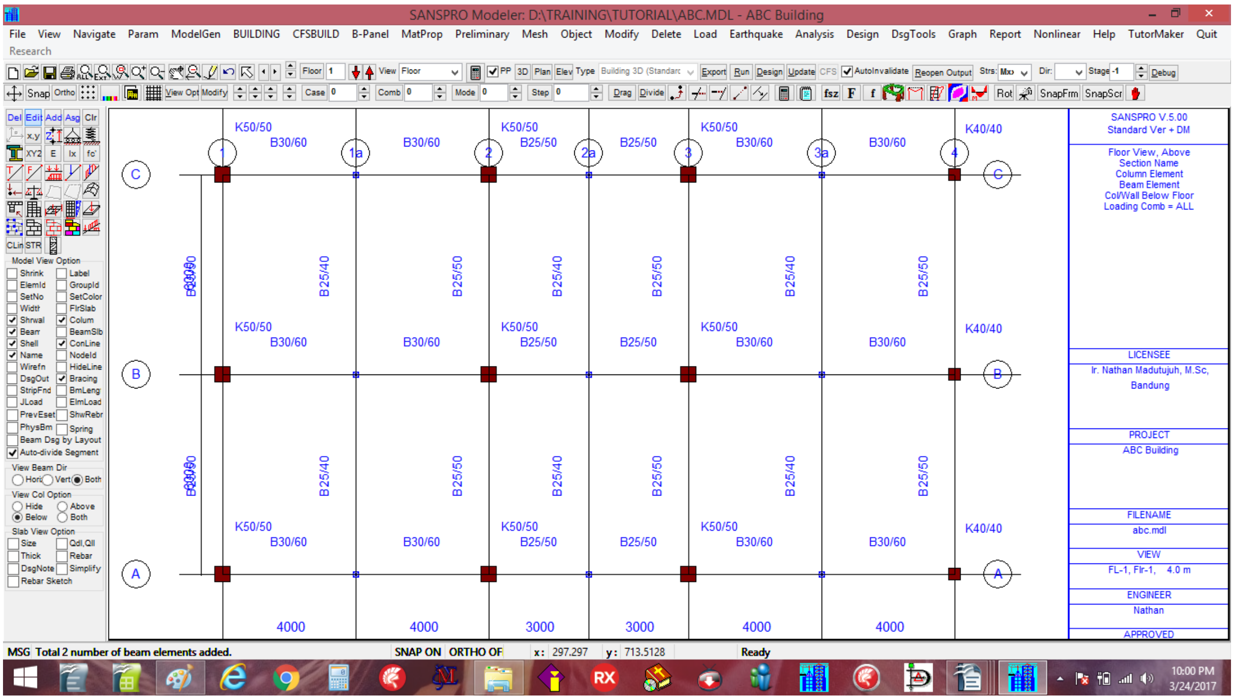
<u>**For other beam line with same size:**</u>

Click first point of any beam with size **B1 = 30/60**
Drag to second point of the beam along the line with same beam size
(If the dragged line passing some points, answer with **[Yes]** for Auto-Divide beams)

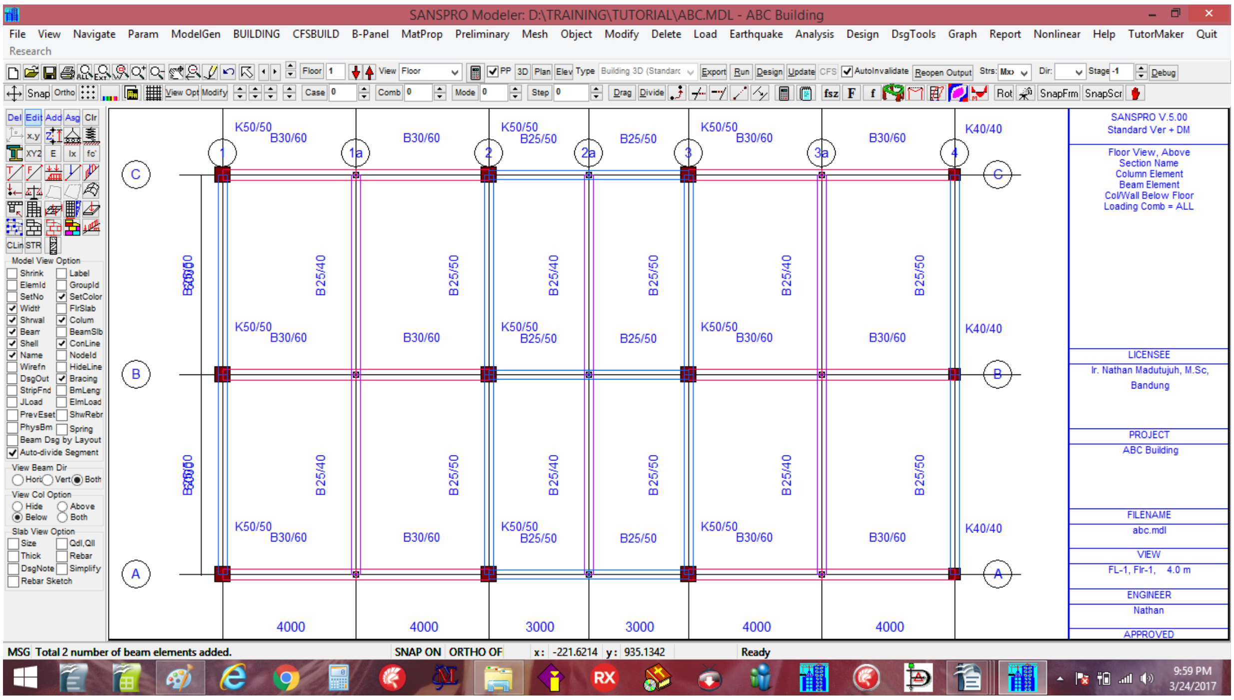
<u>**For other beam line with different size:**</u> **B25/50, B25/40**

Click icon Click icon Add
Repeat other steps (select Elset 11 for B25/40 and Elset 13 for B25/50)

After all beams has been generated, the floor layout will be :

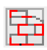


Click checkbox [x] Width to show beam size
Click checkbox [x] Set Color to show beam size in different color



Because of the layout oriented modelling, all columns and beams at floor 1 and 2 will be automatically defined accordingly. Please check it by moving from floor 1 to 2.
Any changes made to any element at floor 1 will be follow automatically by floor 2 and vice verse.

Note:

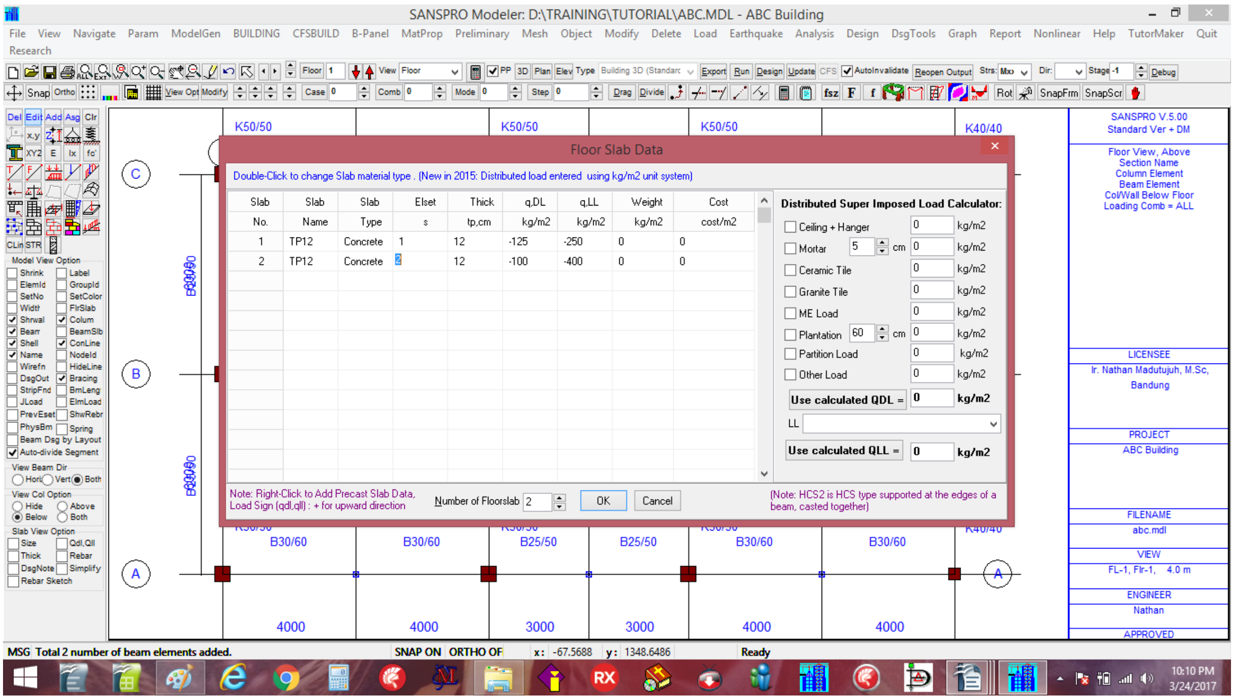
- For adding shearwall elements, the procedure is similar with beam elements exexcept using icon 
- Shearwall below the floor will be added

12. Generate Floor slab Table

After all beams defined, we can generate floor slab at this floor. But first we must define the floor slab table. Click Building – Building Floor Slab Table

Change TP10 Section and Design to be TP12. Change the rebar diameter from 0.8 to 1.0 cm, change fy = 2400 to 3900 kg/cm2

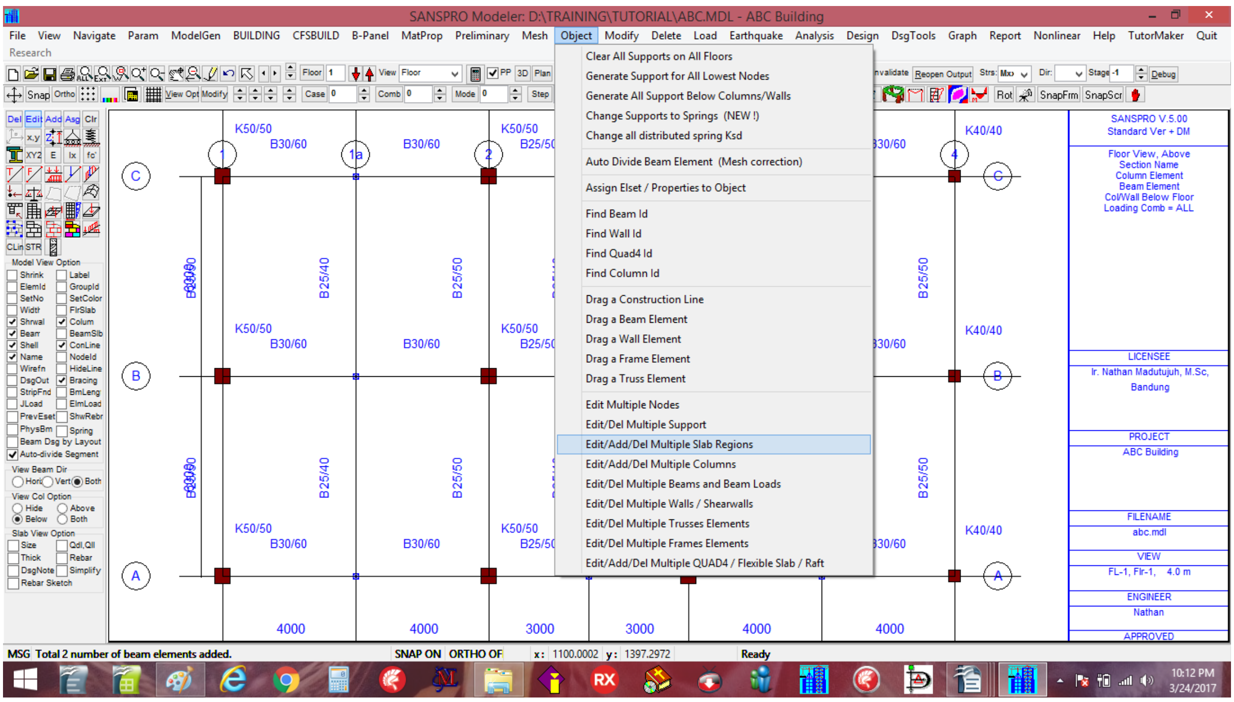
Edit for qDL dan qLL as follows:



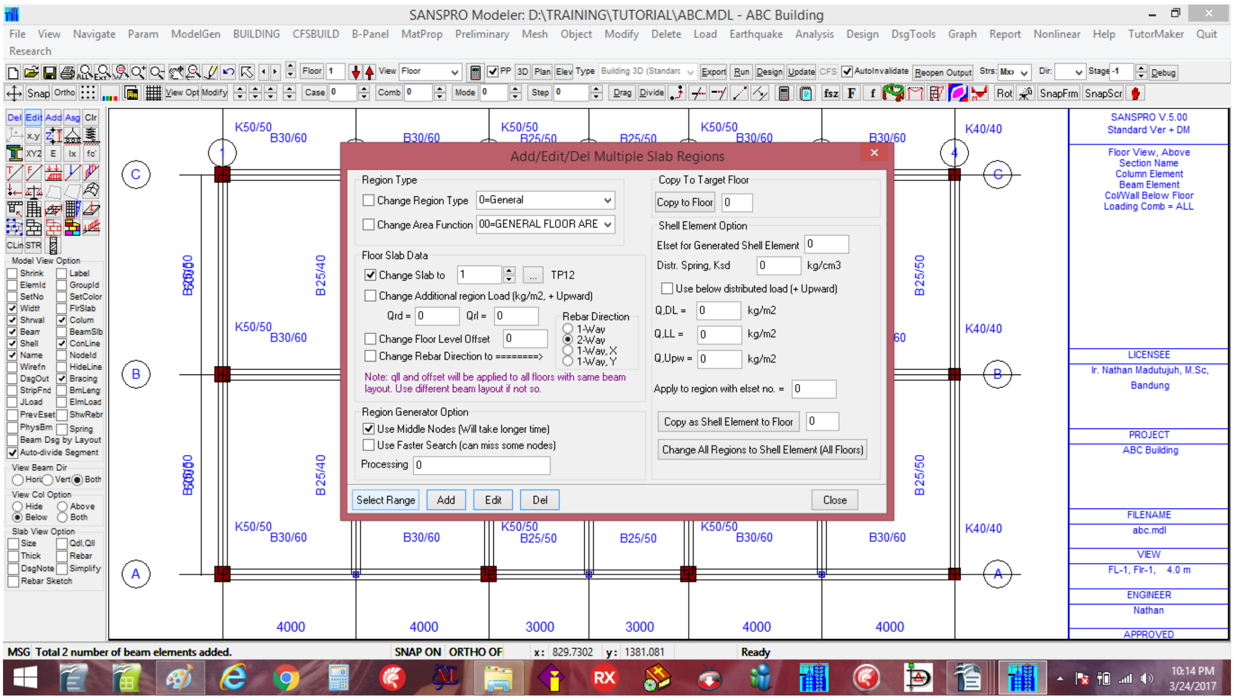
13. Generate Floor Slab Elements

Because the layout regions are rectangular we can use floor slab region generator for this purpose. Any missing region of slab can be generated later manually.

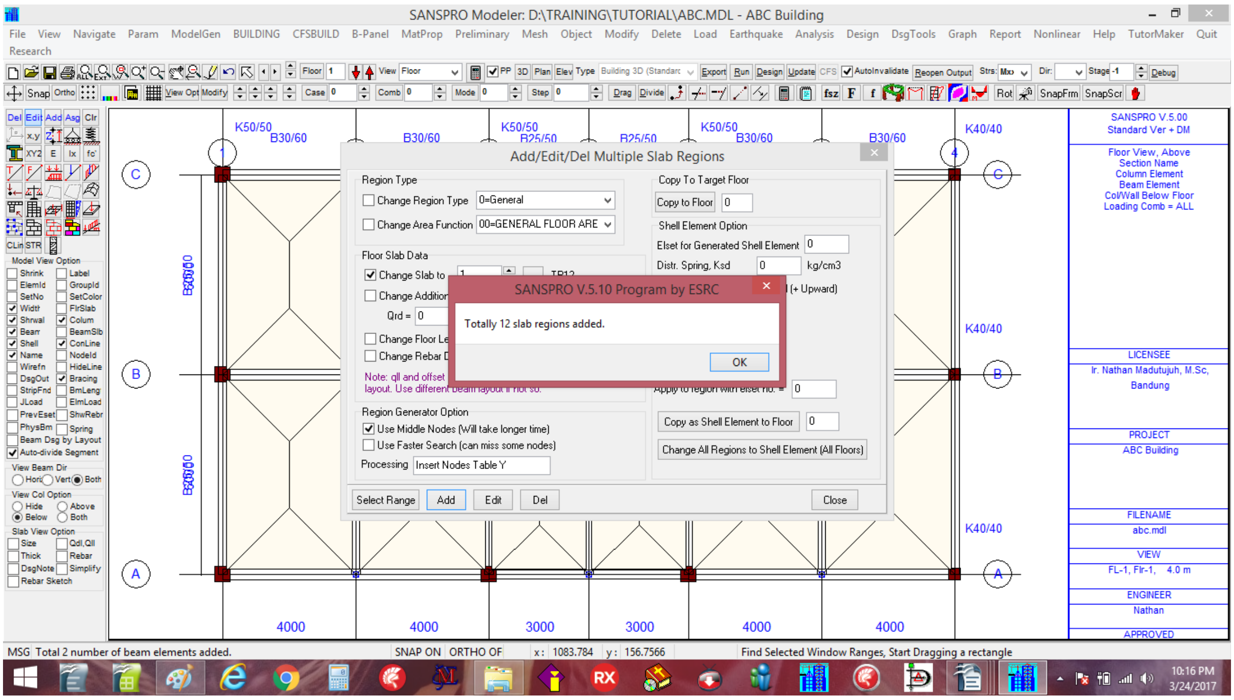
Click menu **Object – Edit/Add/Del Multiple Slab Regions**



click **[x] Change Slab to [1] Tp12**



Select a window range, click **[Add]**
Totally 12 Regions will be added.



Floor slab regions at floor 2 will also automatically follow floor 1.

Floor with different layout:

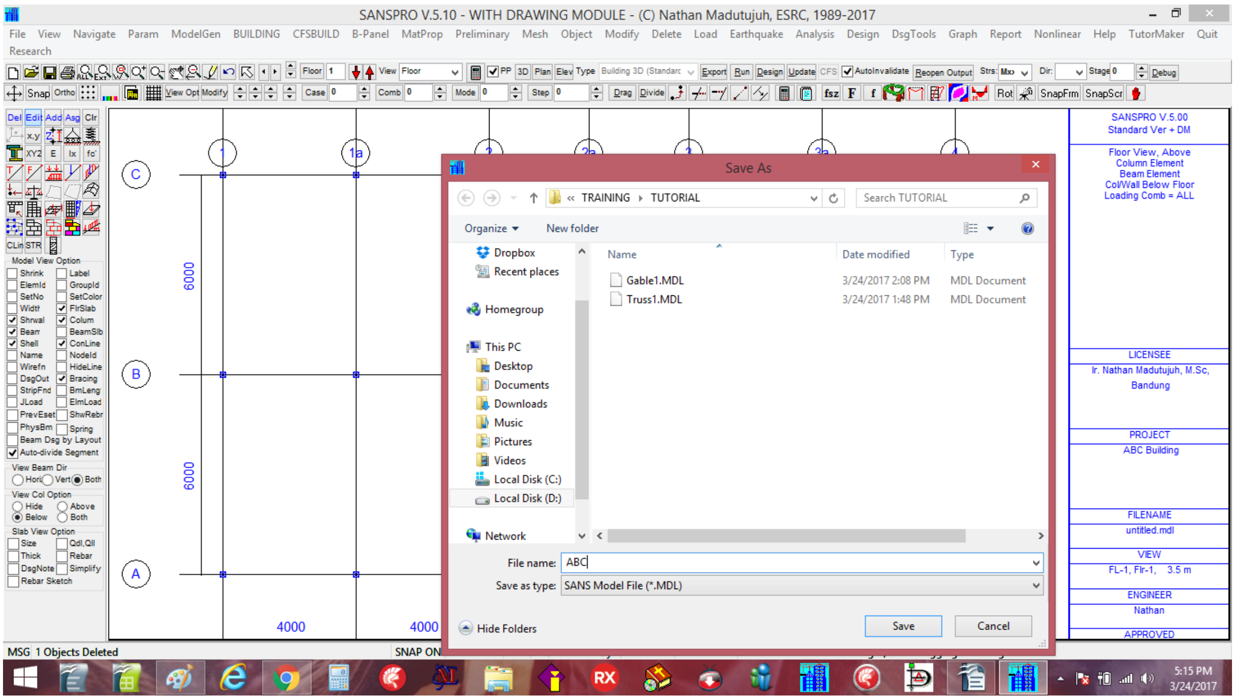
For floor 3,4 layout, we will use other method, by copying current layout and deleting some elements. We will do this after adding some beam loads.

Adding a floor region manually:

User can add a certain floor region manually by clicking icon, click icon Add and click 4 nodes in counter-clockwise direction. For a slab region with 3 nodes, the third nodes is clicked twice at the same location.

Saving the model

Before we continue, we will save the model :
Click **File – Save** or press **F2** and give name : ABC
Avoid file name with space or dot or commas or other strange characters.



14. Generate Beam Load Table and Beam Load

Beam load from weight of brick wall will be added as follows:

hi = 3.5 m – hb = 3.5 – 0.5 = 3.0 m, qwall = 150 kg/m2

qbeam = 150 x 3 = 450 kg/m = 4.5 kg/cm (- sign means downward)

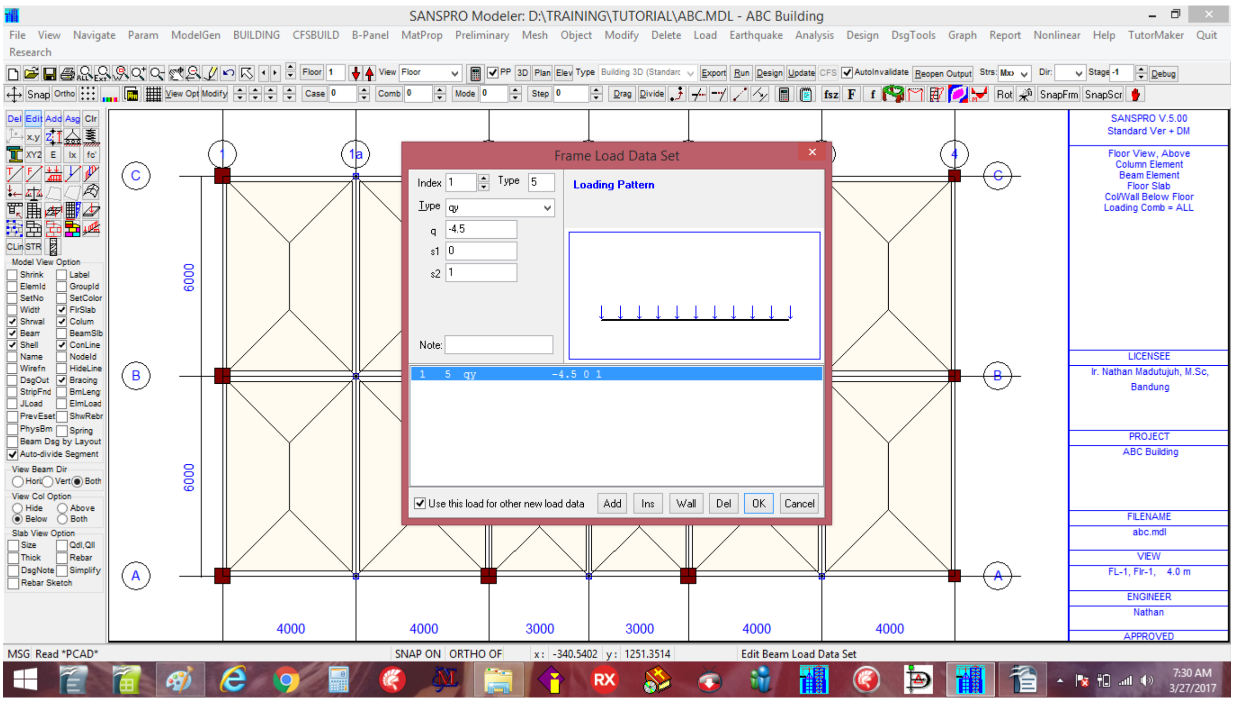
Click icon  and add the following data:

Click icon [Add], enter Type = **qy**, q = **-4.5**, s1 = **0**, s2 = **1**

Click the shaded row, the right beam load data will appear on shaded row

Repeat if there are more than one beam load type

This beam load type can be used by many beams later.

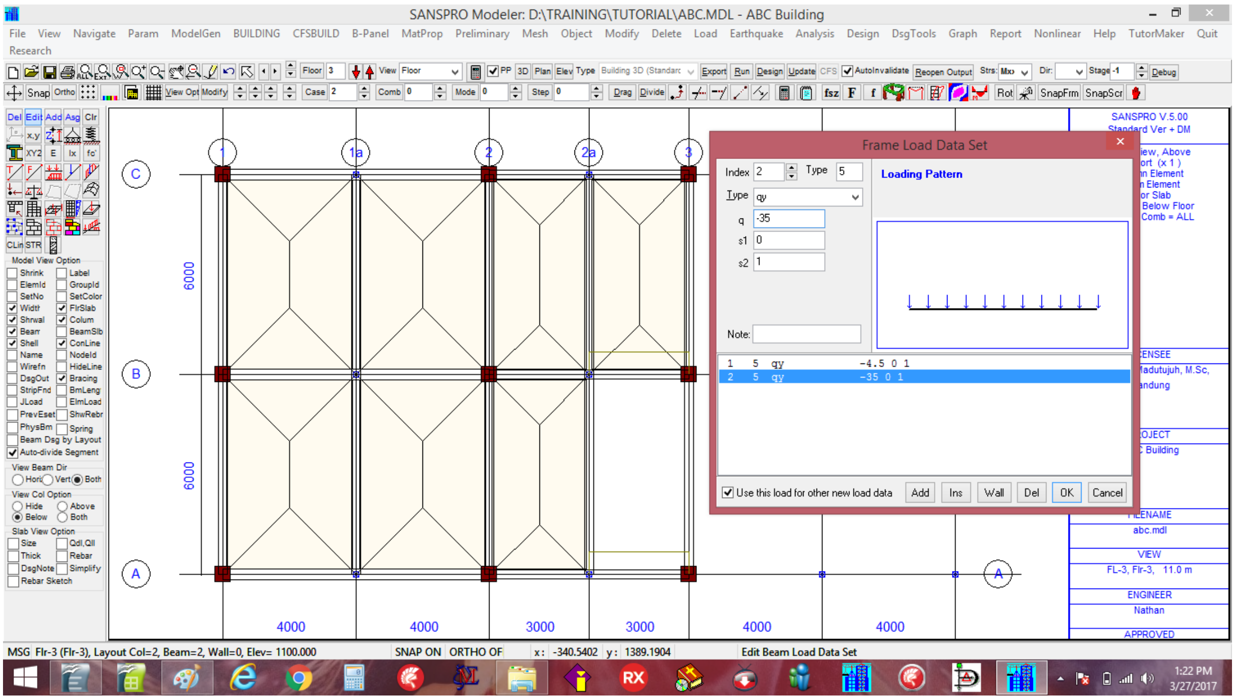


Add beam load for staircase:

For staircase with U-shape, the stiffness of the staircase is not very significant to the whole structure, and only its weight will be included in the model as a beam load.

Total length of staircase = 6m / cos(30 deg) = 6.92 m → take as 7.0 m
Total load of a staircase = 1000 kg/m2
qs = 1000 x 7 = 7000 kg/m → the load is divided into two beams, upper and lower beams :

qb = 7000/2 = 3500 kg/m = 35 kg/cm



Add the staircase beam load to upper and lower beams.

Note: - Floor slab at staircase void must be deleted if the staircase beam load already defined.

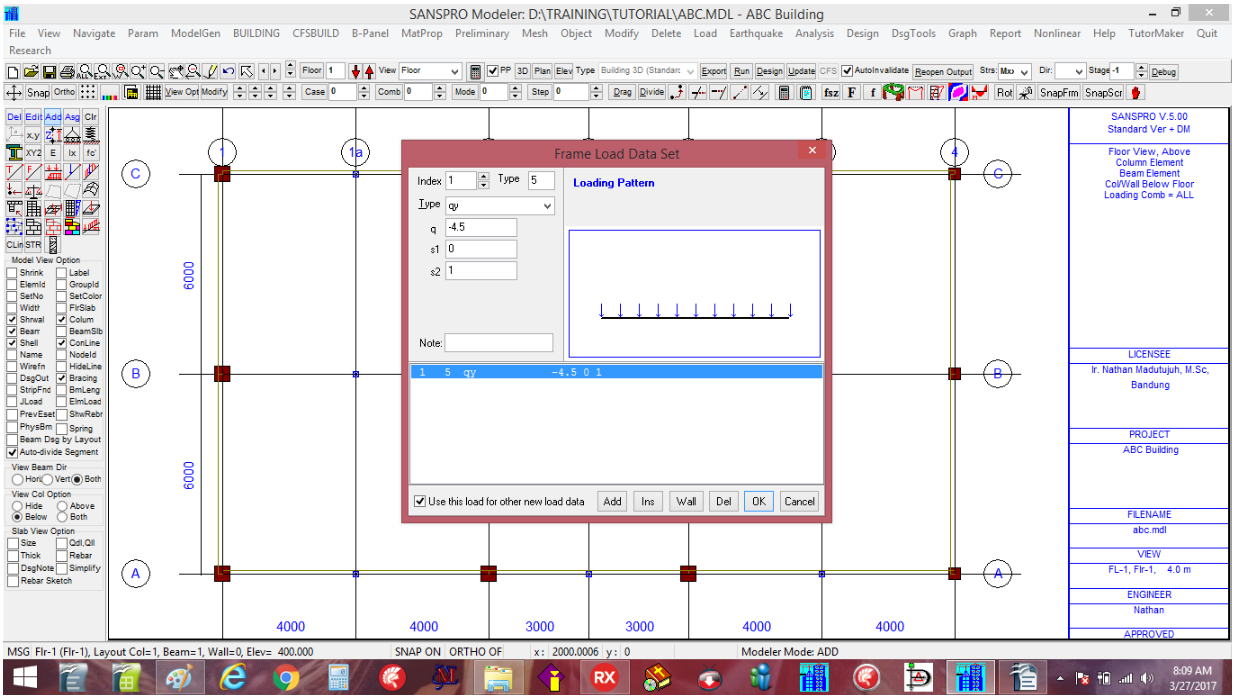
15. Add Beam Load to beams

To add a beam load to a beam:

– Hide Floor slab regions first to easier the visual checking
– **Go to floor 1**
– Click icon , click icon 
– Click an edge beam
– Select the beam load from table
– Click [OK]
– A second line will appear on the selected beam.
– If a beam has two or more copies of the beam load, the second line will change its color

(This to enable user easily check if he has click twice or just once. In some model, the double beam load may be needed.)

– Repeat for other beams



16. Create model for Floor 3 and 4

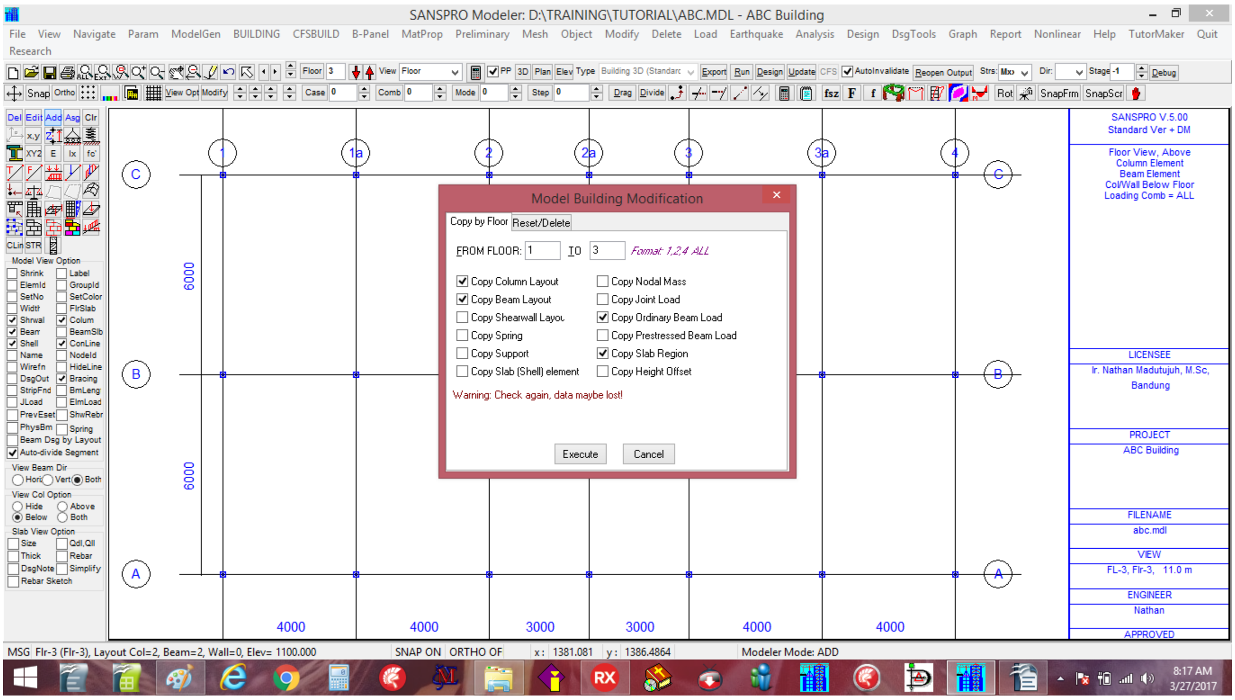
Floor 3 and 4 use the same layout no. 2
We can go to floor 3 or 4 and repeat the same above process OR
we can use a facility from SANSPRO called MODIFY BY FLOOR command as follows:

Warning:
- This is a very useful faility to work on a floor or several floors using only simple command
- We can copy or delete a floor easily
- Please use this modify by floor comamnd carefully because it can add or erase a whole floor in one click

The steps to create data for floor 3 and 4 are as follows:

– Go to Floor **3** (Floor 4 has the same layout as floor 3)
– Click menu **Modify – Modify/Copy/Delete Objects by Floor**
– Enter the following data:
– Click **[Execute]** after entering the data, answer with **YES**

– Now floor no. 3 and 4 will have their data appeared
– We must delete some objects to reflect the actual model

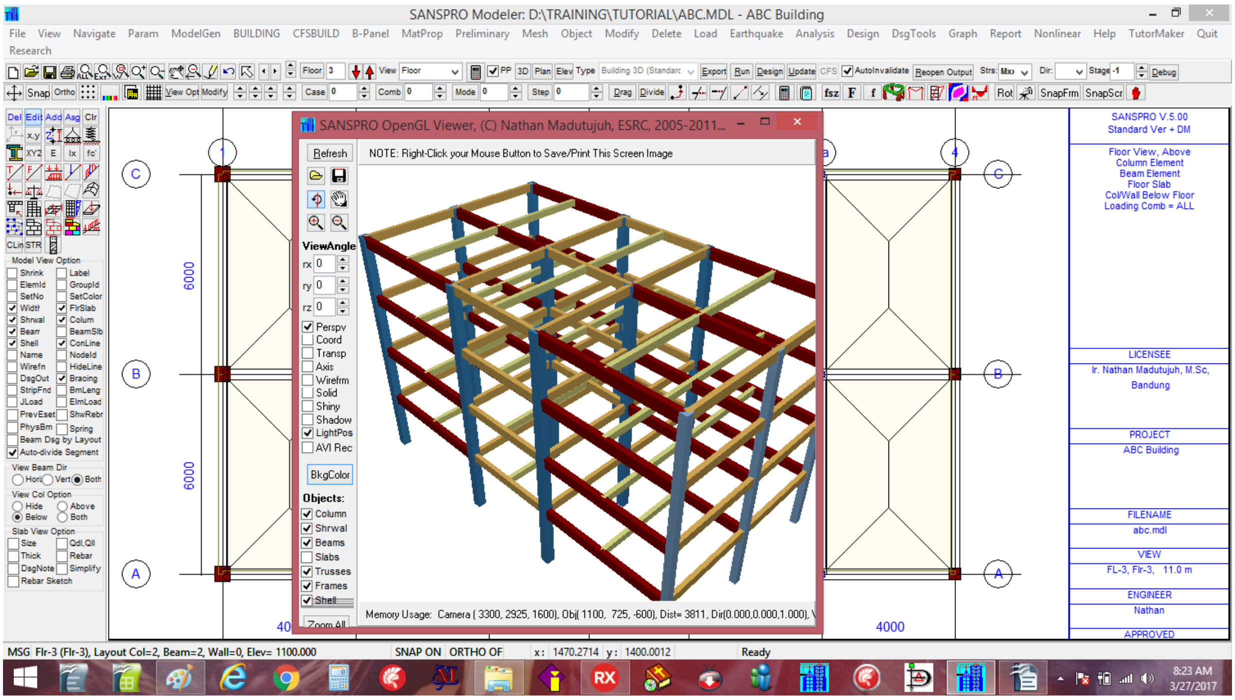


Here we want to copy data from floor 1 (with layout no. 1) to floor 3 (with layout no. 2) :

[x] Copy Column Layout
[x] Copy Beam Layout
[x] Copy Ordinary Beam Load
[x] Copy Slab Regions

Although the floor layout no. 2 is smaller than layout no. 1, we will copy it then delete some objects to reflect the actual model.

Checking the 3D view of the model (Click icon)



From the 3D view we see that the floor 3 and 4 now have same layout as floor 1 and 2.
We will delete some objects from floor 3 and 4 to reflect the real model.

16. Deleting some objects

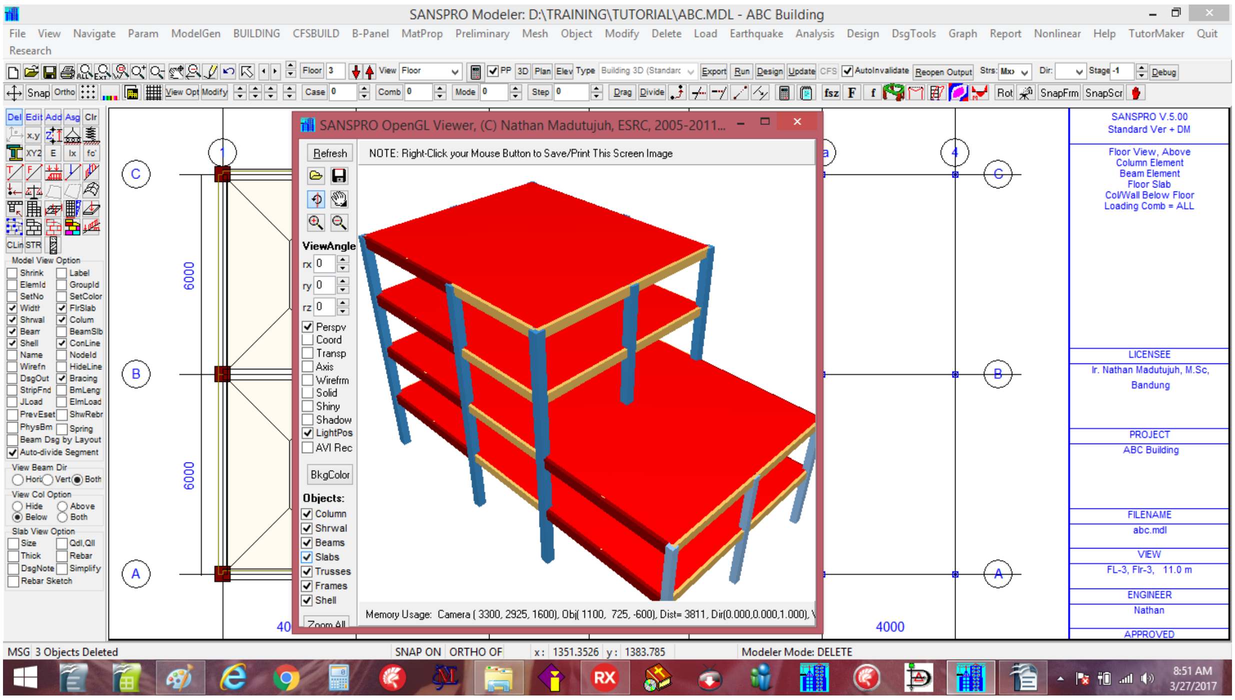
Note: - Be careful on using the delete commands, because SANSPRO only provides limited undo for delete commands.

To delete objects :

- click the icon of the object
- click Del Icon
- click near centroid of any object to be deleted (repeat for multiple objects to delete)
- Selected objects will change color to red
- click icon Del again to confirm deletion → selected objects will be deleted

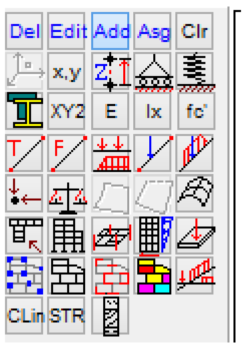
Note: Repeat for columns, beams, slab regions objects.

- Check the 3D View again:



Now floor 3 and 4 have the right objects and smaller floor layout.
Click icon [x] Slabs to view slab region object in 3D View.

17. Define Nodal Supports

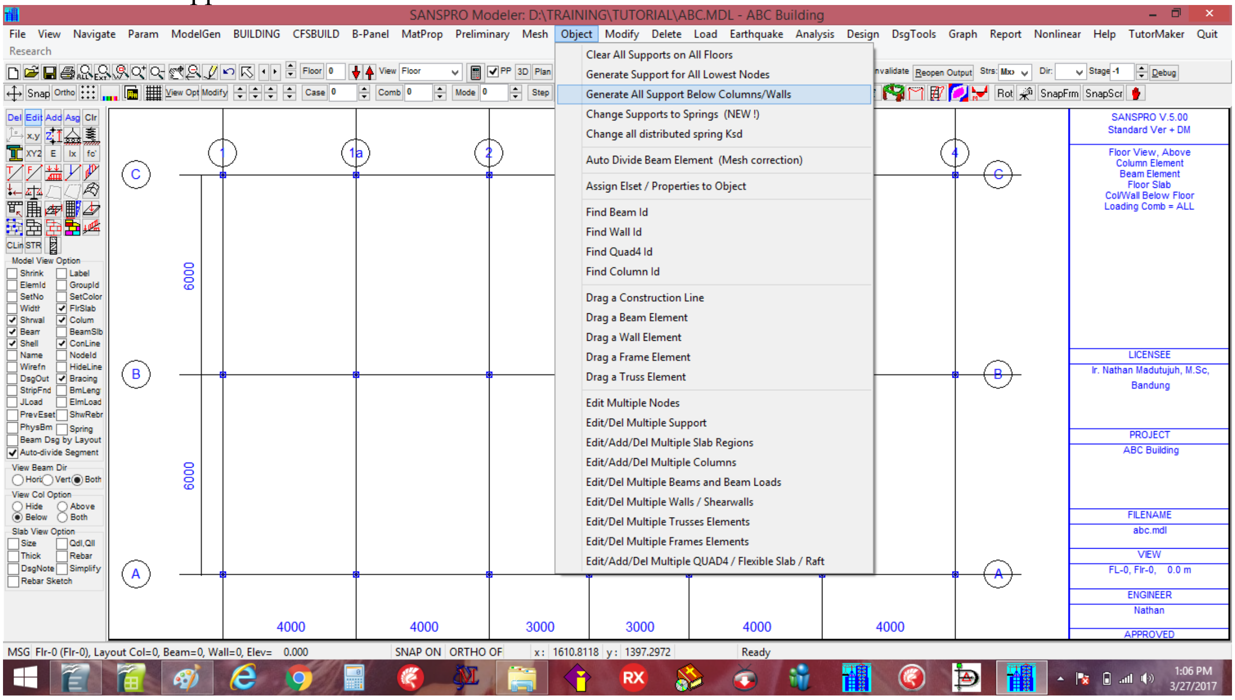


The last objects that needed to be defined are nodal supports (2 methods).
First, move to floor 0 or the lowest floor.

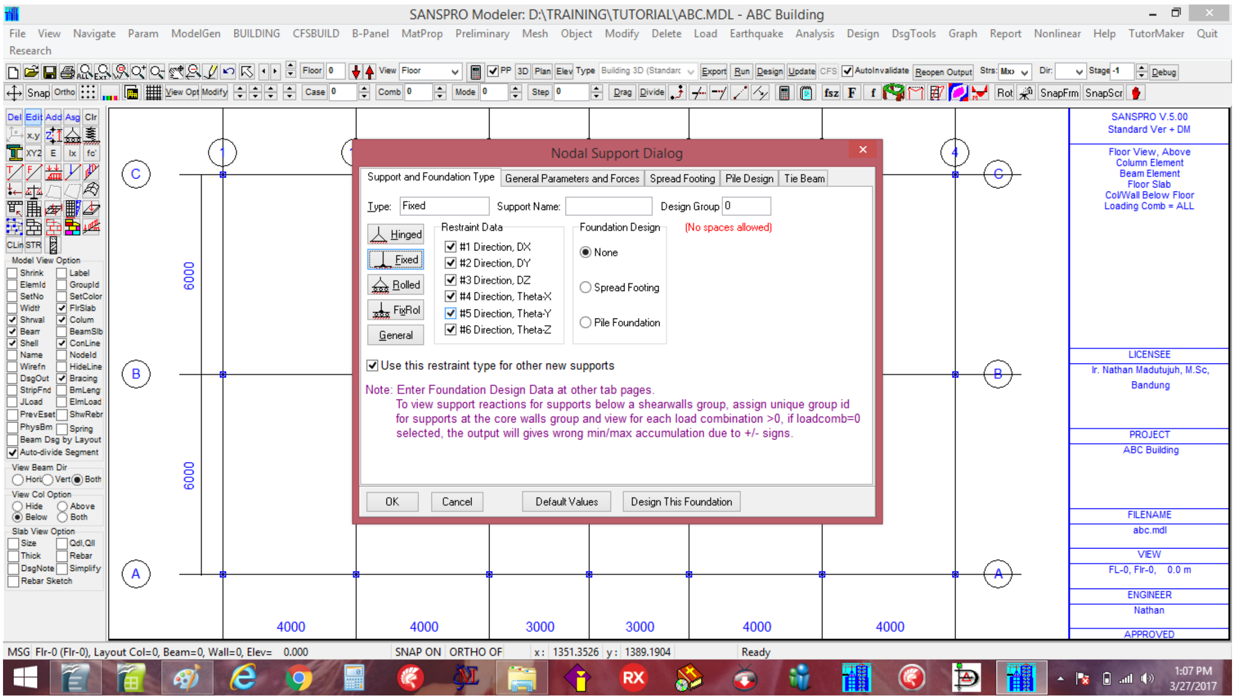
User can add a nodal support manually similar to adding a column, by clicking the icon and icon Add, then select support type available: hinges, rolled, fixed or general type (user defined type).
Repeat for all supports.

Other easier method to generate all supports easily is by clicking menu **Object – Generate all supports below columns or walls**, then select a support type**.** All nodal supports below column or wall will be added automatically.

Generate all supports below columns or walls

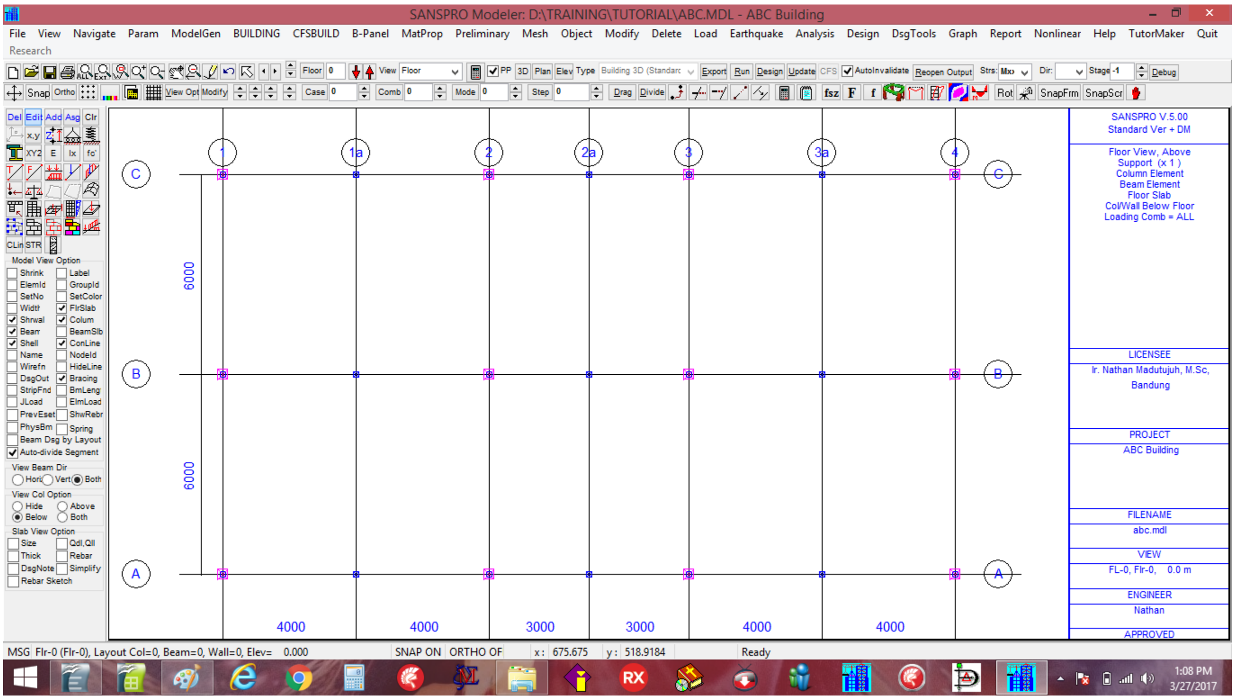


Answer question "Clear all support at this floor ?" with **Yes.**
Select a **Nodal Support type** and click **[OK]**.

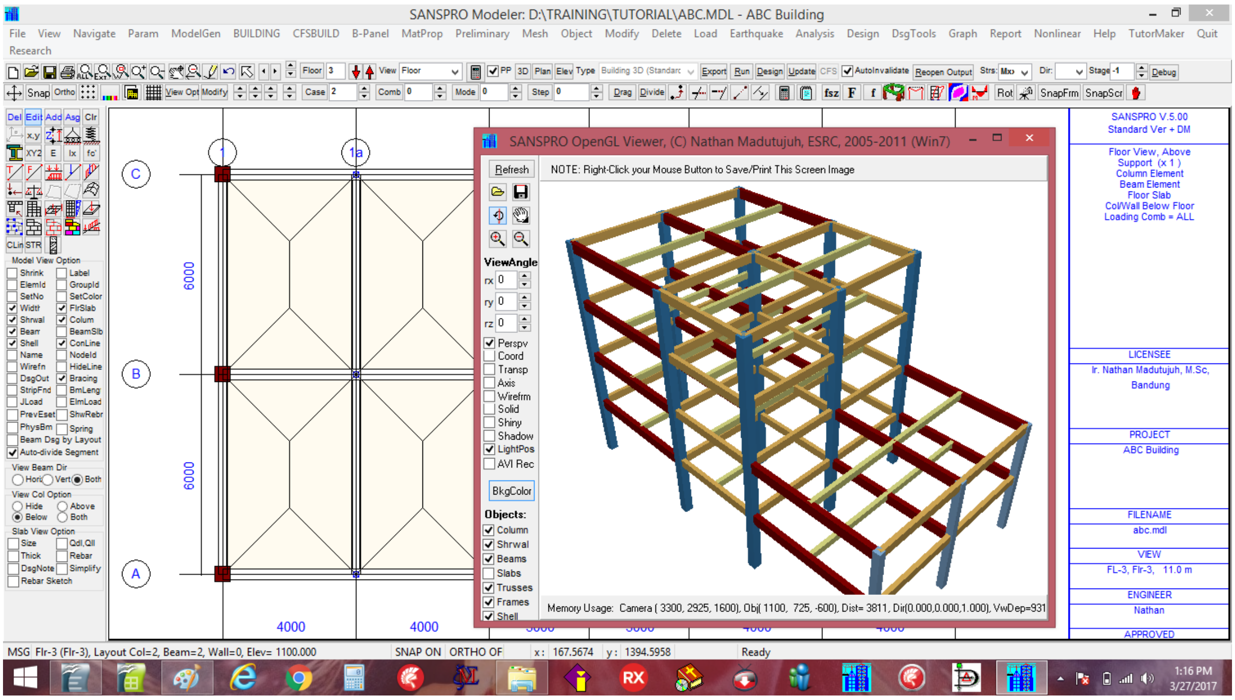


All supports will be generated automatically.

All nodal supports generated:

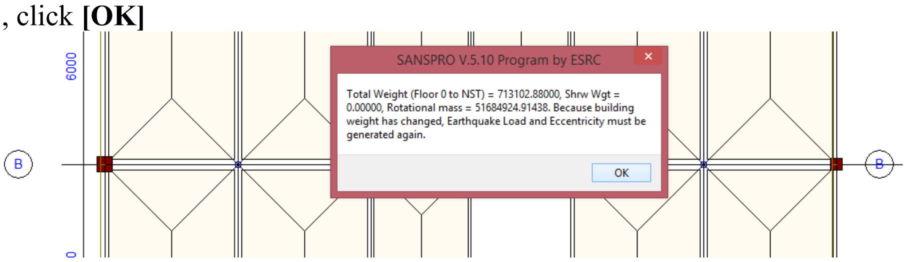


Now a complete geometry model of a building has been defined.
We can continue with generating earthquake load, analysis and design process.



18. Compute Floor Weight

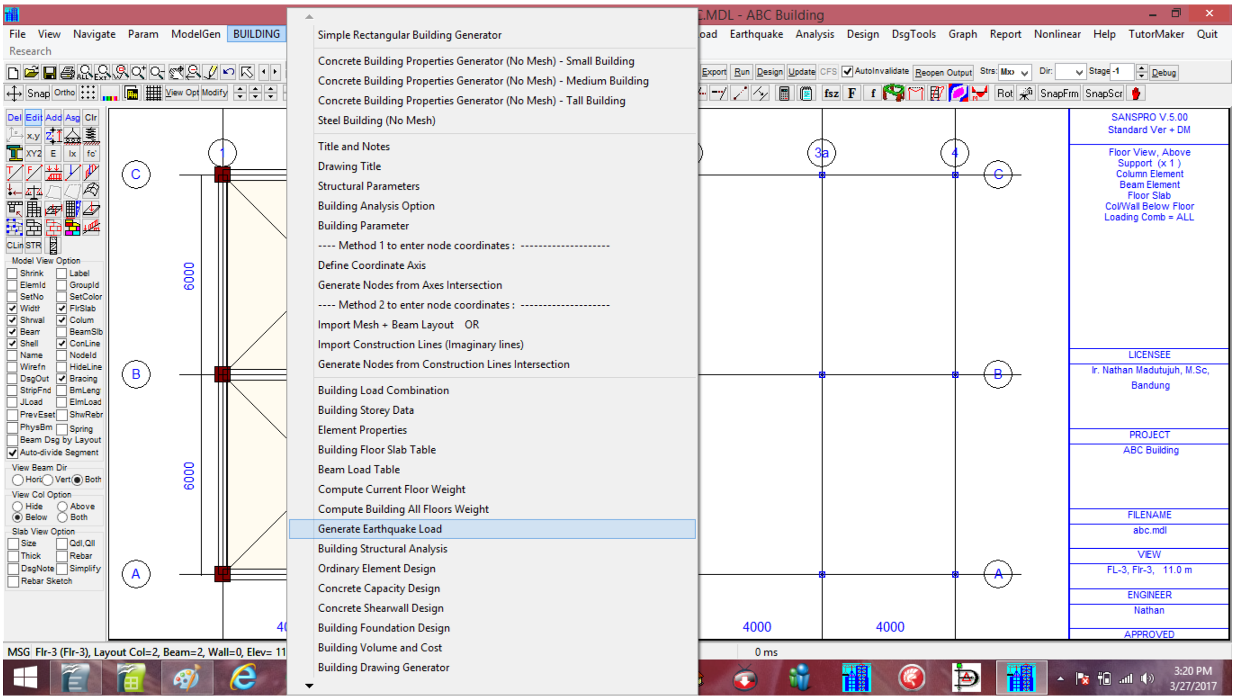
Building floor weight must be defined before continuing with next steps.
Click menu **Building – Compute Building All Floors Weight**
Click **[OK]**, click **[OK]**



Total Weight = **713102.88** kg

19. Generate Earthquake Load

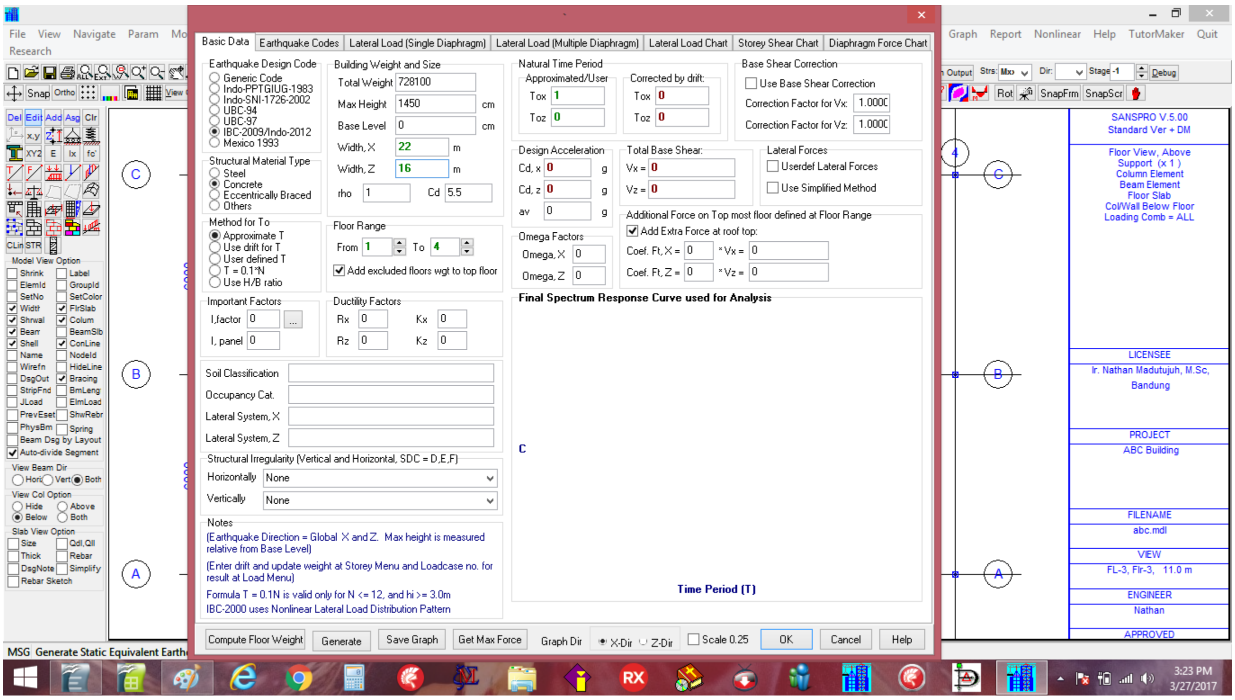
After Building Weight computed, we can generate Static Equivalent Earthquake Load.
Click **Building – Generate Earthquake Load**



Menu for Earthquake Load generator will appear:

At **Basic Data Page**:

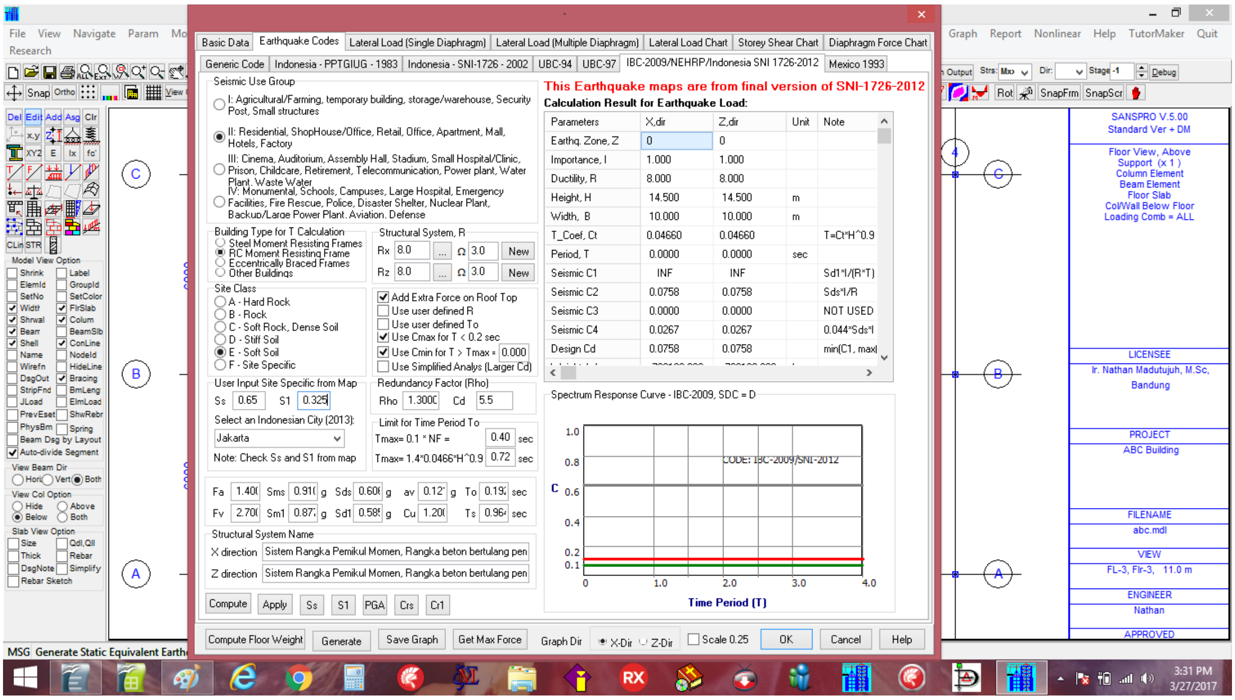
Earthquake Design Code : **IBC-2009/Indo-2012 (SNI-1726-2012)**
Enter Building Width,X = **22m**, Width,Z = **16m** *(distance between farthest columns in X,Z direction)*
Building Height will be automatically calculated by the program.



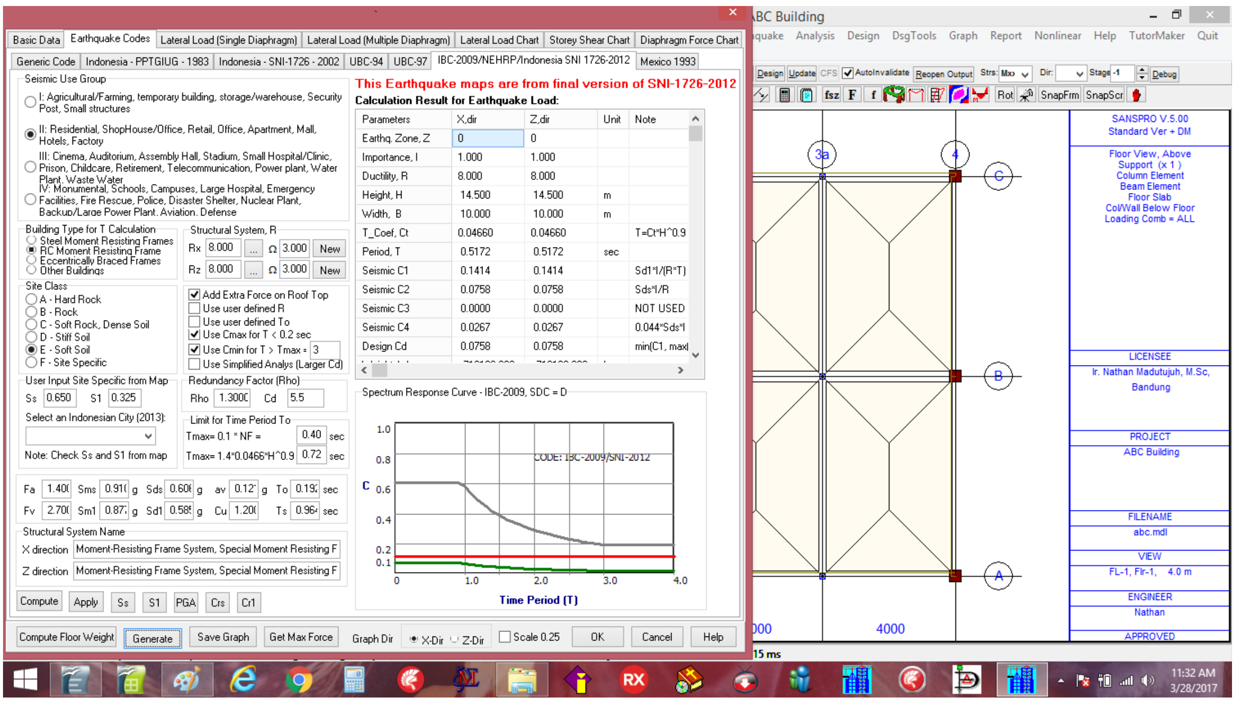
Go to page **Earthquake Codes**:

<u>Enter the following data:</u>

Seismic Use Group : II : Residential, Shophouses, Office, Retail, Apartment, Mall, Hotel, Factory
Building Type for T Calculation : RC Moment Resisting Frames
Site Class : E - Soft Soil
Rx = 8.0, Omega = 3.0, Rho = 1.3, Cd = 5.5
Rz = 8.0, Omega = 3.0, Ss = 0.65, S1 = 0.325 (for Jakarta area)
Use Cmin for T > Tmax = 3.0 secs



Click **[Compute]**, Click **[Apply]**, Click **[Generate]**
Response spectrum curve now has been defined.

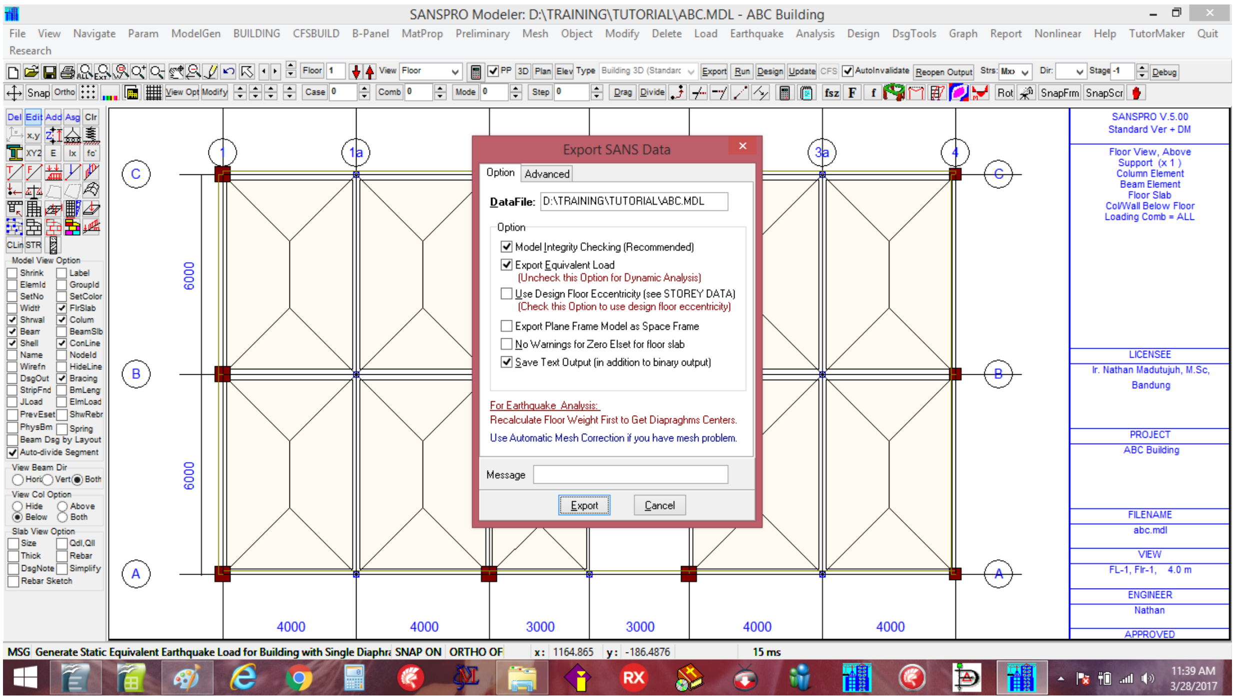


<u>Preparing for Analysis:</u>

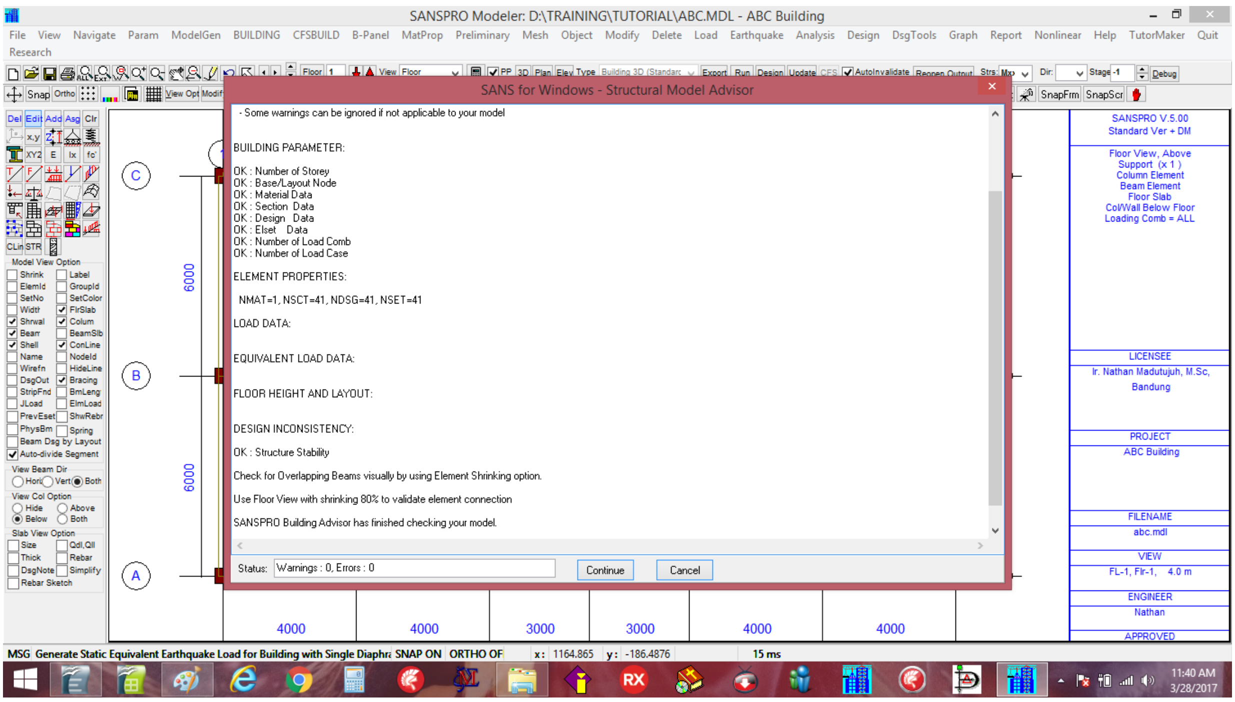
– Save data by clicking **File – Save** or **F2**

– Export data by clicking **File – Export** or **F4**

When we export data for analysis, SANSPRO will do some important model checking to ensure that the model is suitable for analysis. SANSPRO will check for most common modelling error and mistakes. If there is an error, it is usually a serious one that must be fixed immediately before running analysis. But if there is only warning, some warnings that are not applicable to the model can be neglected.



Click **[Export]** to continue with Model Advisor Checking.

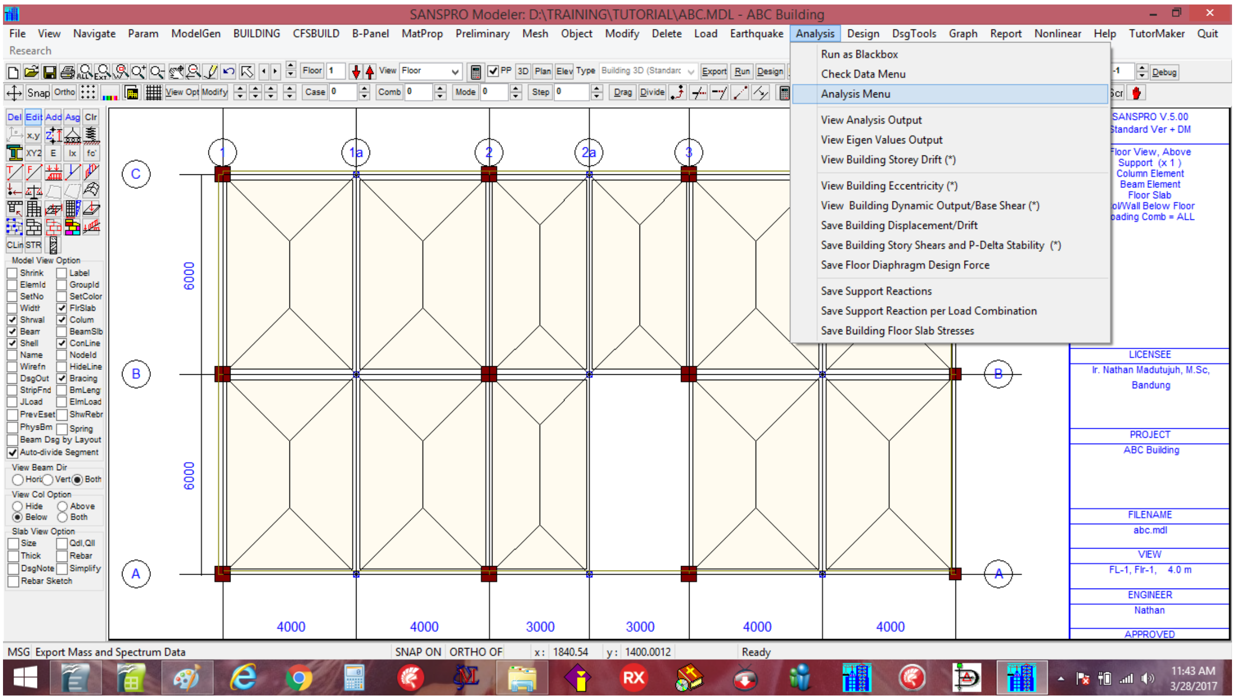


Good. No warnings for this model. So we can continue with export data for analysis by clicking **[Continue]**. Click [Ok].

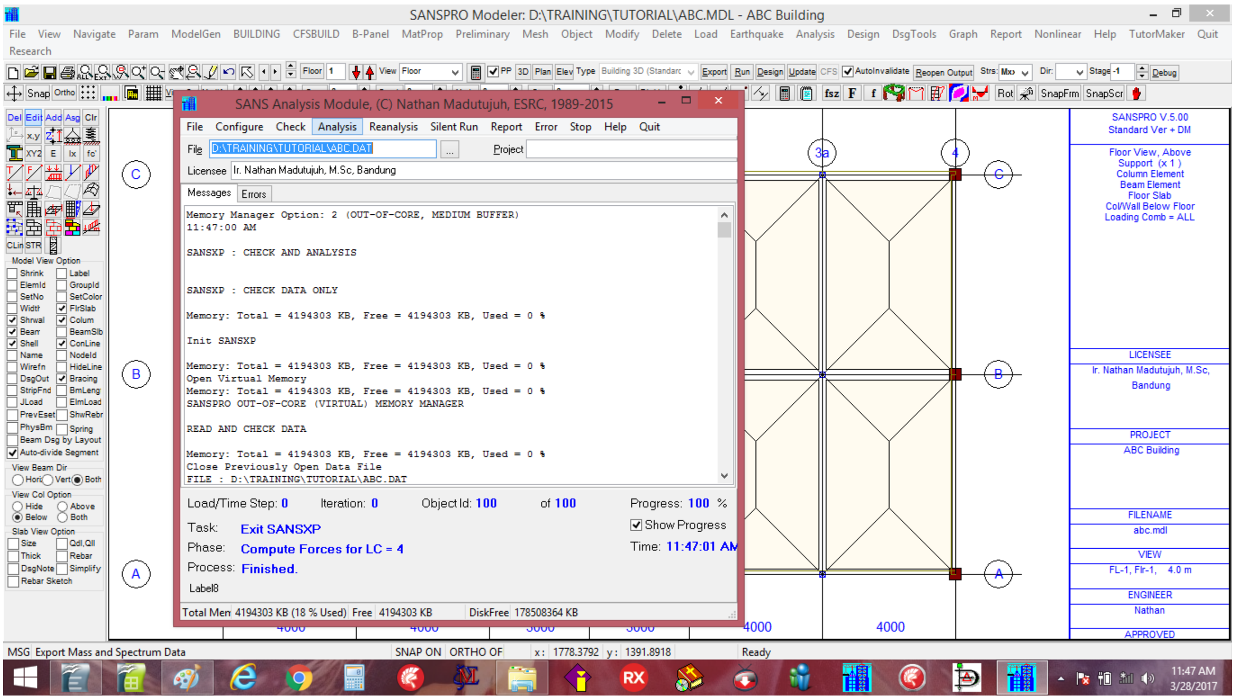
Now the model has been prepared for analysis.
We will run static analysis because this model is using Static Equivalent Earthquake Load.
Analysis process also may gives some warnings or error messages.

20. Continue with Analysis

Click menu Analysis – Analysis Menu



SANSPRO Analysis Menu will appear:
Click **Analysis** to Start Analysis.



SANSPRO Analysis module is very fast and can be run inside the model.
This model can be run in 0.953 secs. Click **[Quit]** to back to model editor.

Because of the analysis time is very fast, user is encouraged to re-run analysis as many times as needed to get the optimized design results. Re-analysis is needed if there is some changes in section size changed, material properties, building height or size, floor height, node coordinates, load values, support conditions, live load reduction, load combination, or earthquake load parameter.

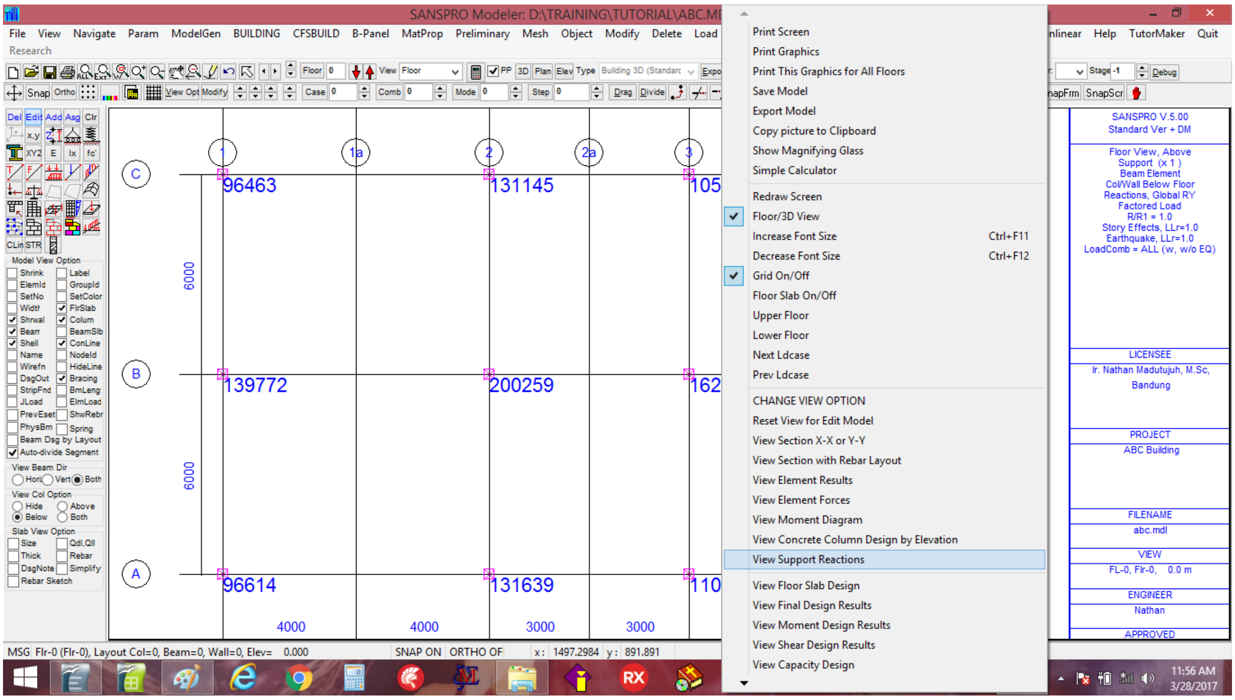
After analysis, we can see : Support reactions, Moment diagram and node displacement of the model

Concrete Floor Slab Design can be visually checked even before analysis if modelled using Slab Region.

Support reactions View

Move to Floor 0.

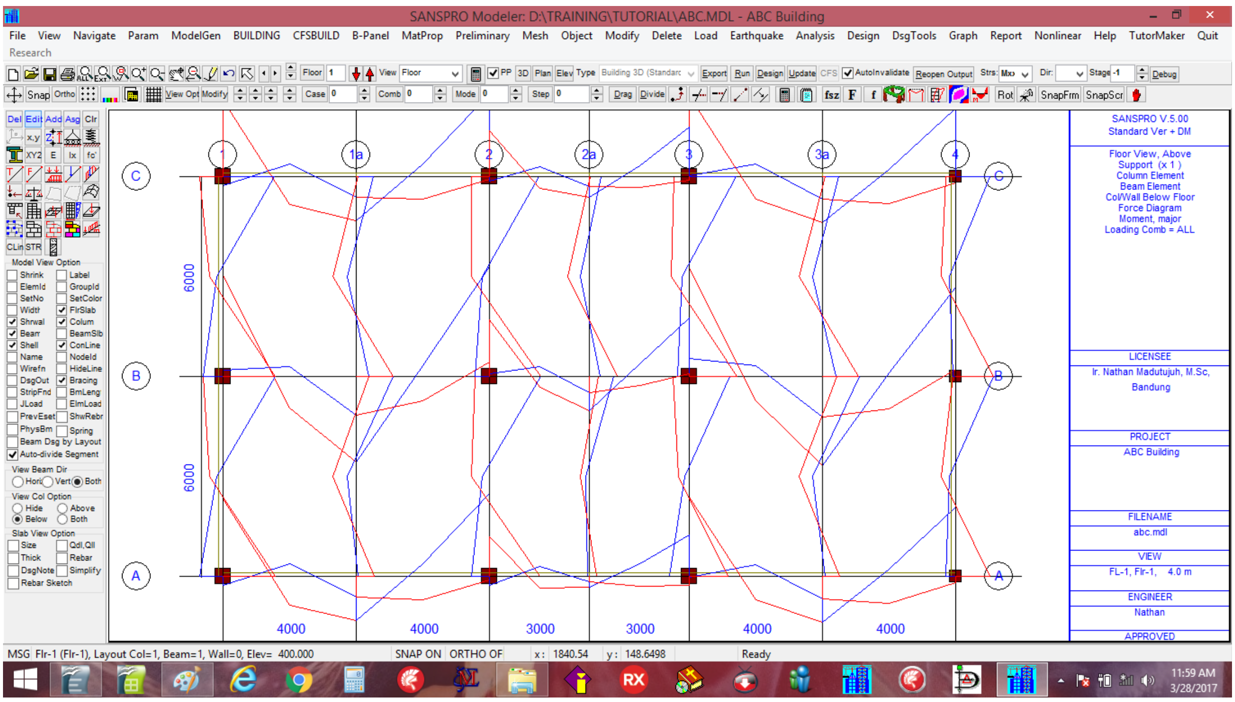
Right-Click, click **View Support Reactions**, all vertical support reactions will be shown (in kg)
(Direction of support reaction and unit system can be changed from **Right-Click**, click **Change View Option**)



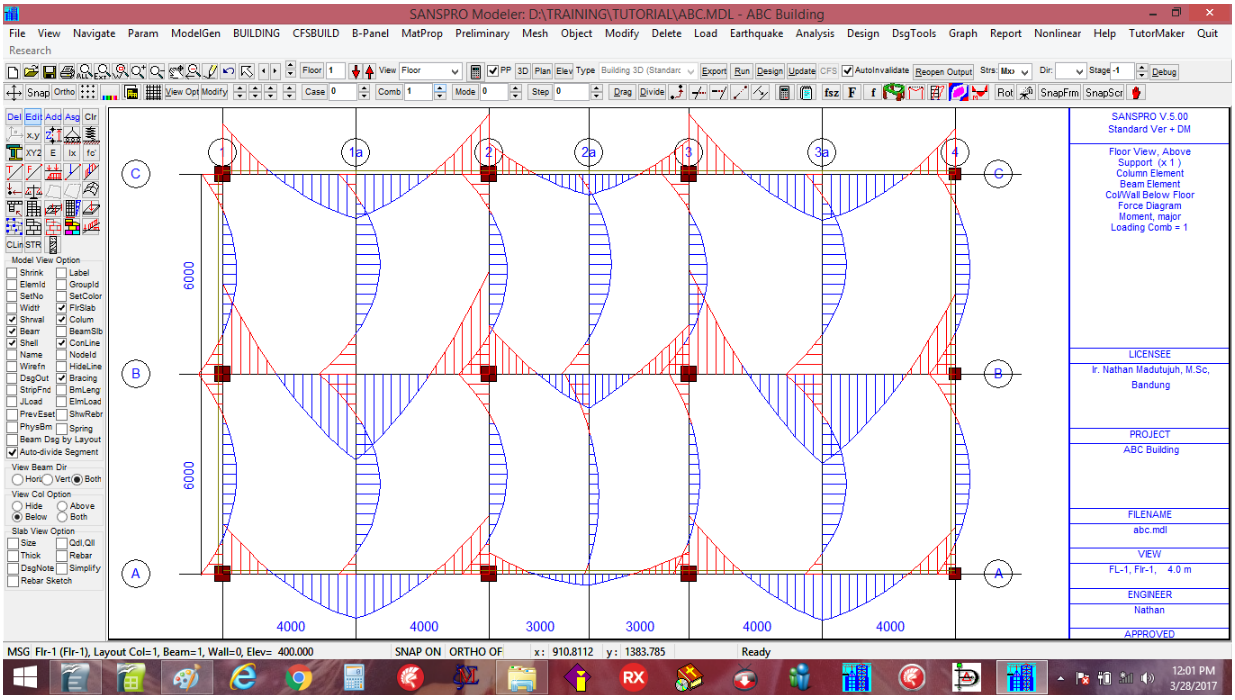
Moment Diagram View

Move to Floor 1 or 2,3,4.

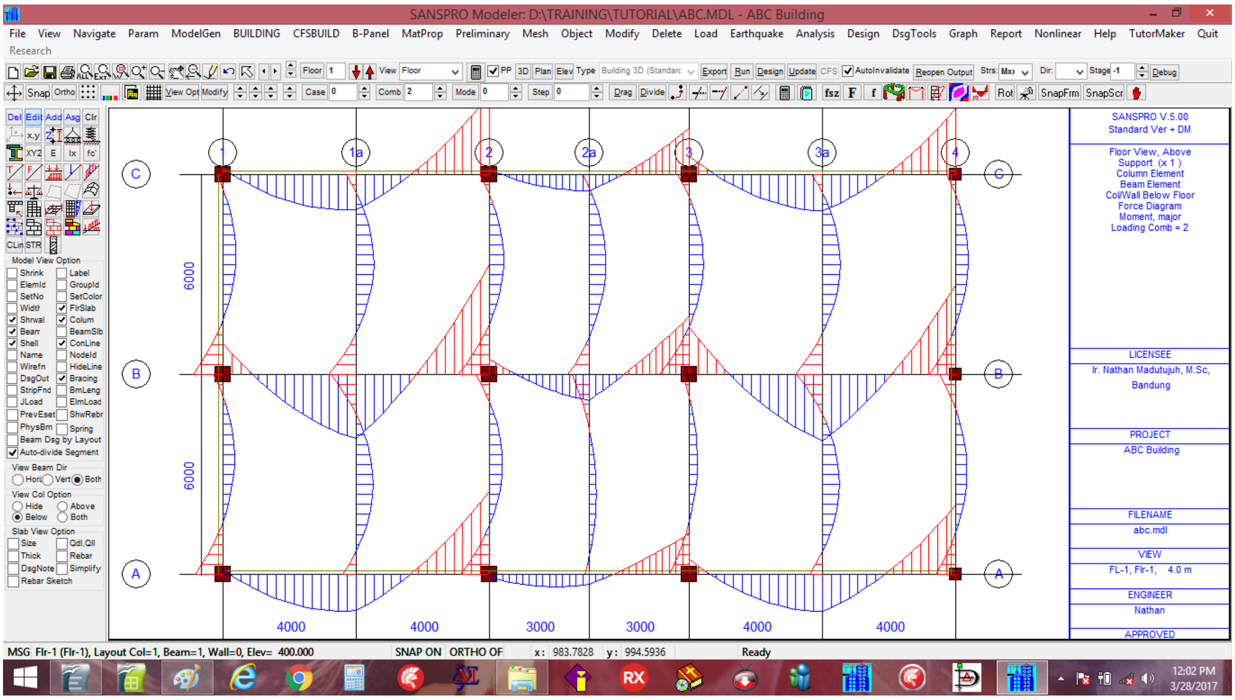
Right-Click, click **View Moment Diagram**, all moment diagram envelope of beams will be shown.
This is envelope of moment diagrams from all load combinations. To see moment diagram from a certain load combination, select the right Load Comb No.



Moment Diagram for Load Comb = 1 : 1.2 DL + 1.6 LL



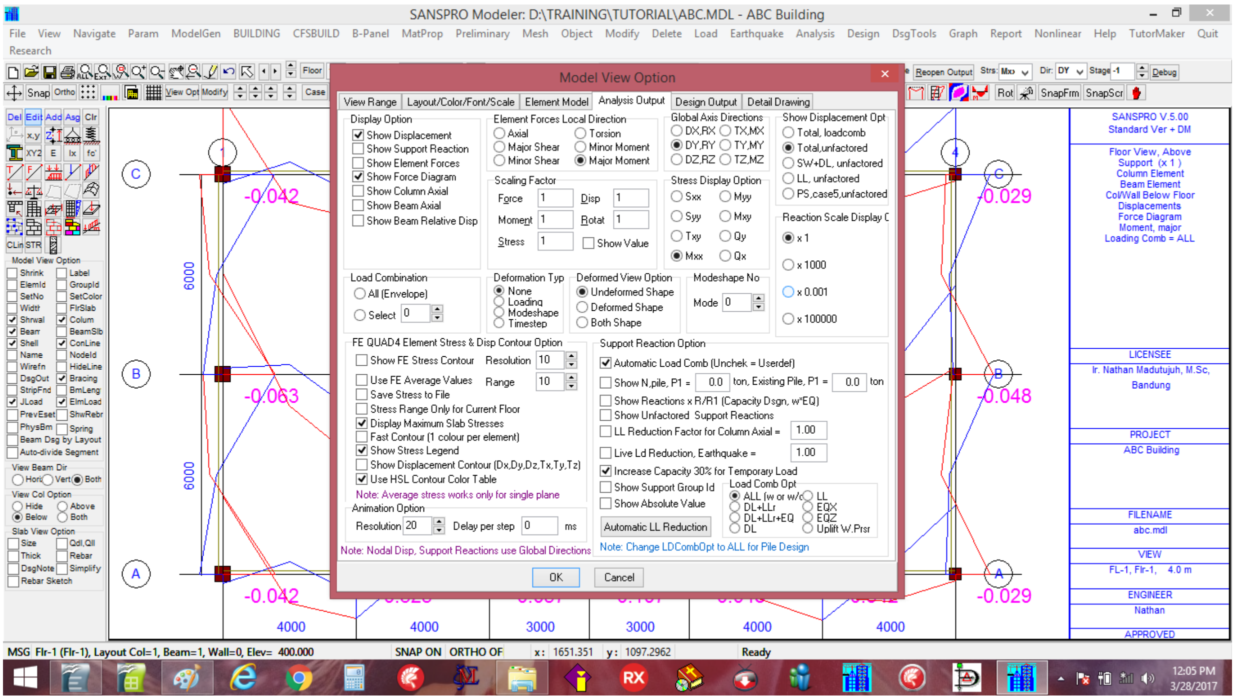
Moment Diagram for Load Comb = 2 (Earthquake in X Direction)



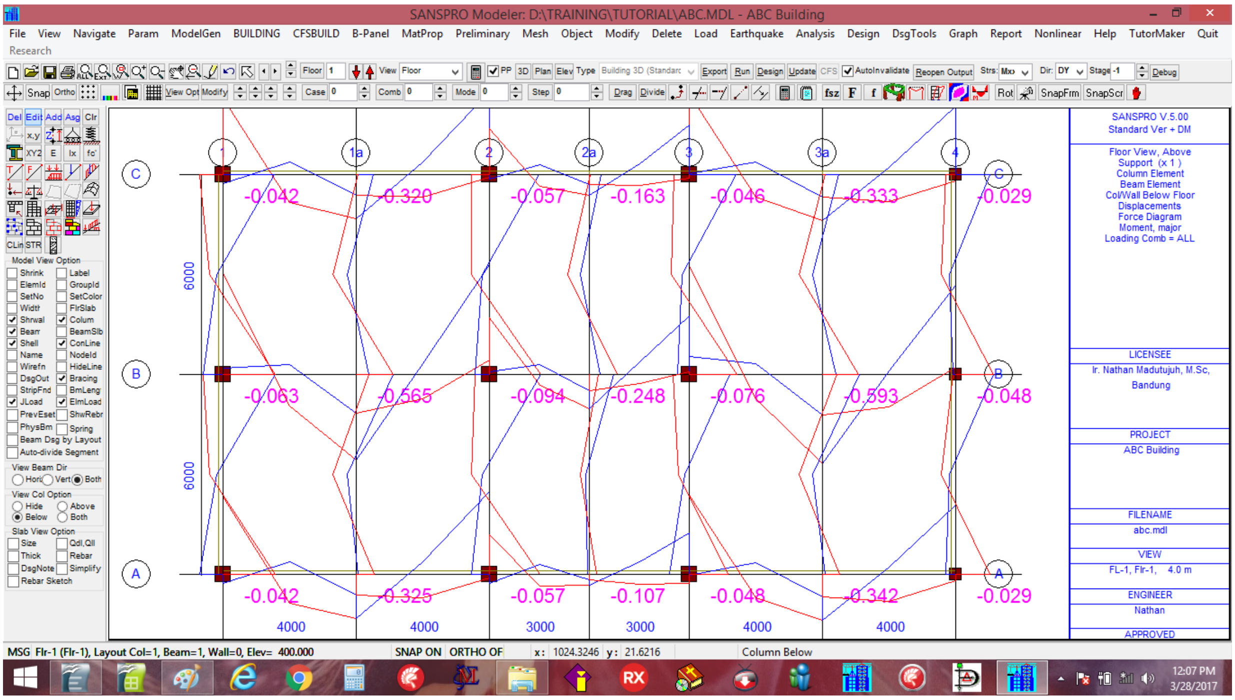
Node Displacement View

Right-Click, select Change View Option, change the following options:

Display Option:	Global Axis Direction	Show Displacement Option:
[x] Show Displacement	[x] DY,RY]	[x] Total, Unfactored



Nodal Displacement in vertical direction will be shown for every nodes in the model (in cm):



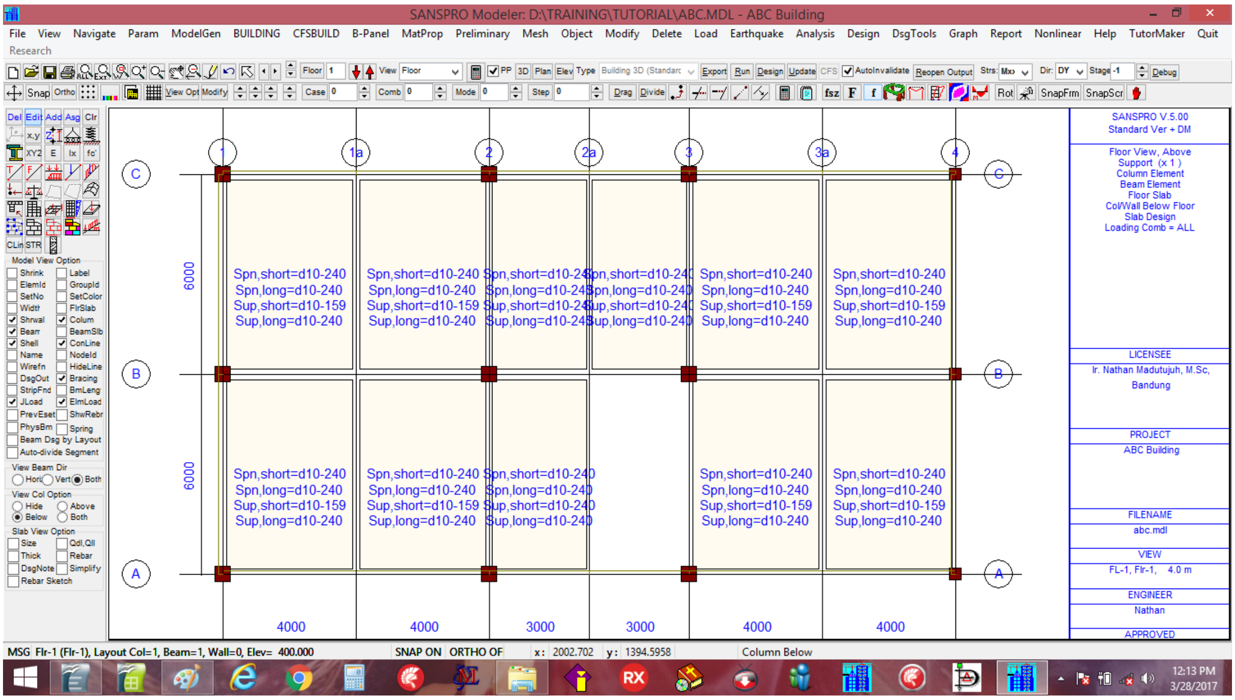
<u>Checking node displacements:</u>

Maximum unfactored displacement usually <= L/300 to L/480.
Maximum displacement is computed as displacement at tip of cantilelever or midspan of beam deducted by displacement at beam ends connected to support or columns.

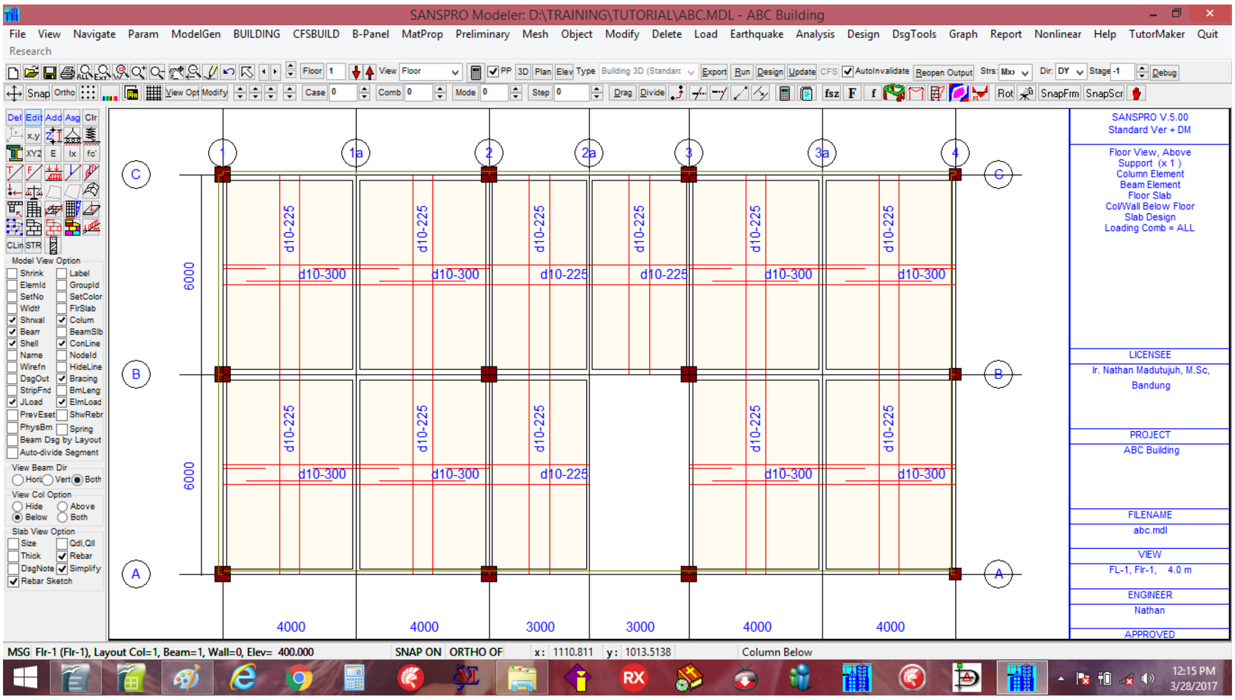
If camber provided, only maximum unfactored displacement from Live Load should be used in above checking.

21. Run Element Design

Concrete floor slab design can be checked by Right-Click, select View Floor Slab Design.

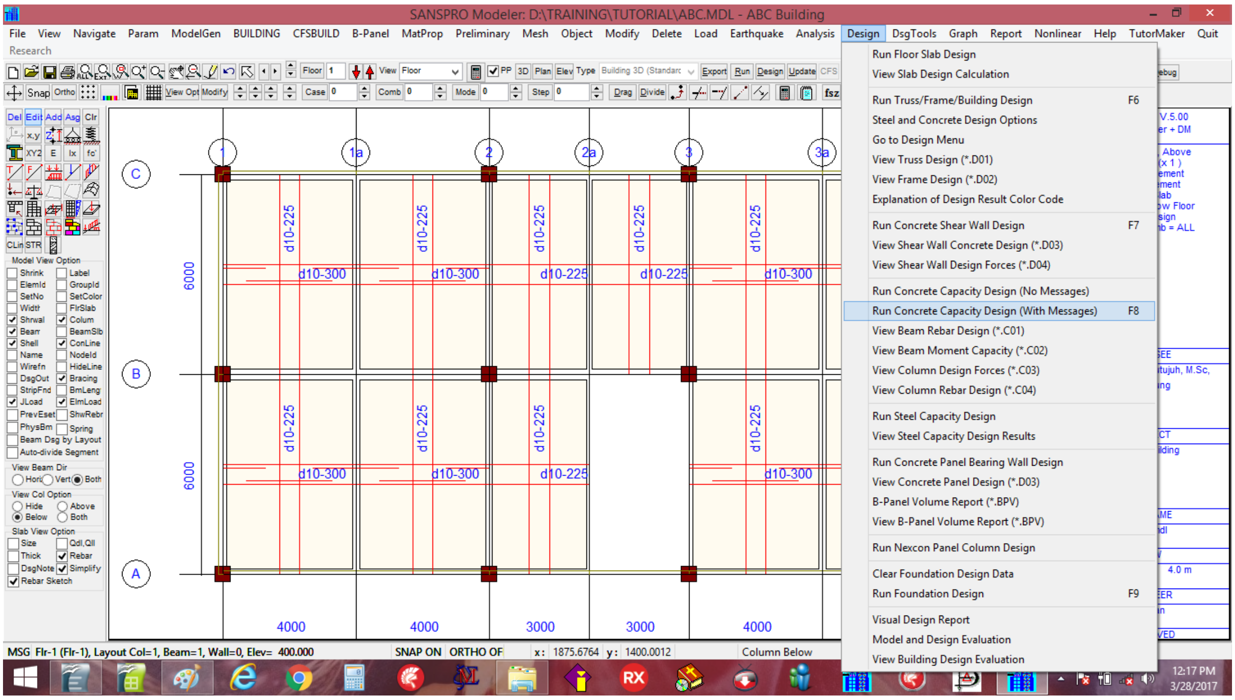


Slab Rebar for mid and support in short and long direction will be shown (in mm).
Click [x] Rebar Sketch, [x] Rebar, [x] Simplify to see the slab rebar sketch for this floor.

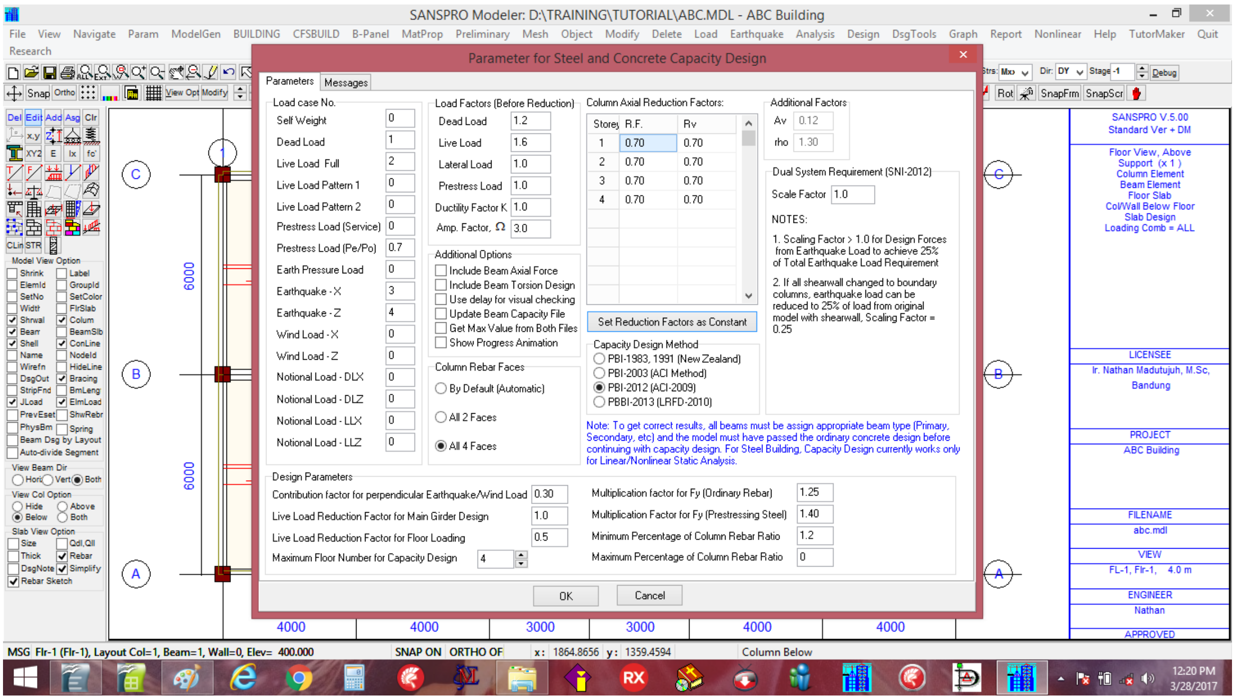


For column, shearwall and beam elements, we must run a certain element design procedure provided. For earthquake zone, after analysis phase, we can run concrete capacity design to find rebar for beams and columns.

Click menu Design – Run Concrete Capacity Design (with Messages) or press F8



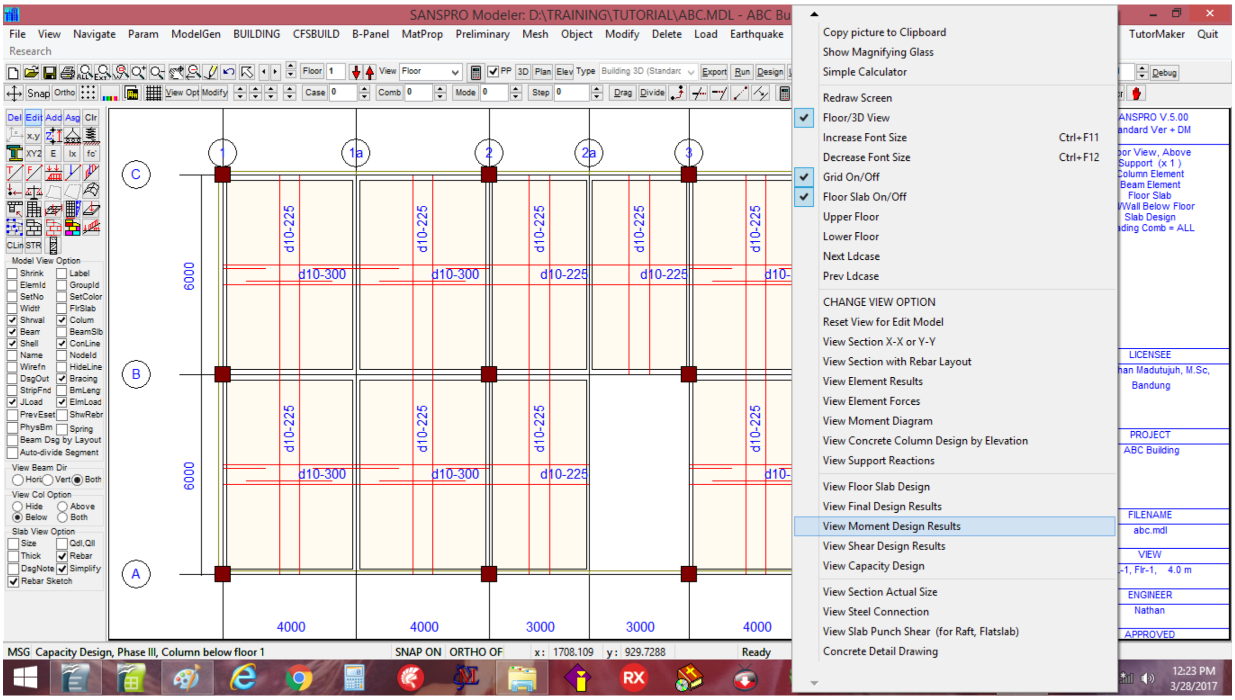
Clik **[Ok]**, then main menu for Concrete Capacity Design will appear.
Click **[Set Reduction Factor as Constant]**, enter **0.7** for column axial reduction factor.



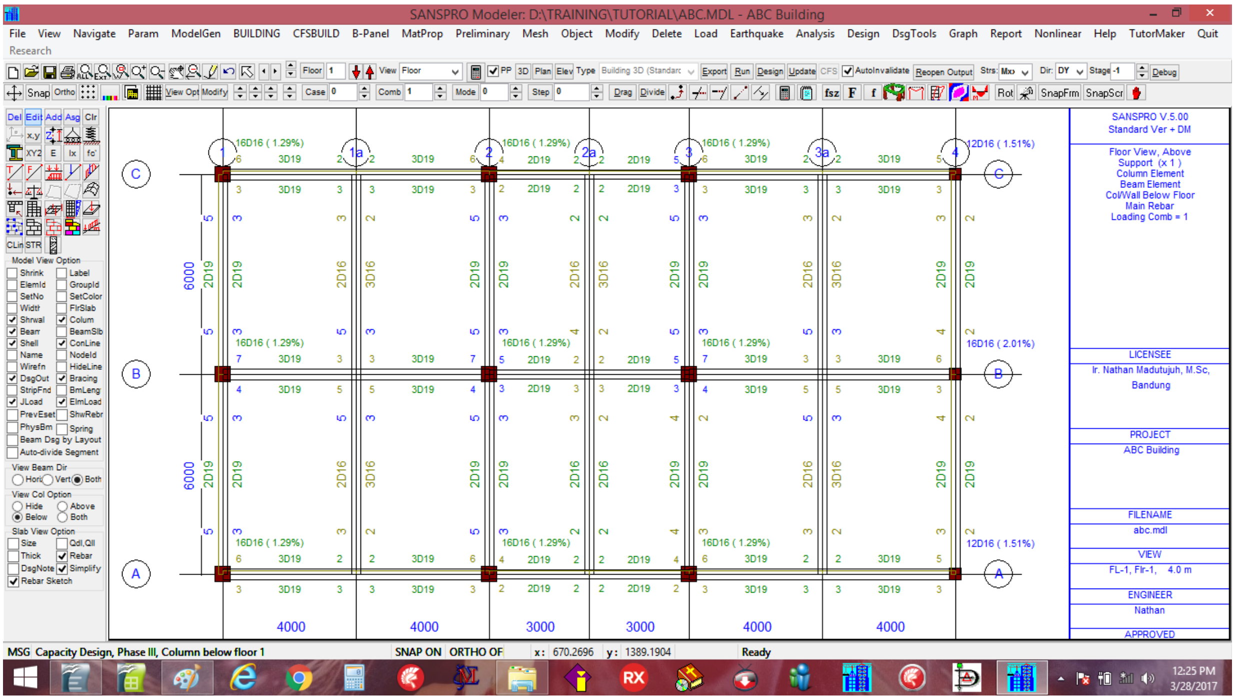
Click **[Ok]** to start Concrete Capacity Design, click **[Ok]**
For Beams : Click **[Yes]**, click **[Ok]**, click **[Ok]**
For Columns : Click **[Yes]**, click **[Ok]**, click **[Ok]**
Click **[Ok]** to quit.

Now we can visually see and check concrete beam and column rebar easily as follows:

Right-click, select View Momen Design Results:



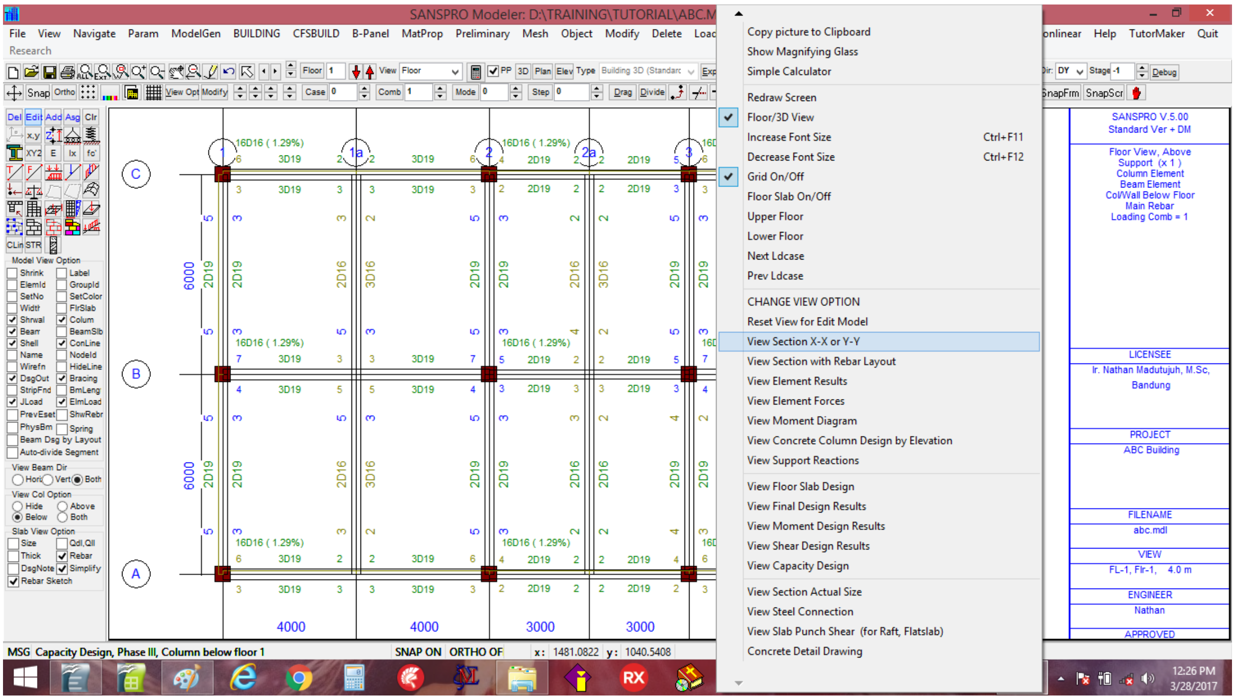
Concrete column rebar with percentage, concrete beam rebar will appear:



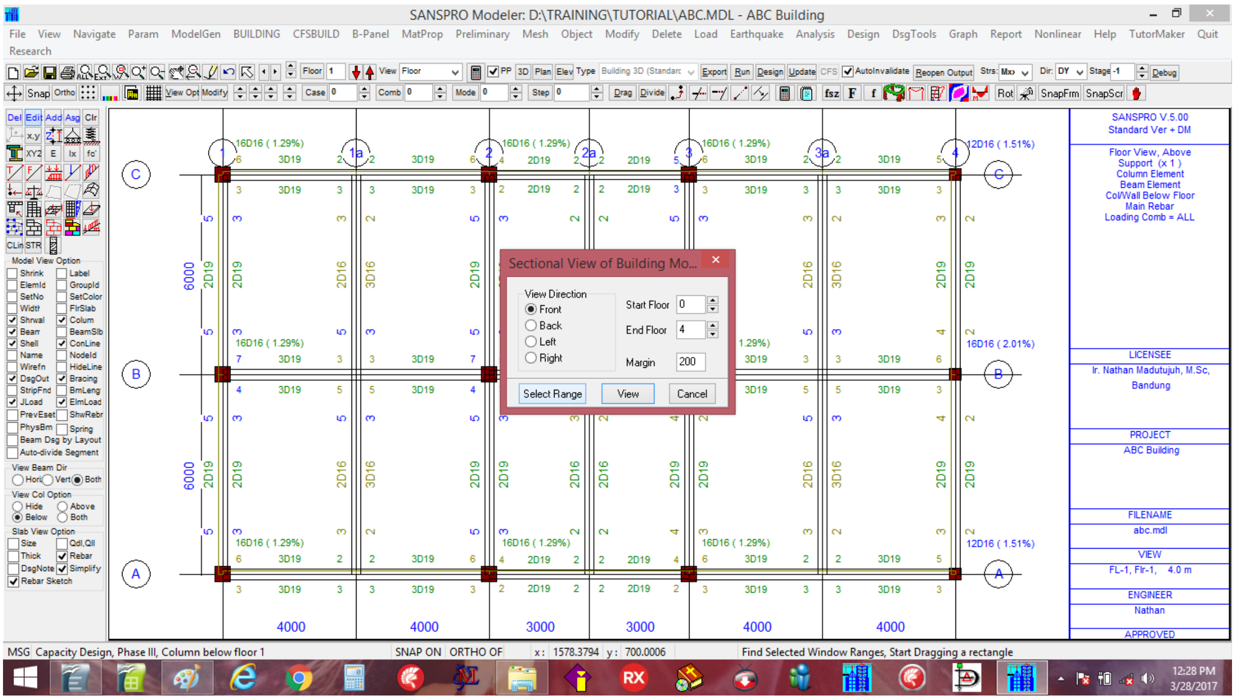
We can see rebar layout at other floor by moving to certain floor.

Rebar Layout Sectional View

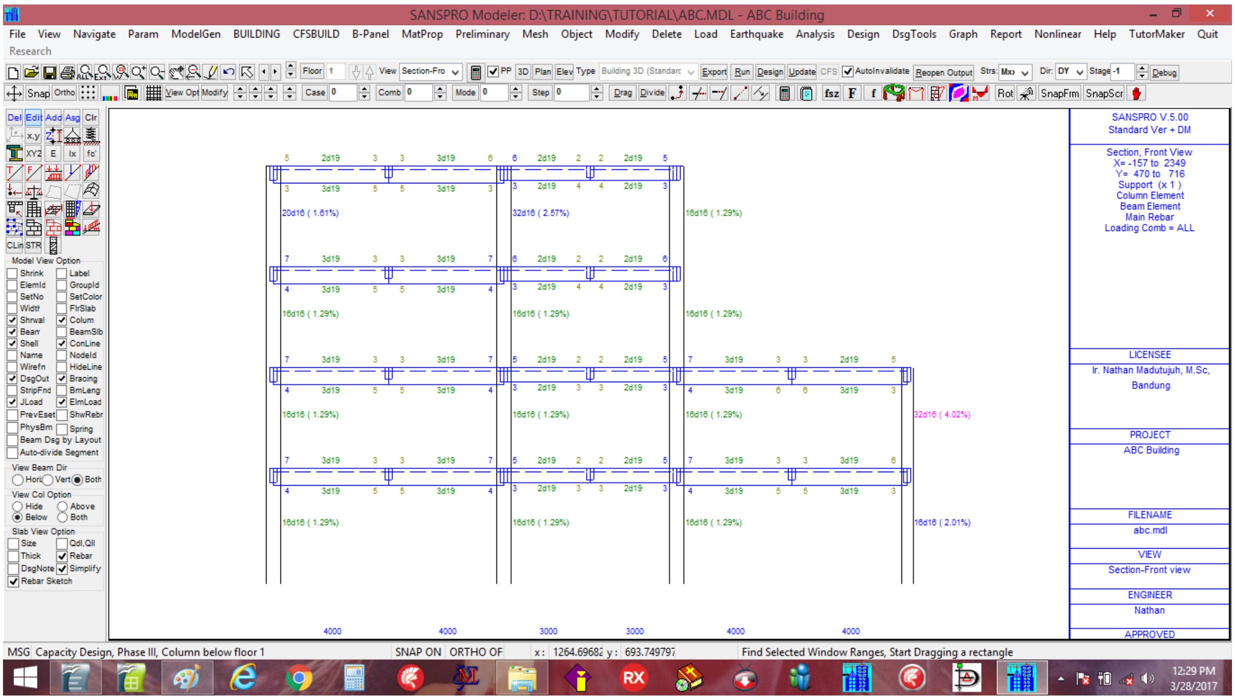
We can see also sectional view (side view) of the rebar detail by :
Right-Click, select View Section XX or YY



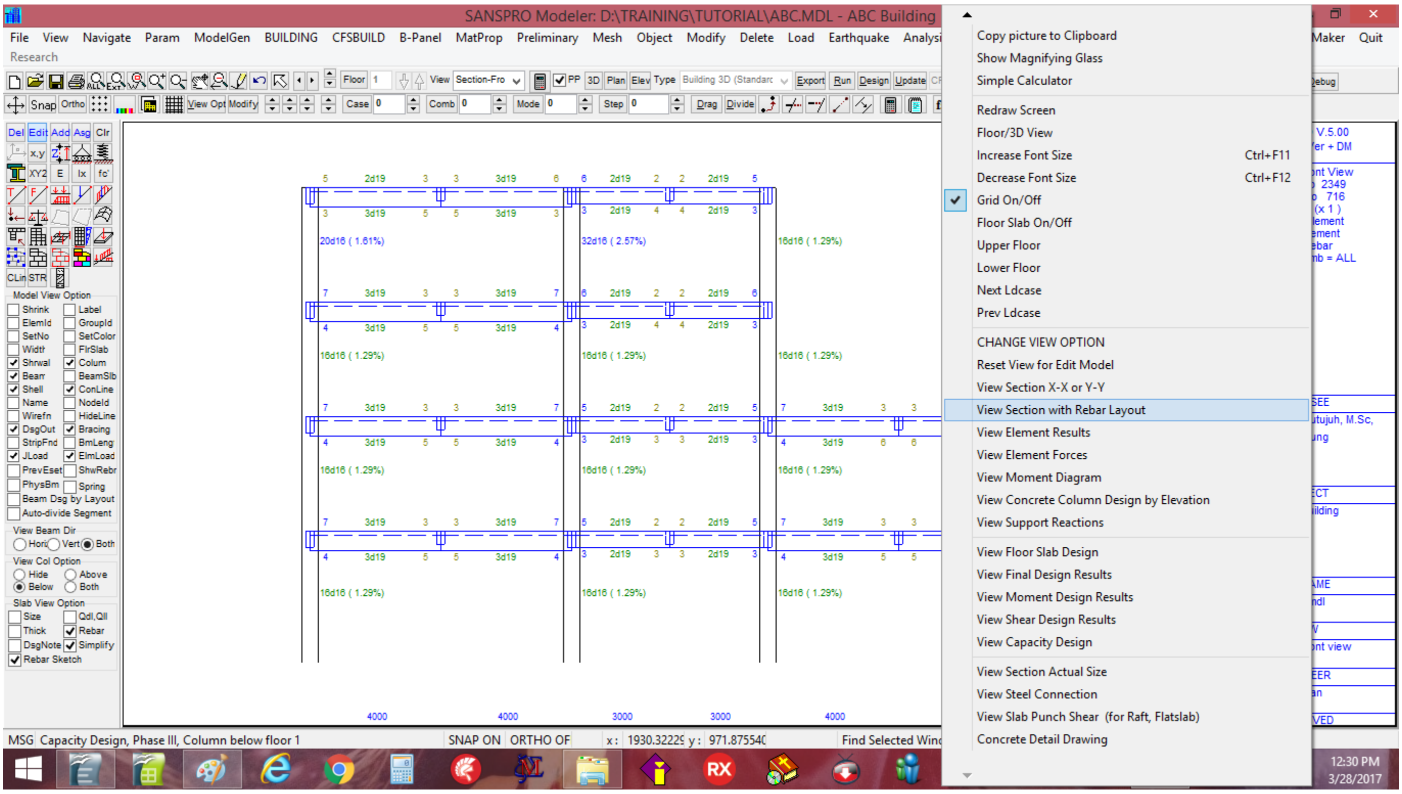
Click [Select Range], and drag a window range of sectional view:



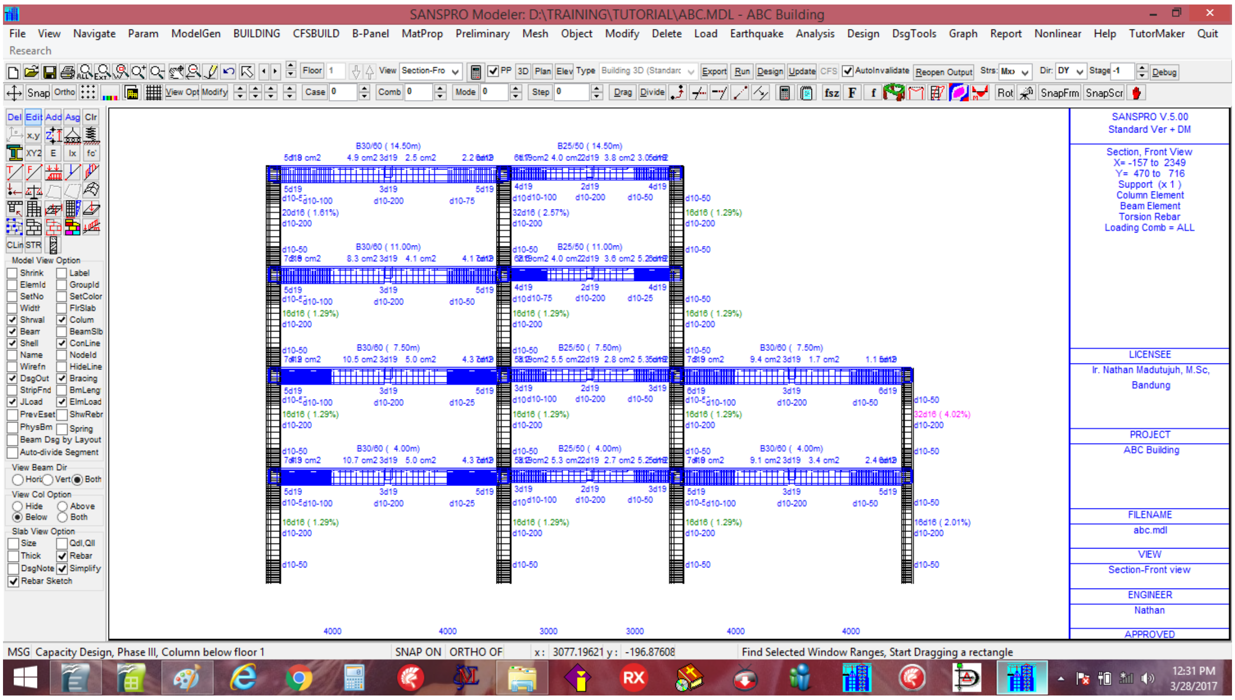
Click [View], now the rebar will be shown in the selected sectional plane.



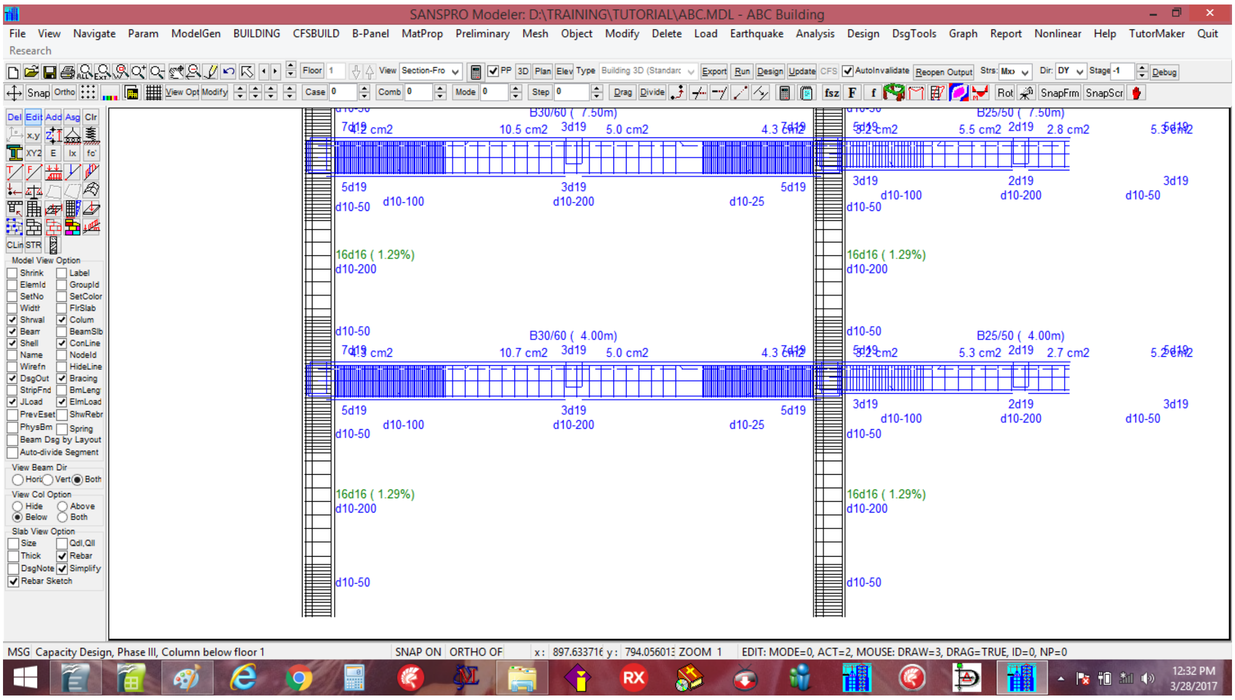
We can see also detail rebar using this sectional view by Right-Click, select View Section with Rebar Layout to get the following View:



View Section with Rebar Layout :



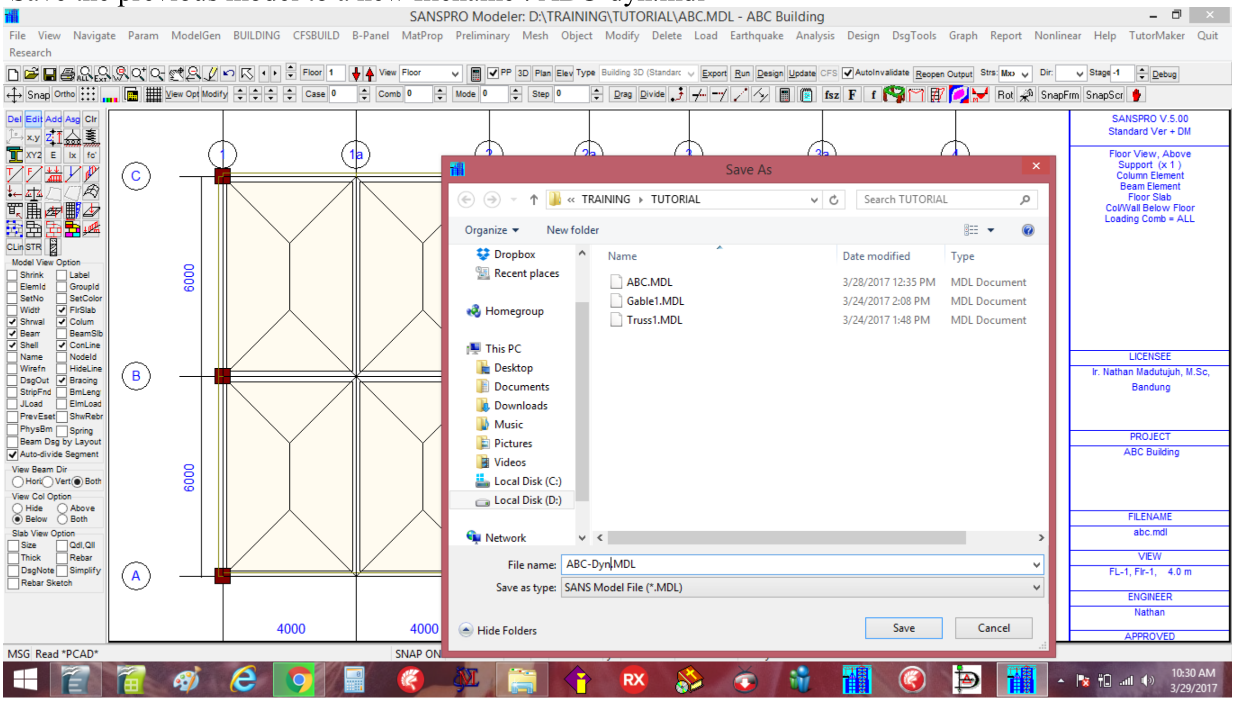
Zoom and enlarge a certain area to see more detail drawing:



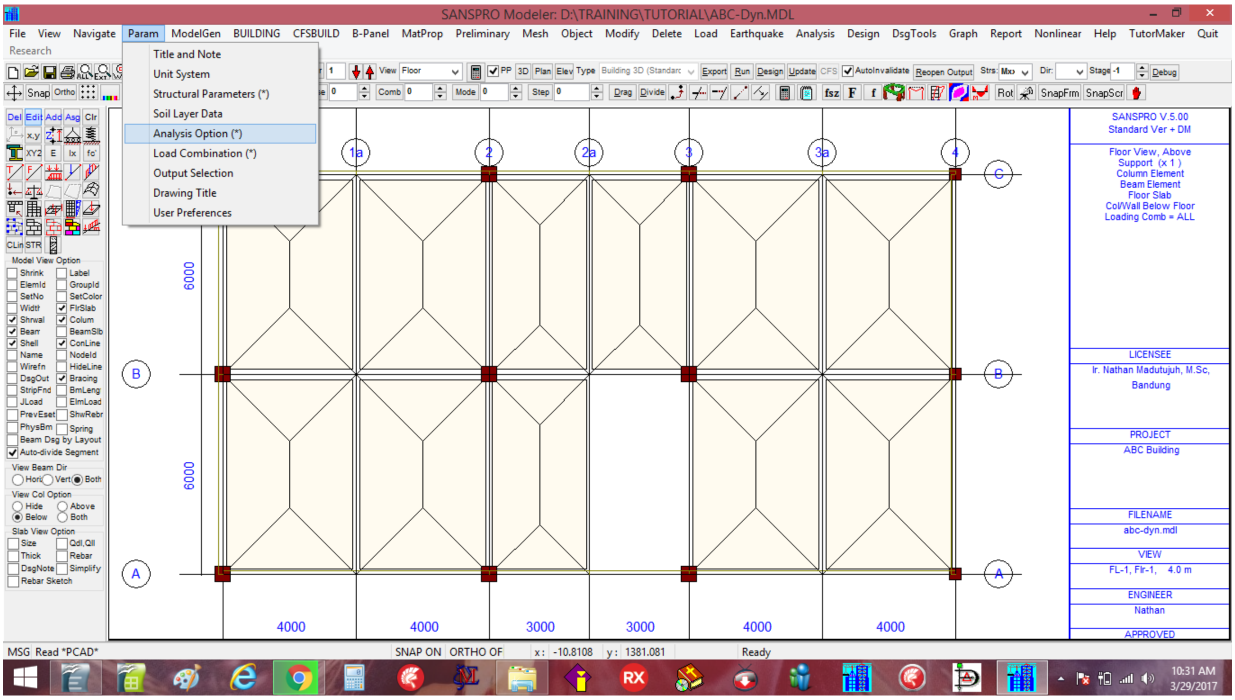
2. Example 4: Building Design (Gravity and Dynamic Earthquake Load Analysis)

We will now repeat the whole process again using Dynamic Analysis.

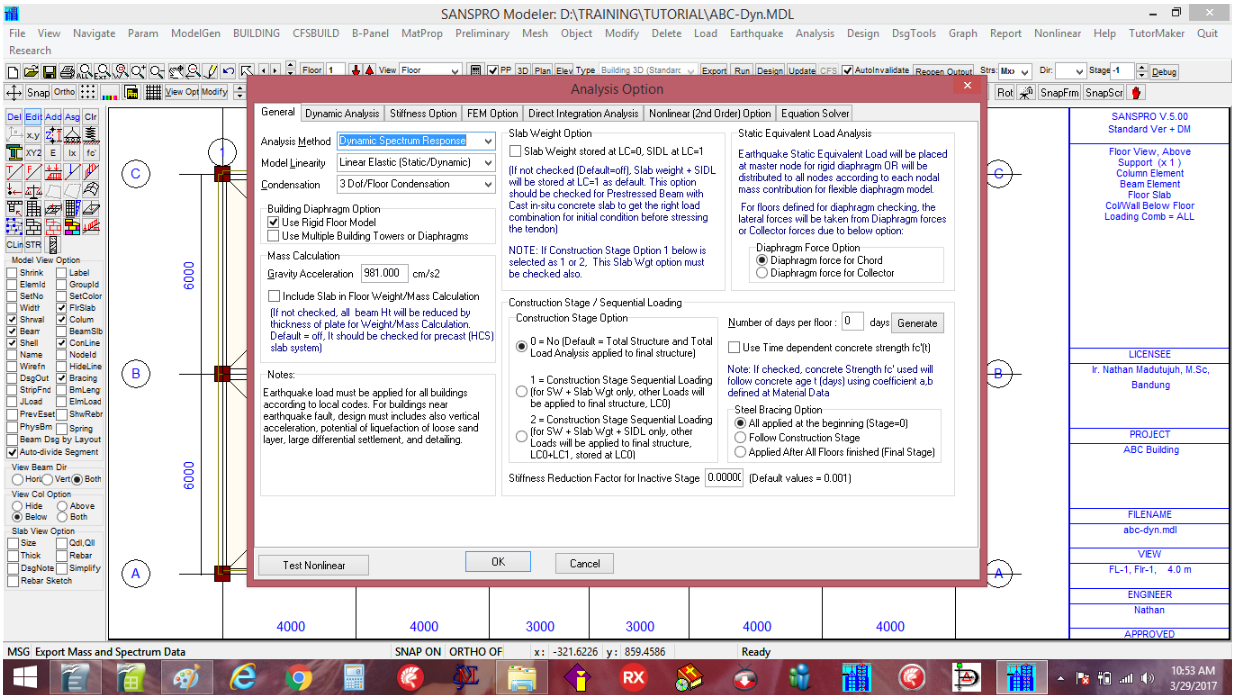
- Save the previous model to a new filename : ABC-dyn.mdl



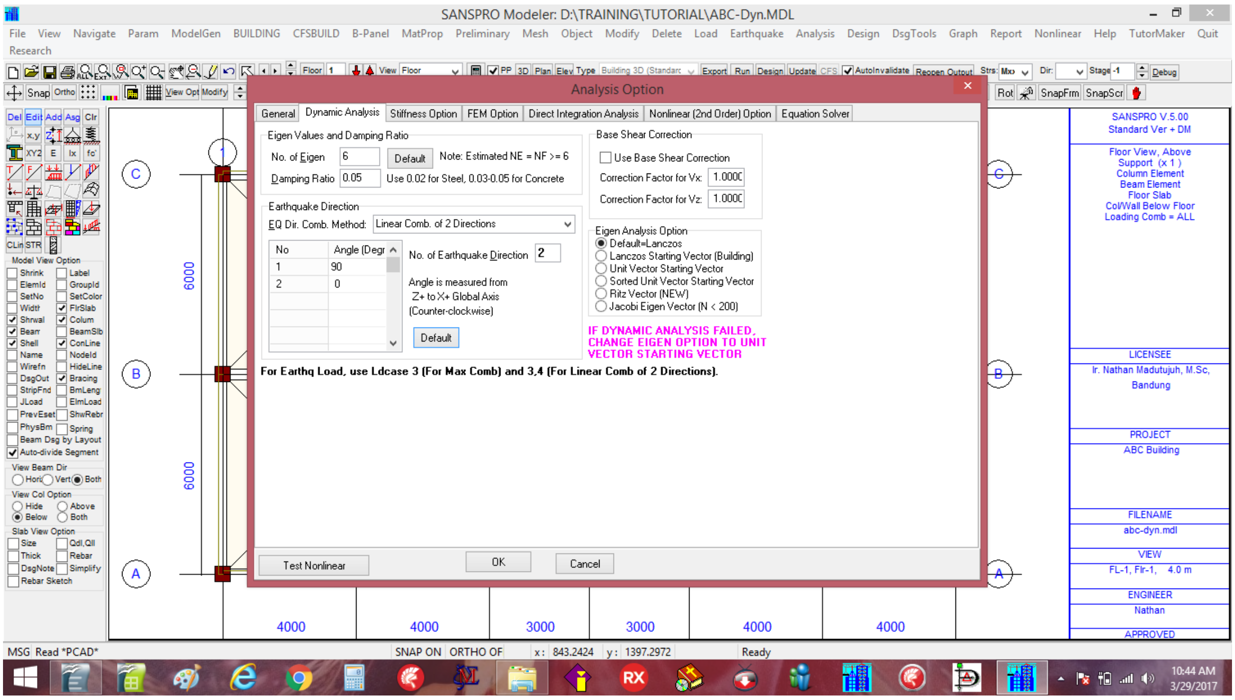
- We will now change some parameters needed for Dynamic Analysis:
- Click menu Analysis Option



Change **Analysis Method** to **Dynamic Analysis**

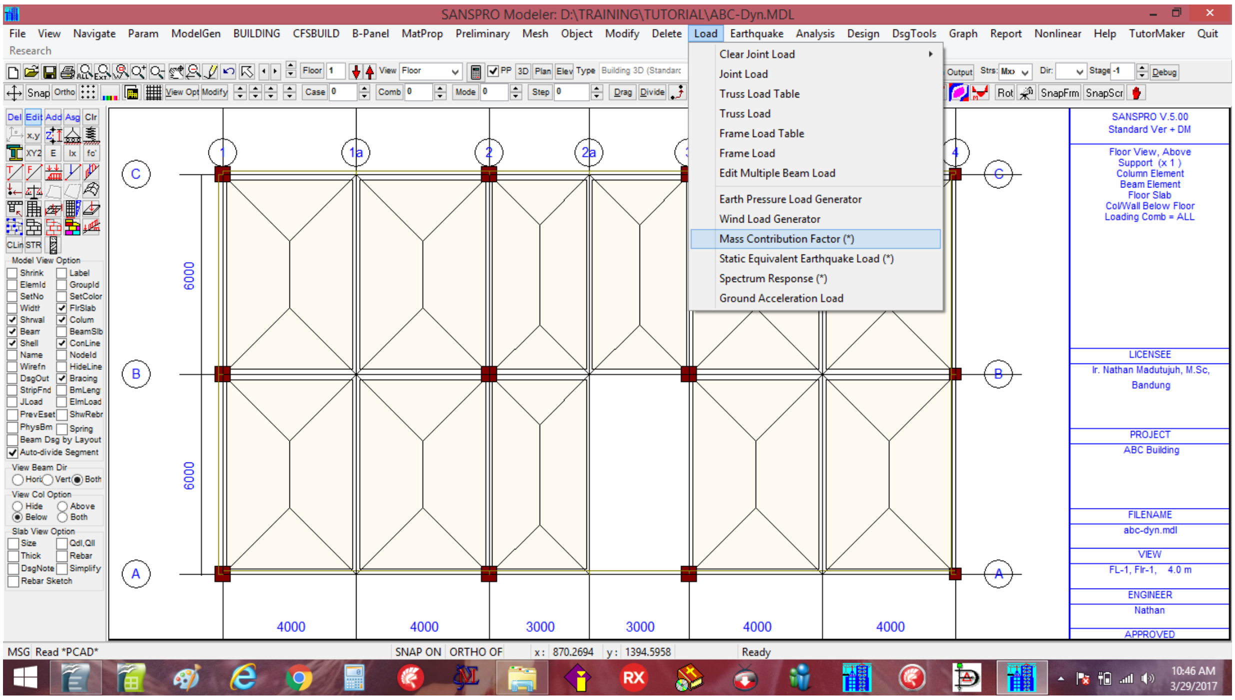


- Go to page : Dynamic Analysis
- click button [Default] and button [Default]
- The Dynamic Analysis page will look as follows:

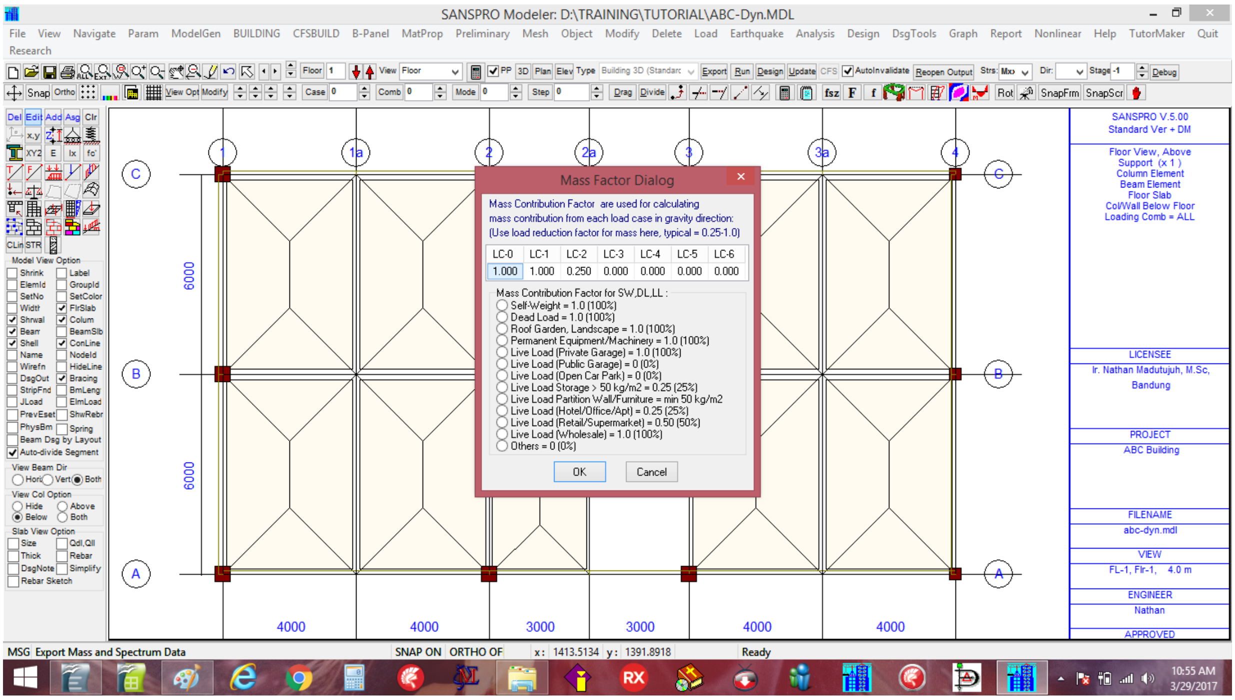


Here we will use Neigen = 6, Damping = 0.05 (for concrete), and Linear comb of 2 directions at 90 and 0 deg. Click [OK] to quit.

Next, we need to define mass from 100% of SW+DL and 25% of LL as follows:
Click menu Load – Mass Contribution Factor



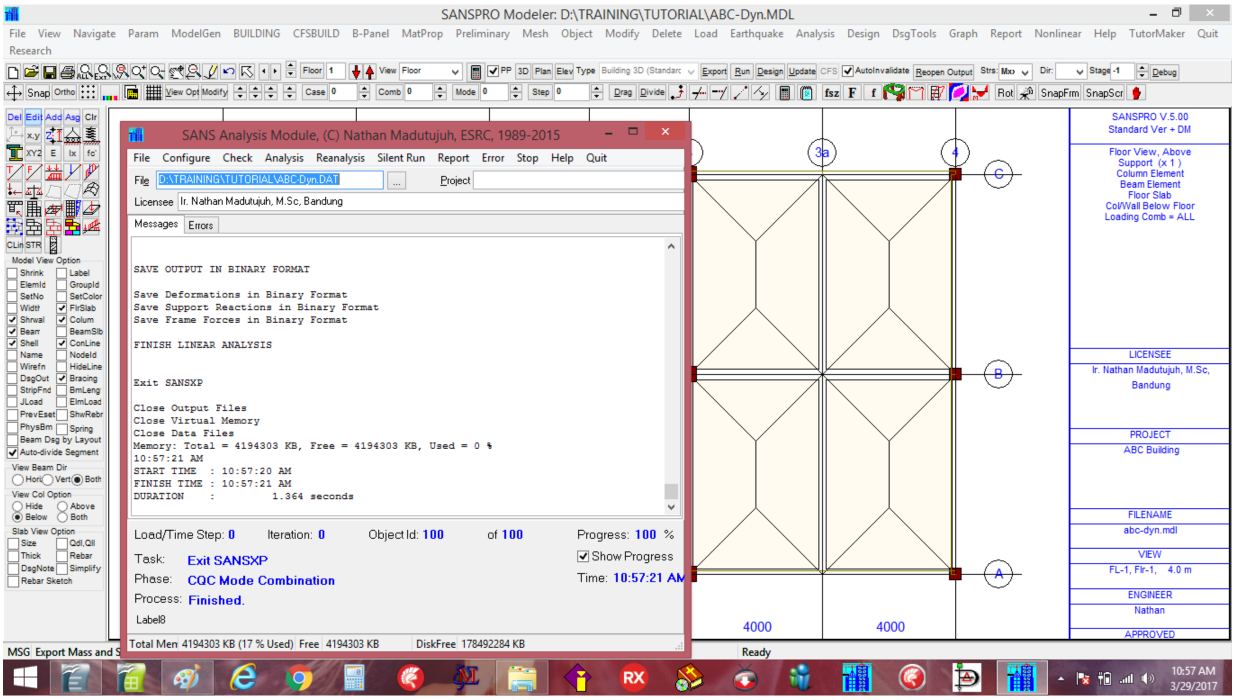
Enter data : **LC0 = 1.0, LC1 = 1.0, LC2 = 0.25** *(for 100% of SW+DL and 25% of LL)*, then click **[Ok]** :



We need to run the analysis again, now using Dynamic Analysis Method:

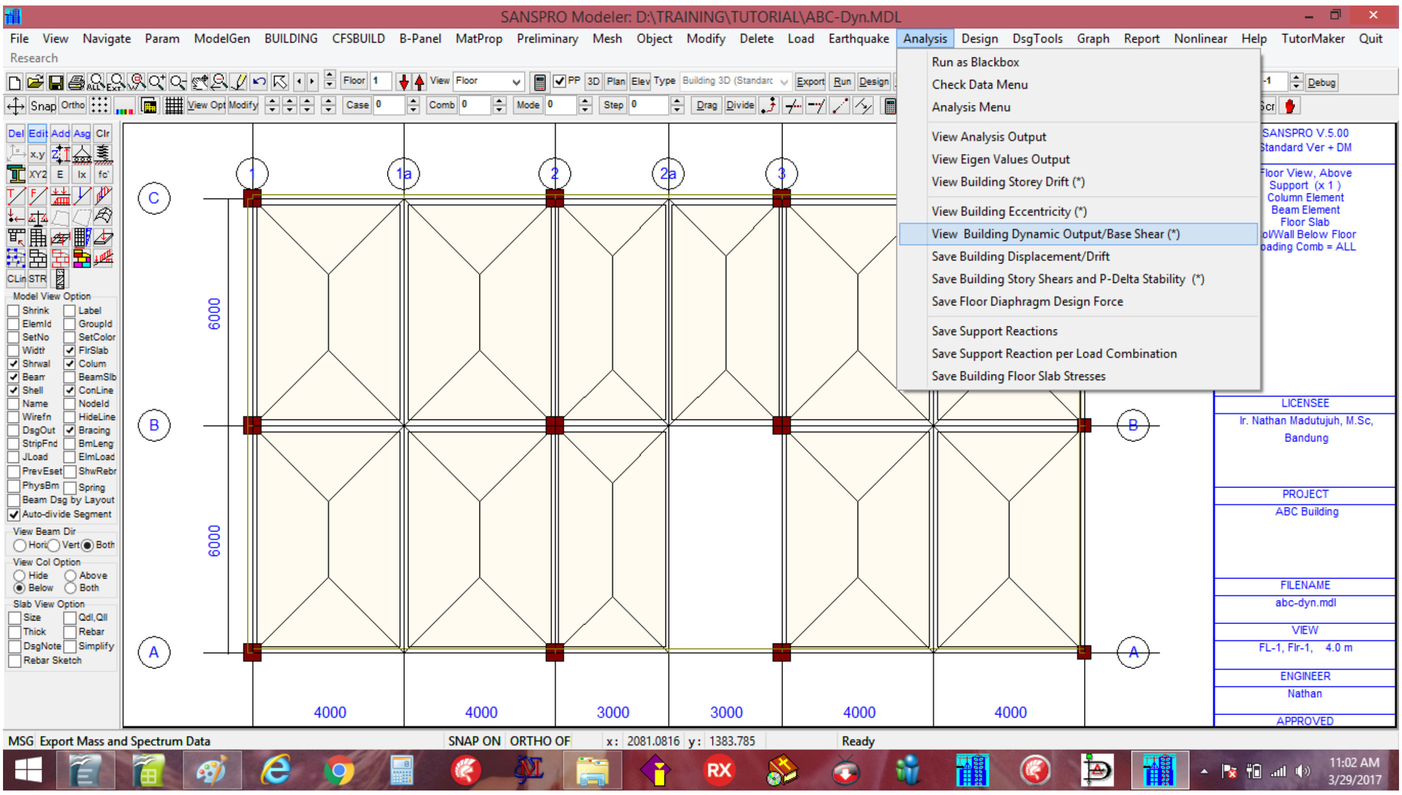
- Click **F2** to save the model
- Click **F4** to export the model for running analysis
- Click menu **Analysis** – click **Analysis Menu** – click **Analysis**
- SANSPRO will finish the dynamic analysis in about 1.364 secs.

- After Dynamic Analysis:



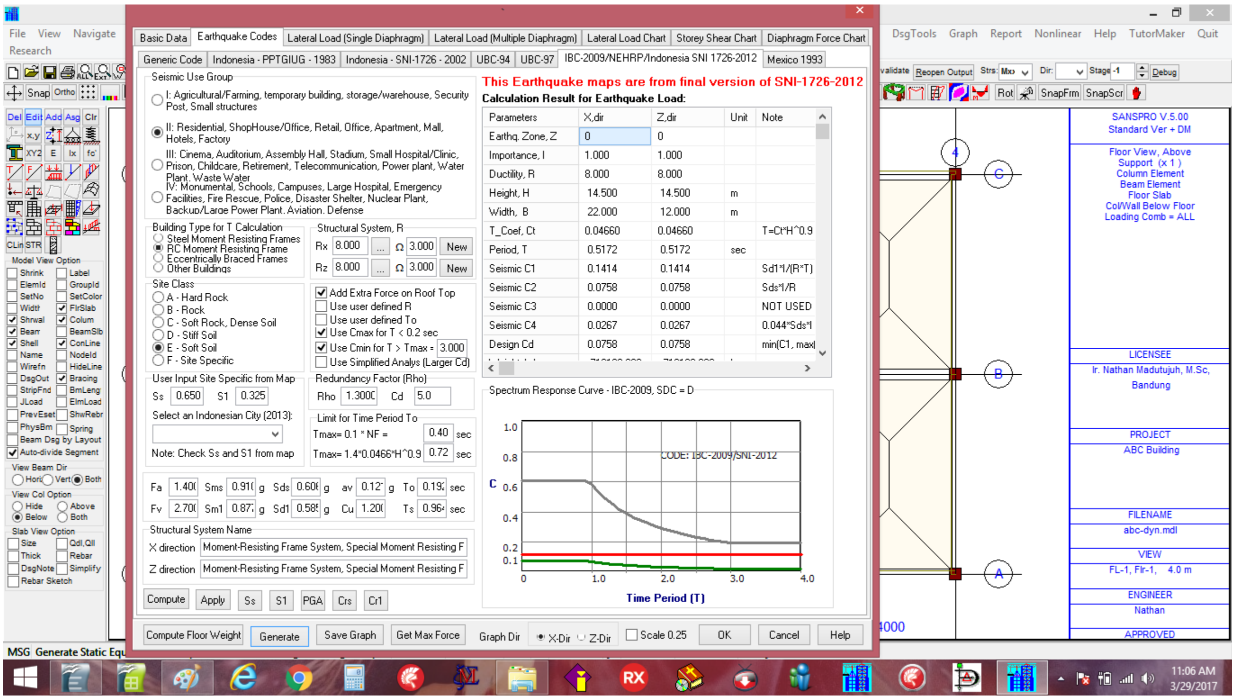
Before we continue with Concrete Capacity Design, Results from Dynamic Analysis must be verified for code requirements as follows:

To1 <= Tmax (see Tmax from **Earthquake** menu, applicable for NF >= 8)
Modal direction of first 2 modes: DX, DZ
Effective Mass >= 90%
Base Shear Vd/Vs >= 0.85

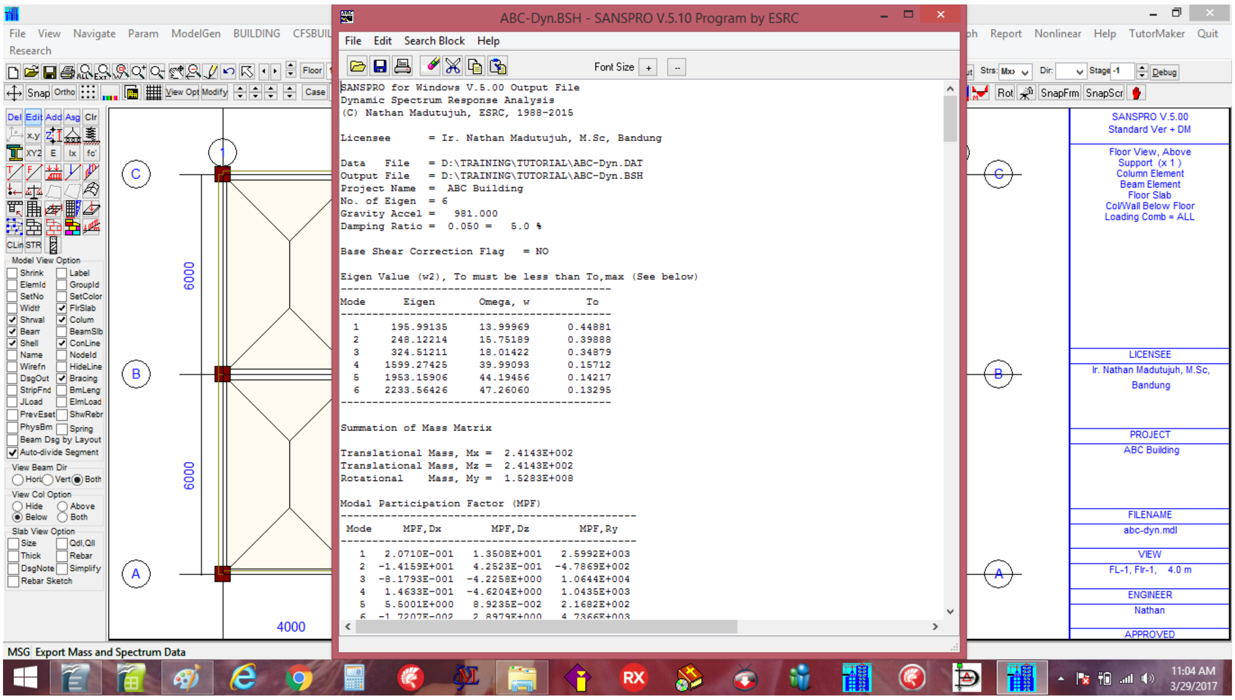


– Check the content of file *.bsh as follows:

– Checking To1 <= 0.72 sec (recommended to keep To1 <= 0.1 to 0.15*NF in secs) :



– Click menu **Analysis – View Building Dynamic Output/Base Shear**
– To1 = 0.45 secs <= Tmax = 0.72 secs → OK



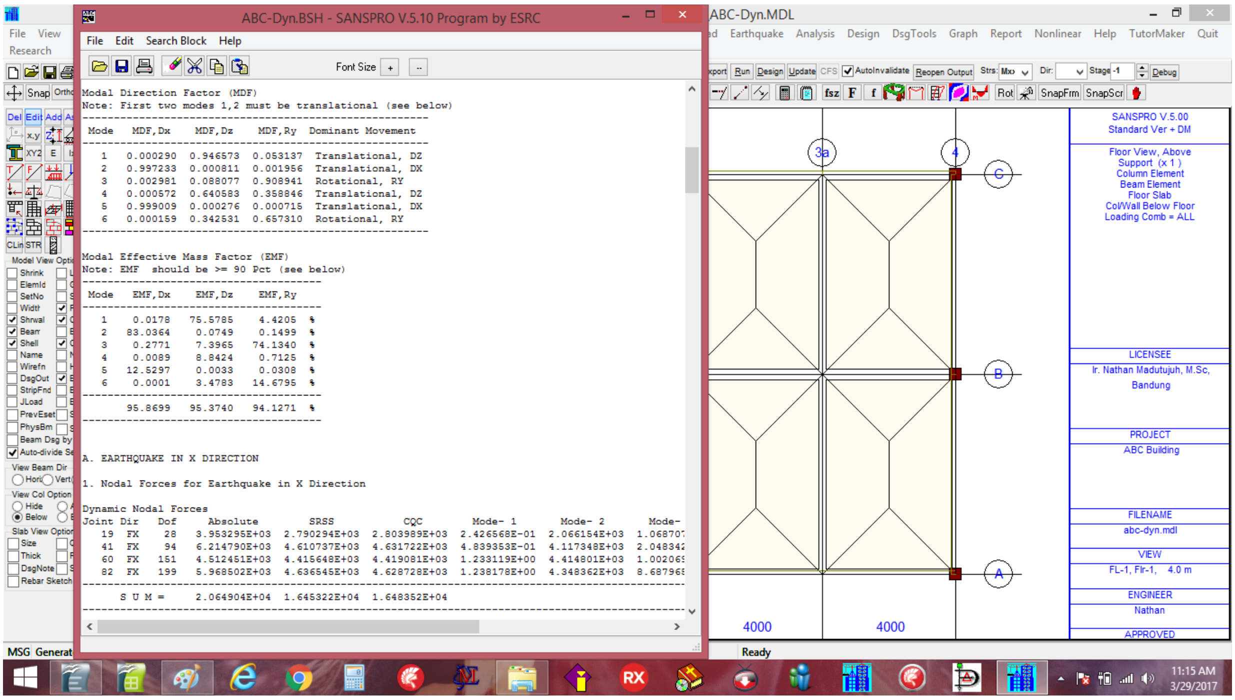
NOTE :

If To is too large, building is too flexible, there will be large drift and also earthquake load generated will be too small.

If To > Tmax, we can correct the model as follows:

– Reduce mass using LL reduction factor if possible (*in this case Llrf = 0.25 is already used*)
– Increase concrete strength fc' → to increase Ec
– Increase column size or shearwall thickness
– Increase beam height (*if beam height is less than L/14*)
– Decrease floor height → to increase column stiffness

Checking Modal direction of first 2 modes: DX, DZ



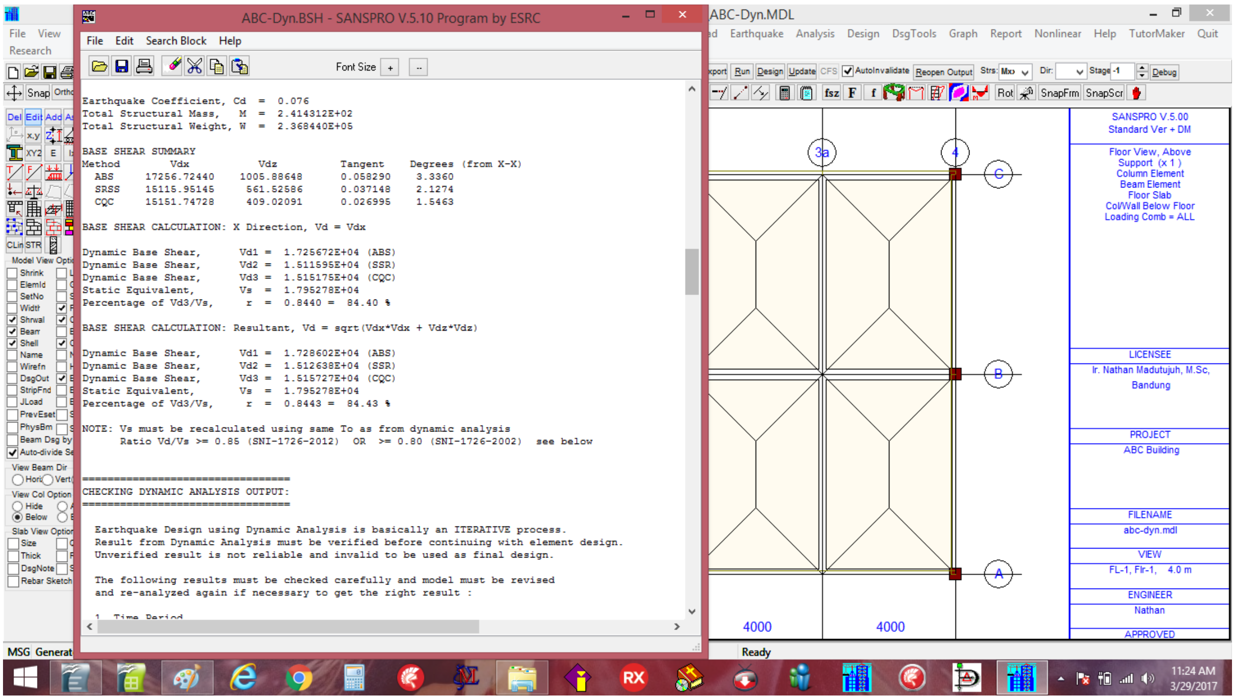
From above report: Mode 1 = Translational in DZ, Mode 2 = Translational in DX → OK
If not satisfied, building may experience torsion vibration, column size must be re-arrange so that the dominant movement will be in translation direction.

Checking for Effective Mass Factor >= 90%
From above results, the Sum of Modal Effective Mass Factor is 94.12% > 90% → OK

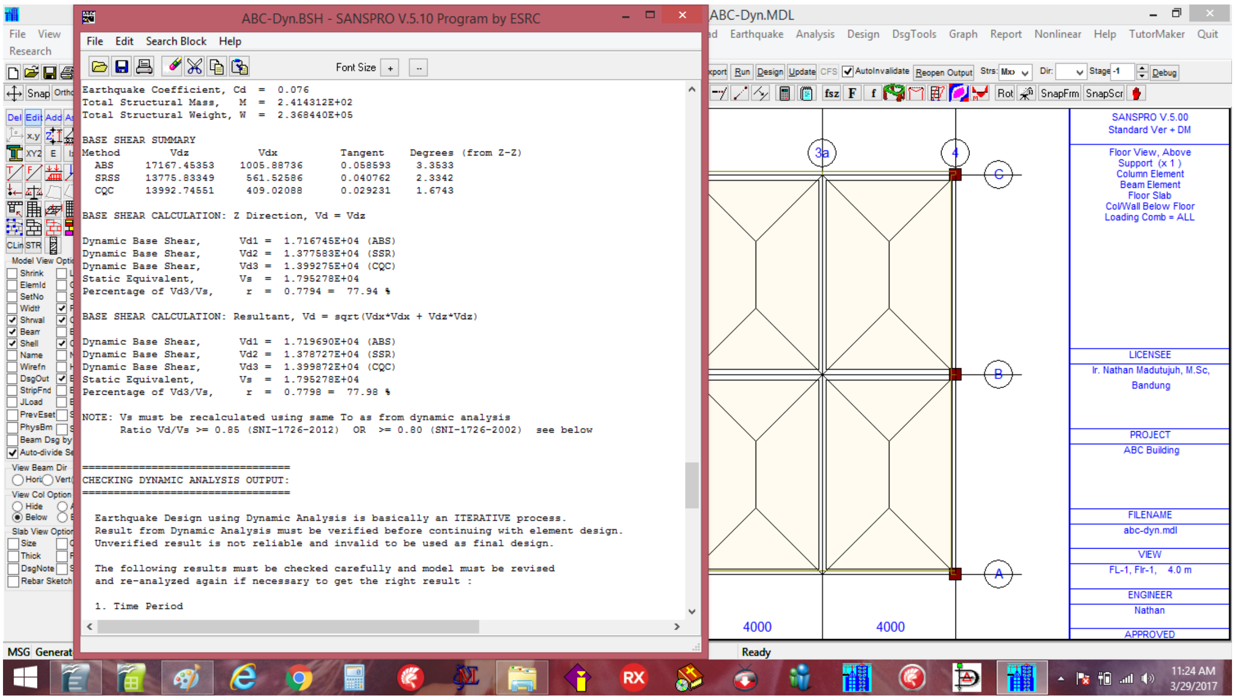
if not satisfied, the number of mode involved in dynamic analysis is not enough. The number of eigen must be increased to fulfill the requirement, but not more than NF*3/2 for small buildings and NF for large building (NF = number of story).

To ensure that the dynamic analysis will absorb same earthquake energy as in static equivalent, the base shear from dynamic analysis is limited to >= 0.85 * Vbs,static for X and Z direction.

Base Shear Vdx/Vsx = 84.4% <= 85%



Base Shear Vdz/Vsz = 77.9 % <= 85%



In this case, both base shear are less than the minimum required.
We can do some revision of the model here:

- Add more eigen values (at Analysis Option) OR
- Use Scaling Factor (at Analysis Option)

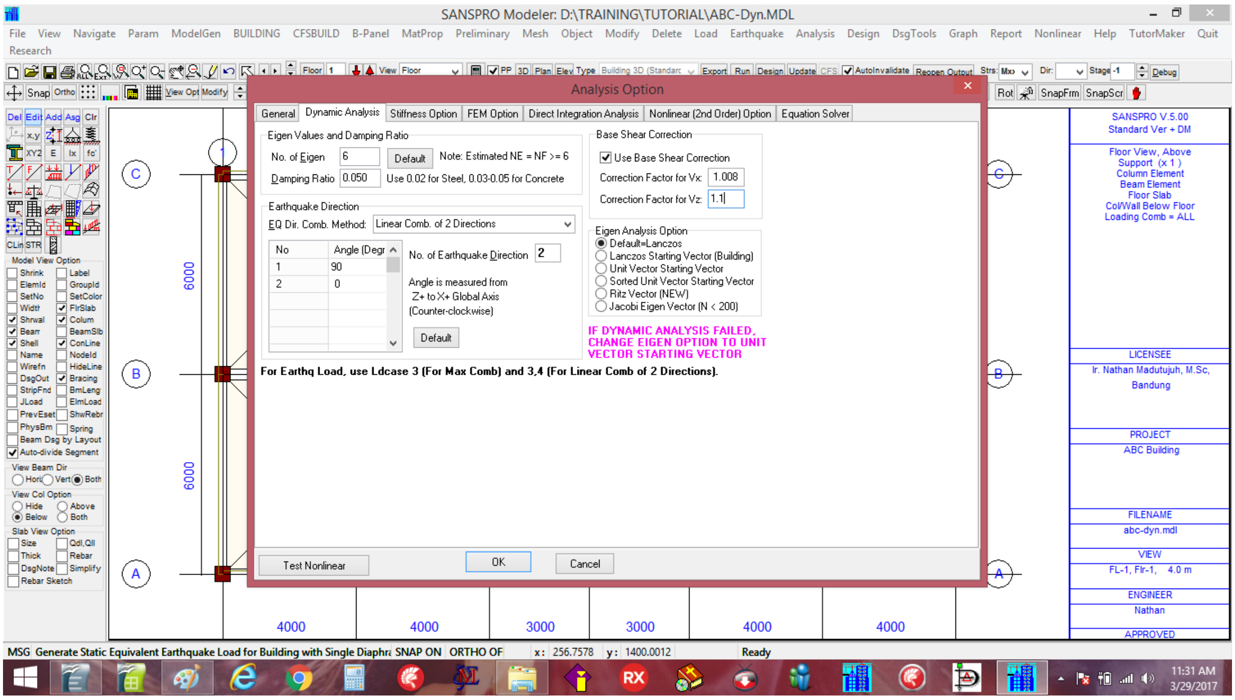
Because we have only 4 floors, we will use second method, using Scaling Factor as follows:

For X Direction: FS,X = 85 / 84.4 = 1.0071 → Use FS,X = 1.008
For Z Direction : FS,Z = 85 / 77.9 = 1.0911 → Use FS,Z = 1.10

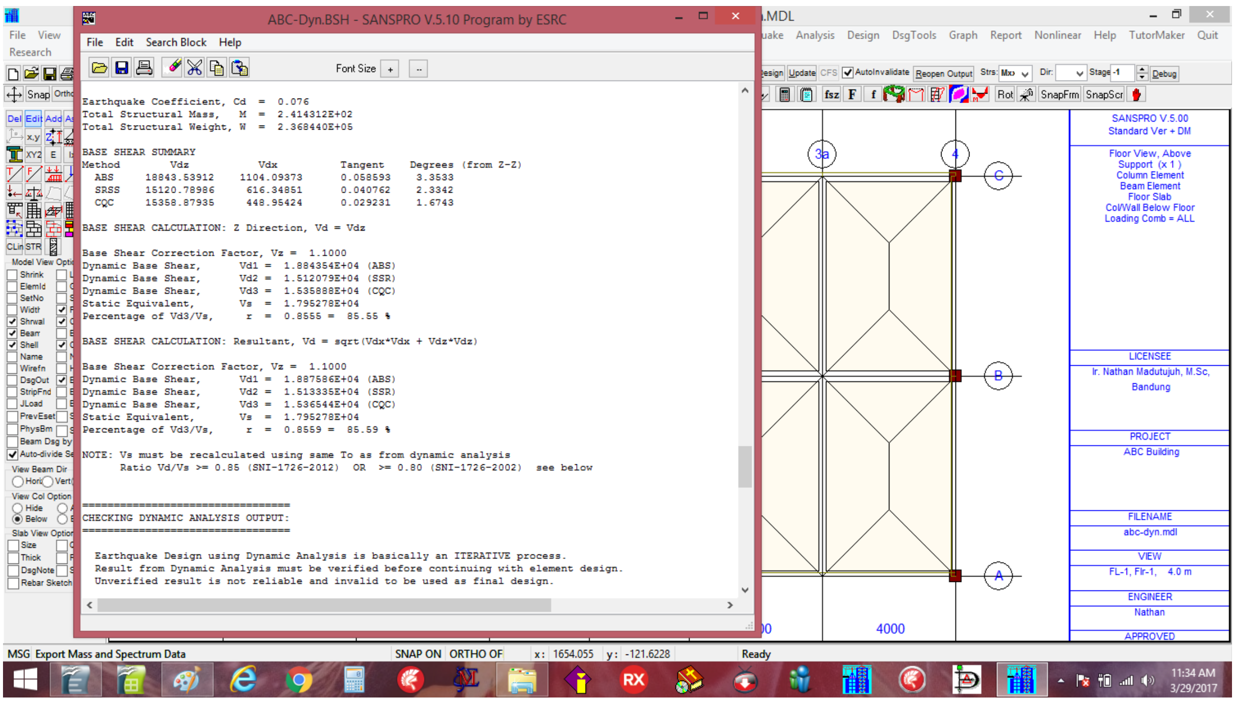
We can enter the Scaling factor at Analysis Option and run the analysis again.
After running, the base shear should be >= 85%.

Entering the Base Shear Scaling Factor:

- Click **[x] Use Base Shear Correction**
- Enter : Correction Factor for Vx = **1.008**
- Enter : Correction Factor for Vx = **1.1**
- Click **[Ok]**
- Click **F2** and **F4**
- Run Analysis again



The Base Shear Vx and Vz now are >= 85% of Vbsh,static.

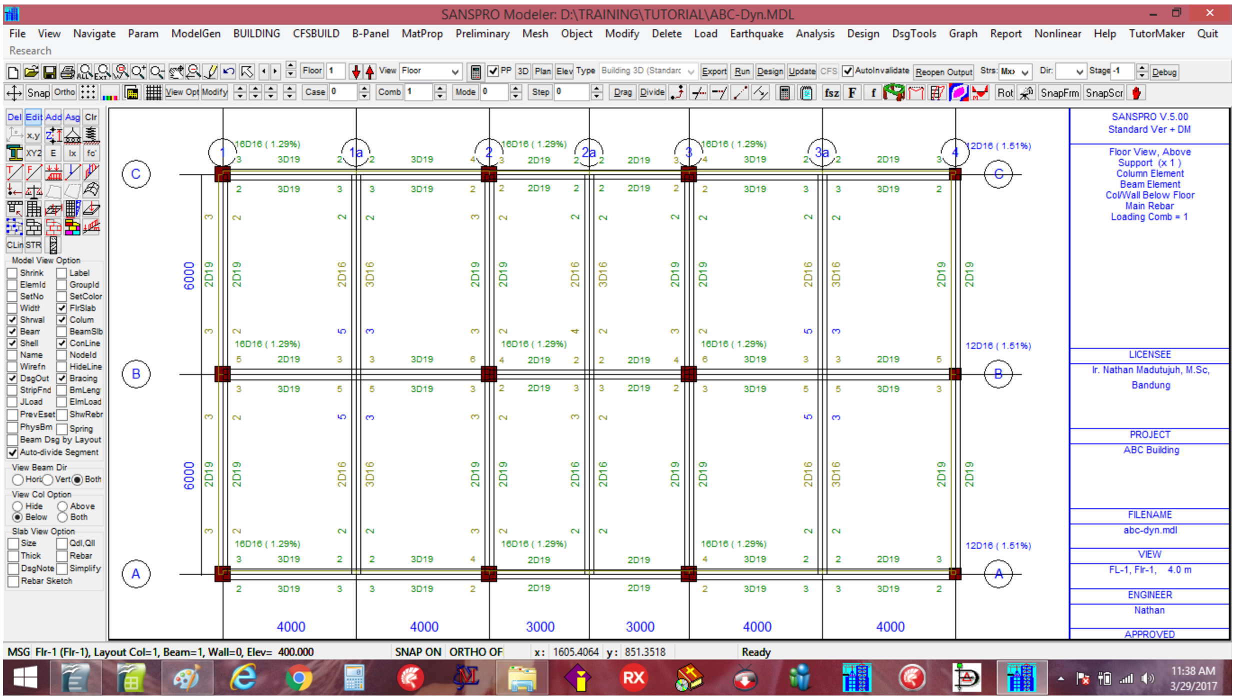


The dynamic analysis is now in good correlation with static analysis and satisfying the code requirements.

Now we can run the Concrete Capacity Design as before.

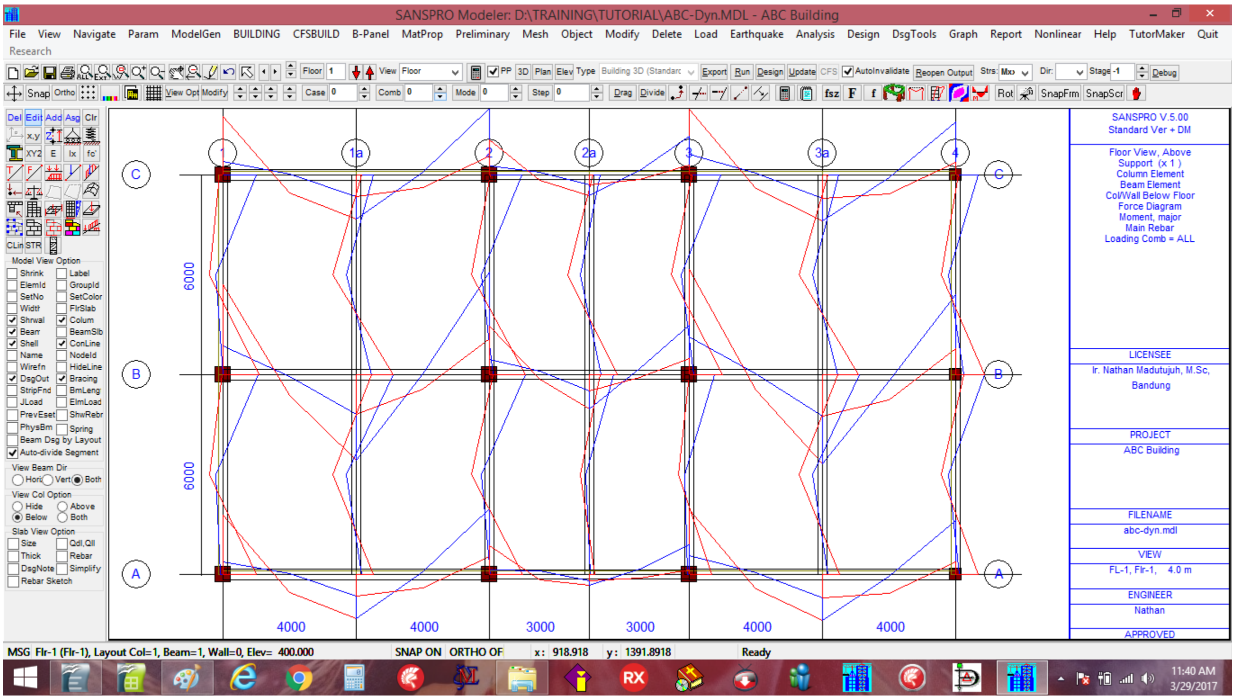
Click menu **Design – Run Concrete Capacity** Design (With Messages)
After the design process, we can visually check the concrete rebar by **Right-Click, View Moment Design**.

View Concrete Rebar (after Dynamic Analysis)



Typically, because the base shear involved is only 85% of static base shear, the concrete rebar shown may be a little bit smaller than from static analysis. But because the lateral earthquake load distribution of dynamic analysis is more accurate than of static analysis, at mid floors the rebar required may be larger.

View Moment Diagram (after Dynamic Analysis)



NOTE:

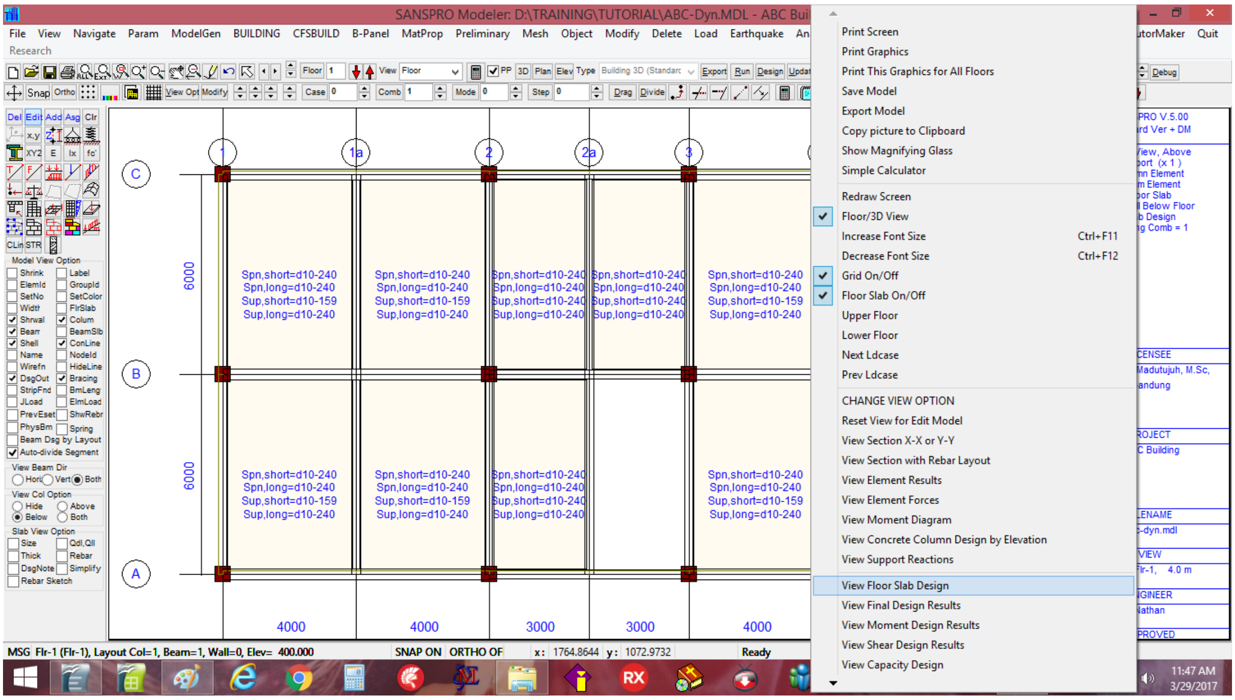
Please note that using Dynamic Spectrum Response analysis will give all + sign for element forces (because of the Square root or CQC combination method), so the moment diagram when combined with results from static load must be taken carefully.

3. Example 5: Concrete Floor Slab Design

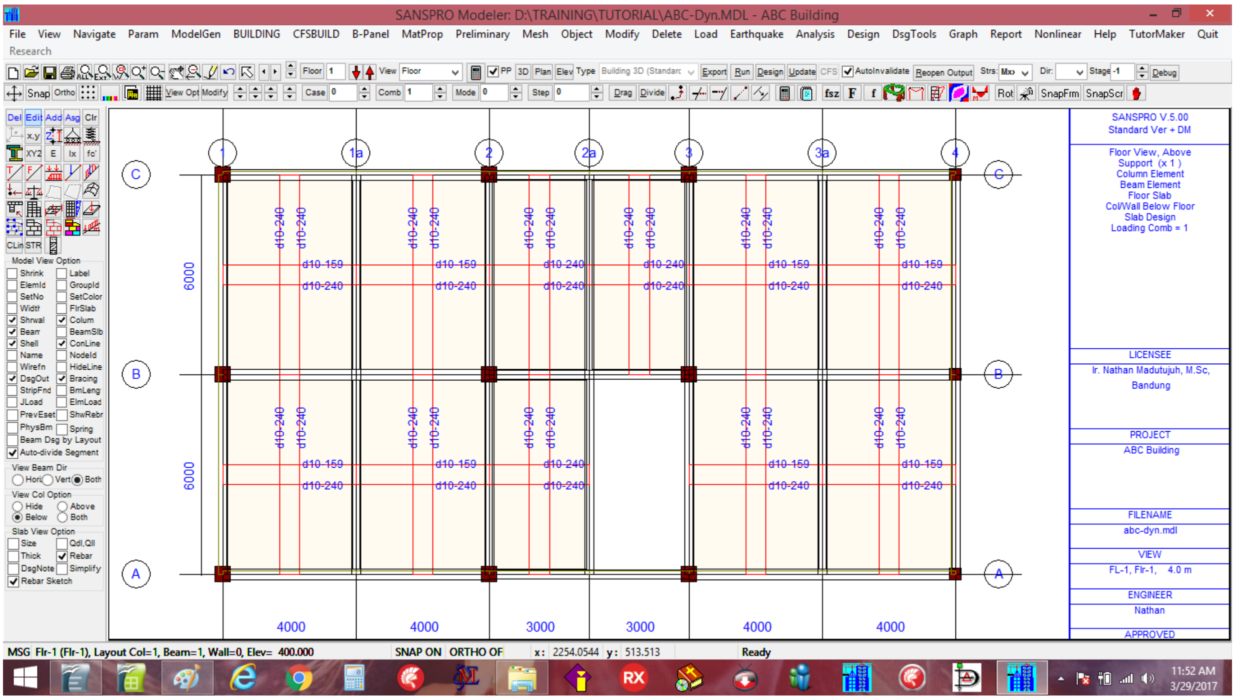
SANSPRO provides many design option for concrete slab:

- Concrete slab design moment (*using plate moment coefficient table*)
- Concrete slab rebar design (*using doubly reinforced concrete beam section*)
- Simplification of concrete slab rebar arrangement (using bent or additional rebar)
- Human induced Vibration analysis for slab
- Short-term and long-term displacement of slab

To design a floor slab, go to the floor level, and **Right-Click**, select **View Floor Slab Design**



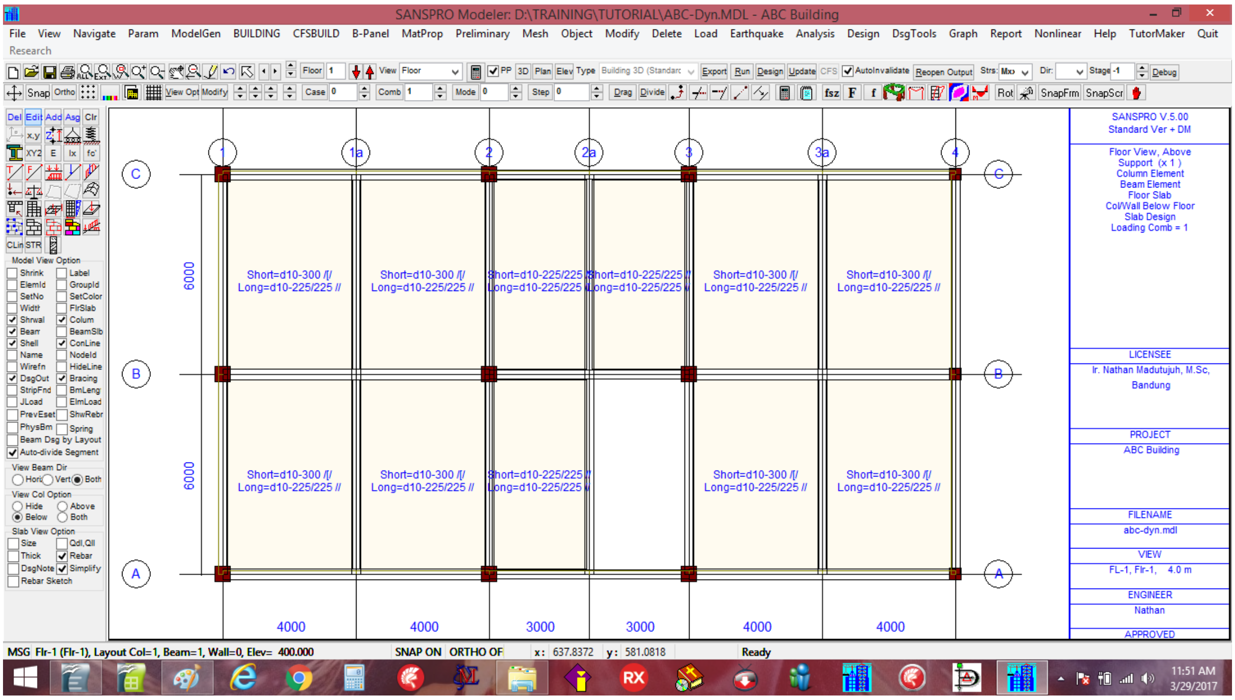
Concrete slab rebar (for M+ midspan and M- support) and for short and long span will be shown. Click bottom-left icon **[x] Rebar, [x] Rebar Sketch** to see the concrete slab rebar



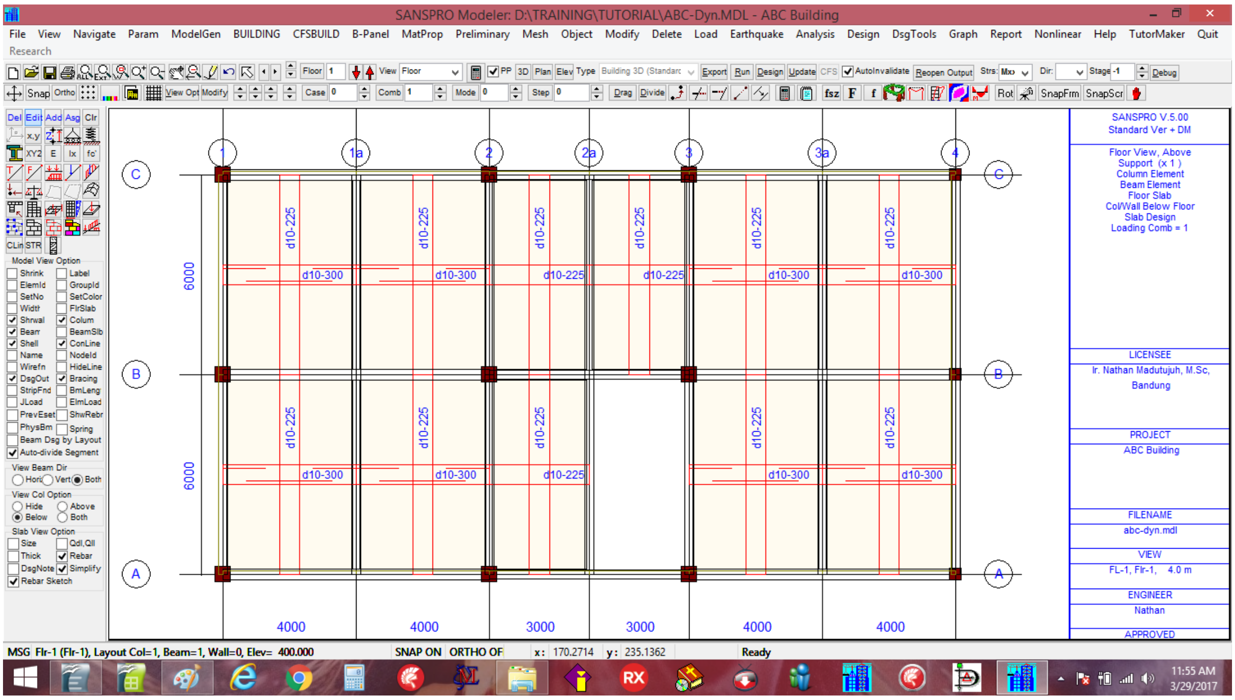
The program will show continuous rebar layout that satisfy the above slab design moment required.

User can click icon **[x] Simplify** to further simplifying the rebar layout to get more economical solution using bent rebar or non-continuous additional rebar as follows:

Required rebar spacing after simplifying:



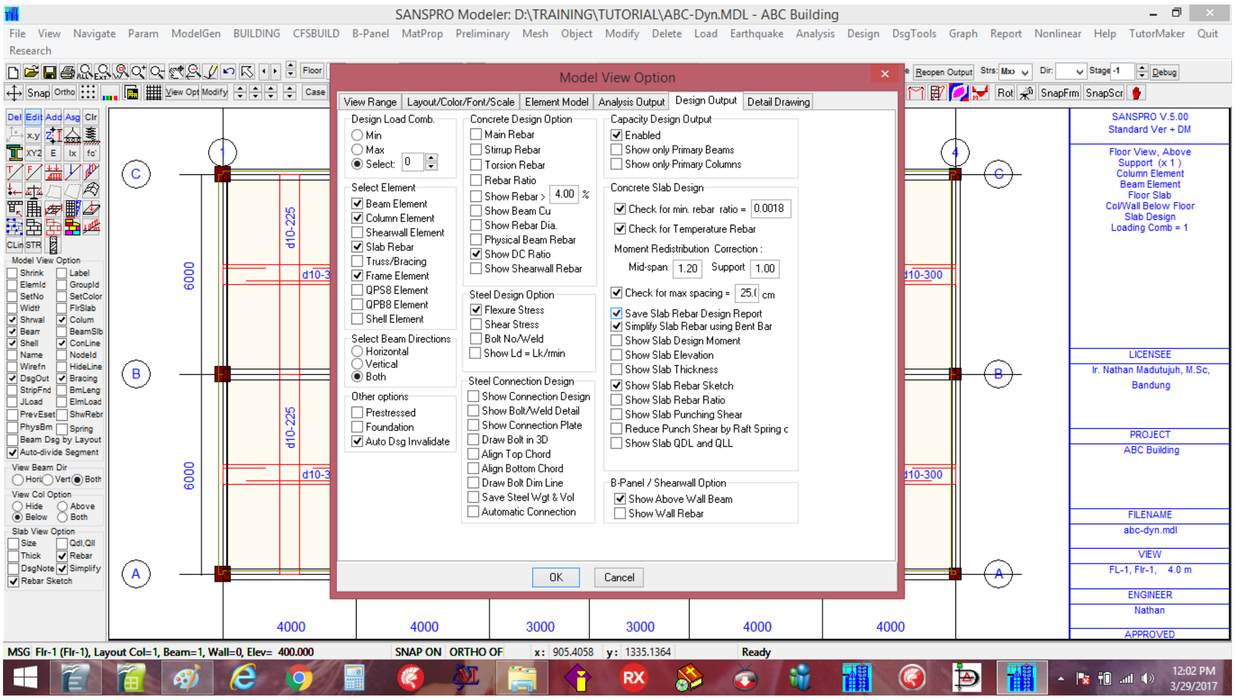
The program will try to simplify the rebar arrangement as follows:



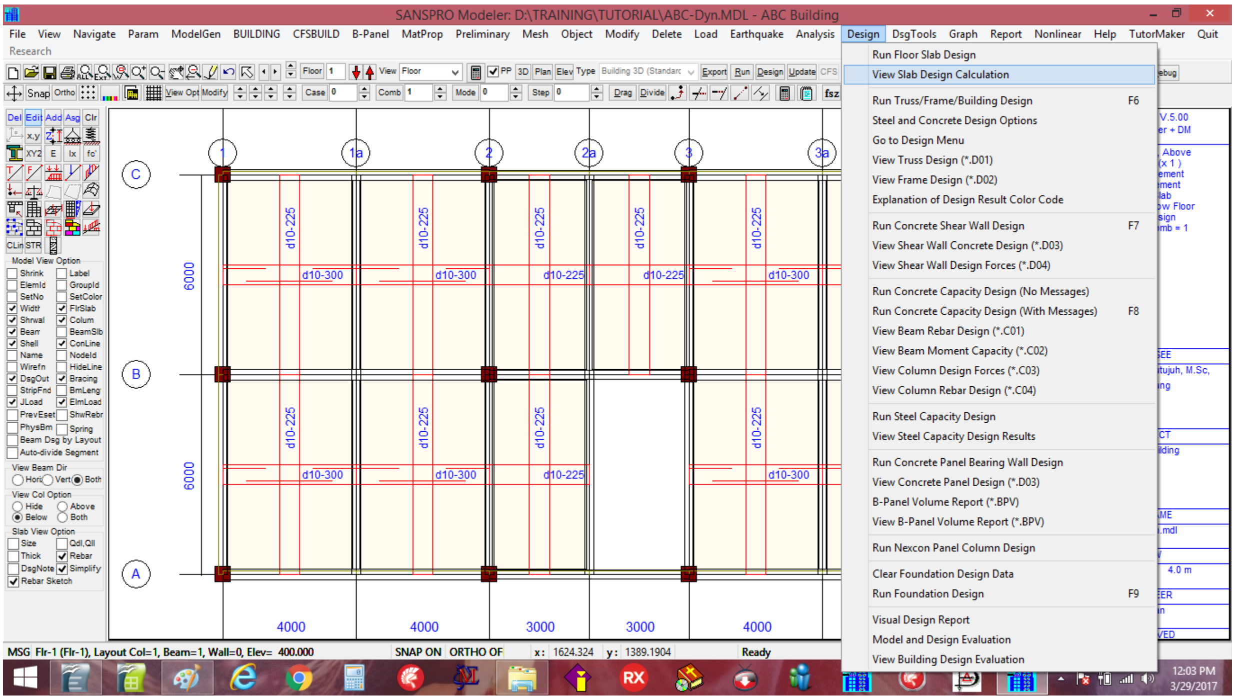
NOTE:

- Max recommended main rebar spacing for concrete slab is 2*tp
- For Metal deck, the rebar shown is the additional rebar needed
- For Hollow Core Slab, the design result is not applicable, because prestressed wire is used instead
- For Half-slab system, add more M+ rebar due to its behavior during construction load (the slab will behave as simple beam before casting the topping)
- Minimum slab thickness should be about L/30 – L/35
- Concrete cover at top and bottom of concrete slab must be <= 1.5 to 2.0 cm to avoid Mn reduction

Slab can be analyzed for vibration and deflection as follows:
- Go to the floor slab to be checked
- **Right-Click**, select **Change View Option**
- click [x] Save Slab Rebar Design Report
- click **[Ok]**



click menu Design – View Slab Design Calculation

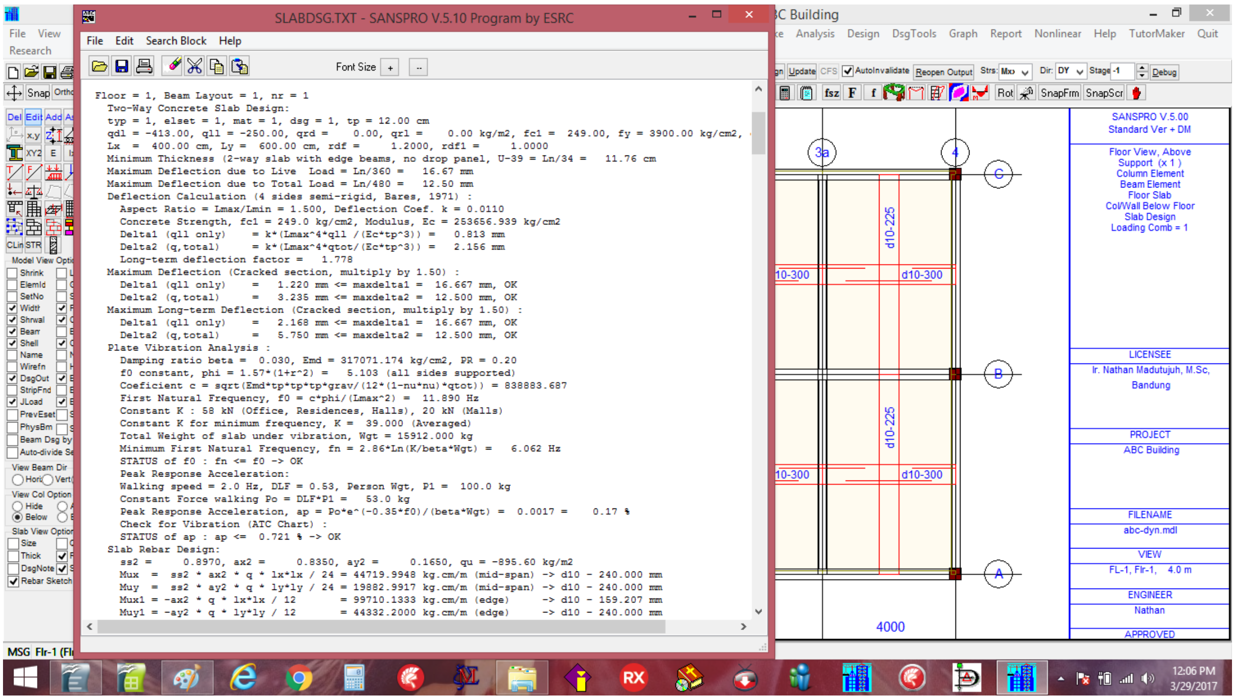


A file named SLABDSG.TXT will be generated for this floor.
The content of this file can be incorporated into final report.

NOTE:

– This option is automatically set to OFF after first click, to speed up the program.
– Repeat again above procedure for other floors.

Slab Design Calculation Report:



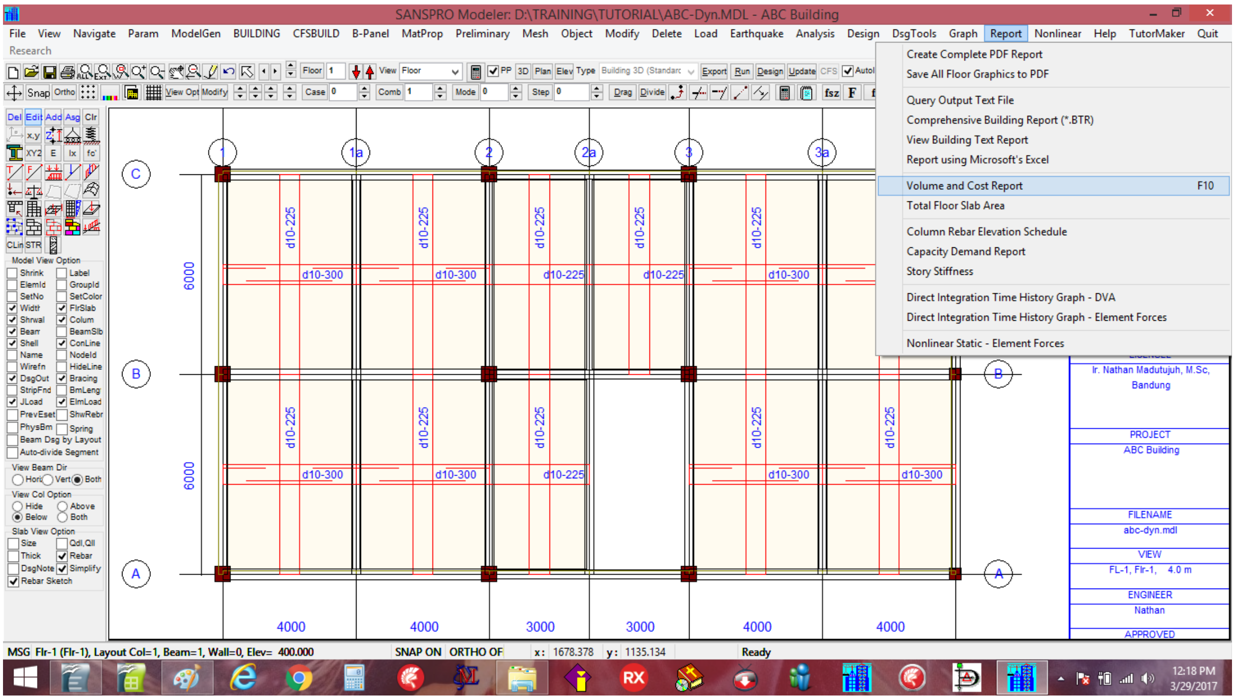
Report for one slab region is as follows:

```
Floor = 1, Beam Layout = 1, nr = 1
  Two-Way Concrete Slab Design:
  typ = 1, elset = 1, mat = 1, dsg = 1, tp = 12.00 cm
  qdl = -413.00, qll = -250.00, qrd = 0.00, qrl = 0.00 kg/m2, fc1 =  249, fy = 3900 kg/cm2, db = 10.00 mm
  Lx  =  400.00 cm, Ly =  600.00 cm, rdf =      1.2000, rdf1 =     1.0000
  Minimum Thickness (2-way slab with edge beams, no drop panel, U-39 = Ln/34 =   11.76 cm
  Maximum Deflection due to Live  Load = Ln/360 =   16.67 mm
  Maximum Deflection due to Total Load = Ln/480 =   12.50 mm

  Deflection Calculation (4 sides semi-rigid, Bares, 1971) :
    Aspect Ratio = Lmax/Lmin = 1.500, Deflection Coef. k = 0.0110
    Concrete Strength, fc1 = 249.0 kg/cm2, Modulus, Ec = 253656.939 kg/cm2
    Delta1 (qll only)    = k*(Lmax^4*qll /(Ec*tp^3)) =   0.813 mm
    Delta2 (q,total)     = k*(Lmax^4*qtot/(Ec*tp^3)) =   2.156 mm
    Long-term deflection factor =   1.778

  Maximum Deflection (Cracked section, multiply by 1.50) :
    Delta1 (qll only)    =   1.220 mm <= maxdelta1 =  16.667 mm, OK
    Delta2 (q,total)     =   3.235 mm <= maxdelta2 =  12.500 mm, OK

  Maximum Long-term Deflection (Cracked section, multiply by 1.50) :
    Delta1 (qll only)    =   2.168 mm <= maxdelta1 =  16.667 mm, OK
    Delta2 (q,total)     =   5.750 mm <= maxdelta2 =  12.500 mm, OK

  Plate Vibration Analysis :
    Damping ratio beta =  0.030, Emd = 317071.174 kg/cm2, PR = 0.20
    f0 constant, phi = 1.57*(1+r^2) =   5.103 (all sides supported)
    Coeficient c = sqrt(Emd*tp*tp*tp*grav/(12*(1-nu*nu)*qtot)) = 838883.687
    First Natural Frequency, f0 = c*phi/(Lmax^2) =  11.890 Hz
    Constant K : 58 kN (Office, Residences, Halls), 20 kN (Malls)
    Constant K for minimum frequency, K =  39.000 (Averaged)
    Total Weight of slab under vibration, Wgt = 15912.000 kg
    Minimum First Natural Frequency, fn = 2.86*Ln(K/beta*Wgt) =   6.062 Hz
    STATUS of f0 : fn <= f0 -> OK

    Peak Response Acceleration:
    Walking speed = 2.0 Hz, DLF = 0.53, Person Wgt, P1 =  100.0 kg
    Constant Force walking Po = DLF*P1 =   53.0 kg
    Peak Response Acceleration, ap = Po*e^(-0.35*f0)/(beta*Wgt) =  0.0017 =    0.17 %

    Check for Vibration (ATC Chart) :
           STATUS of ap : ap <=  0.721 % -> OK

  Slab Rebar Design:
    ss2 =     0.8970, ax2 =     0.8350, ay2 =     0.1650, qu = -895.60 kg/m2
    Mux  =  ss2 * ax2 * q * lx*lx / 24 = 44719.9948 kg.cm/m (mid-span) -> d10 - 240.000 mm
    Muy  =  ss2 * ay2 * q * ly*ly / 24 = 19882.9917 kg.cm/m (mid-span) -> d10 - 240.000 mm
    Mux1 = -ax2 * q * lx*lx / 12       = 99710.1333 kg.cm/m (edge)     -> d10 - 159.207 mm
    Muy1 = -ay2 * q * ly*ly / 12       = 44332.2000 kg.cm/m (edge)     -> d10 - 240.000 mm

  Short=d10-300 /[/,  Long=d10-225/225 //
```

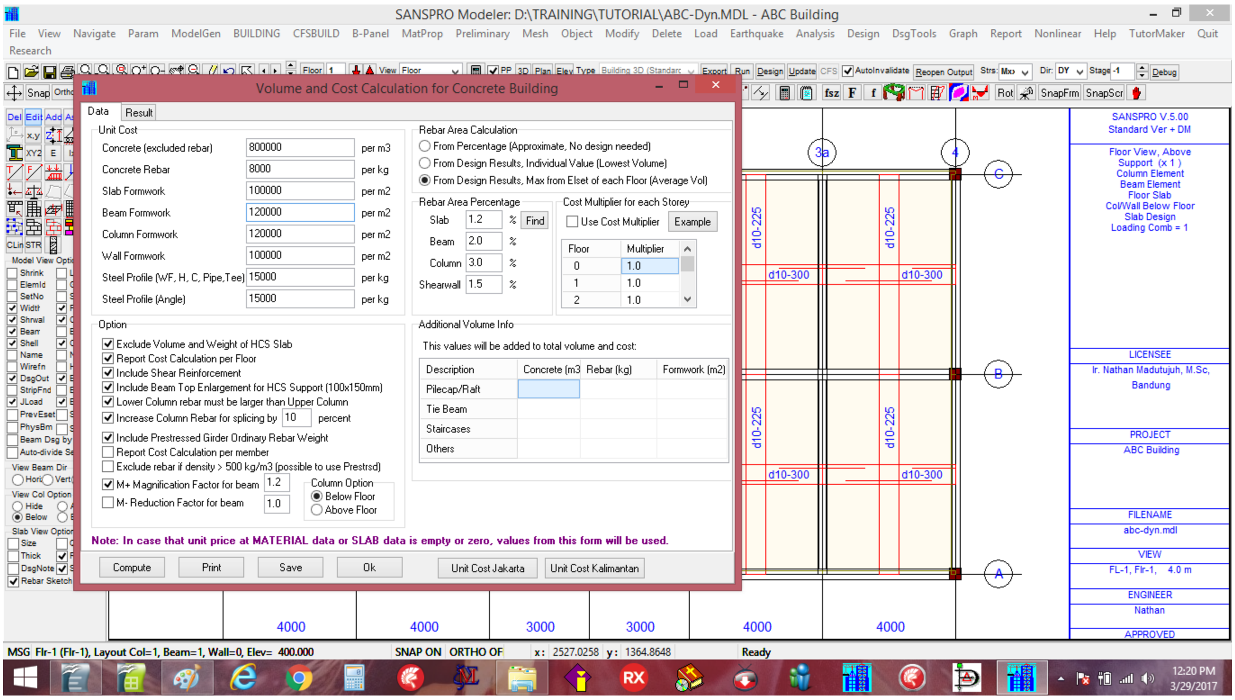
4. Example 6: Building Volume and Cost Design

SANSPRO has an useful feature for evaluating the total volume, weight and cost of concrete, rebar, formwork used in the model. The cost calculation will follow user defined unit price.
This facility is only available for concrete building.

Click menu **Report** – select **Volume and Cost Report**



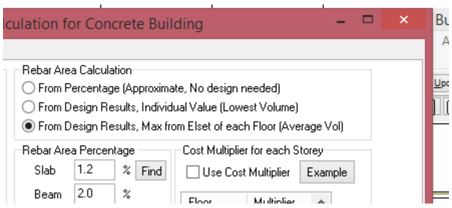
Enter the following unit price: (*in this case we use Rupiah/IDR currency, use any currency you prefer*)
Select (x) **From Design Results, Max from Elset**, then click **[Compute]**



The program will give some useful results :

- Volume, weight, area and price per floor, divided into shearwall, columns, beams, slabs, precast slab
- Total building volume, weight, area and price for concrete, rebar and formwork
- Concrete ratio : total concrete volume / total floor area = equivalent thickness (usually 0.20 – 0.45)
- Rebar ratio : total weight of rebar / total concrete volume (usually 125 – 250 kg/m3)
- Rebar usage per diameter (if design stage has been finished)
- Warnings if some ratio is outside typical values range

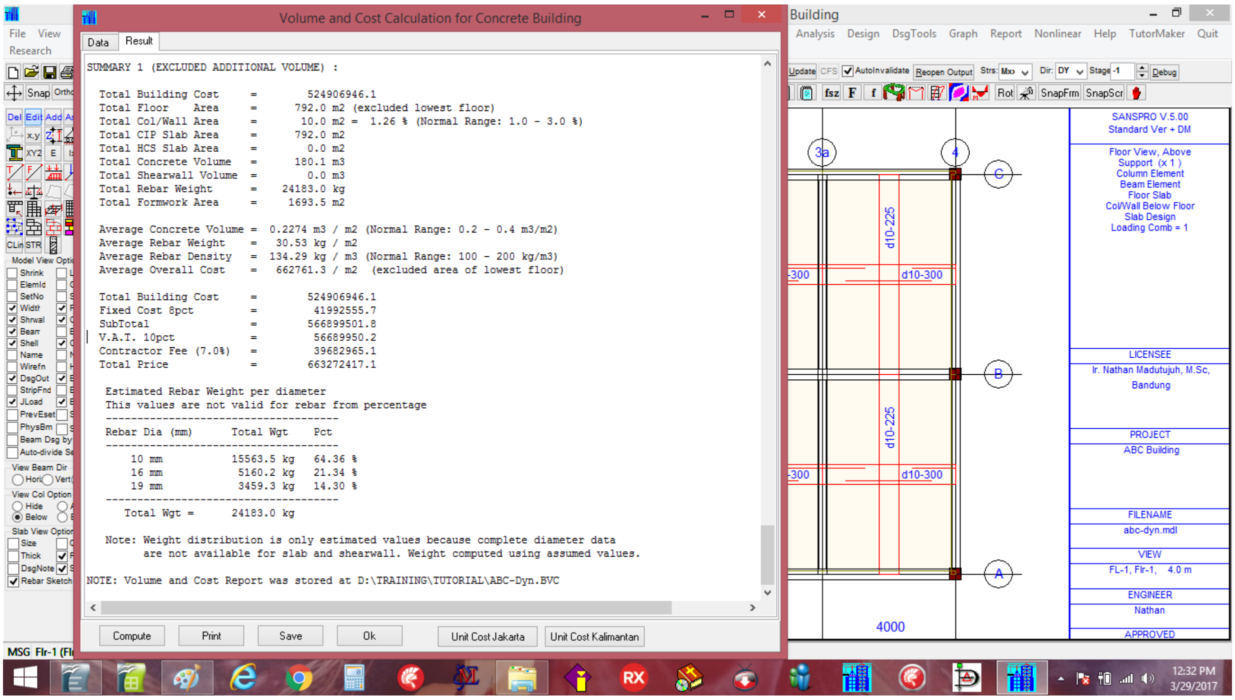
For concrete volume, it will give very accurate results (beam height will be corrected by slab thickness, etc).



For rebar weight, there are 3 options available:

1. From percentage (estimation by user defined % based on section size, no design needed)
2. From Design Results, Individual value (Lowest value)
3. From Design Results, Max from Elset of each floor (maximum value)

The actual weight of rebar needed typically is between average of 2 and 3.



Concrete ratio = 0.224 m3/m2

Rebar ratio = 134.3 kg/m3 to 139.8 kg/m3
Average rebar = 137.05 kg/m3

Overall cost = 662 761 Rp/m2 to 672 813 Rp/m2
Average cost = 667 787 Rp/m2

NOTE:

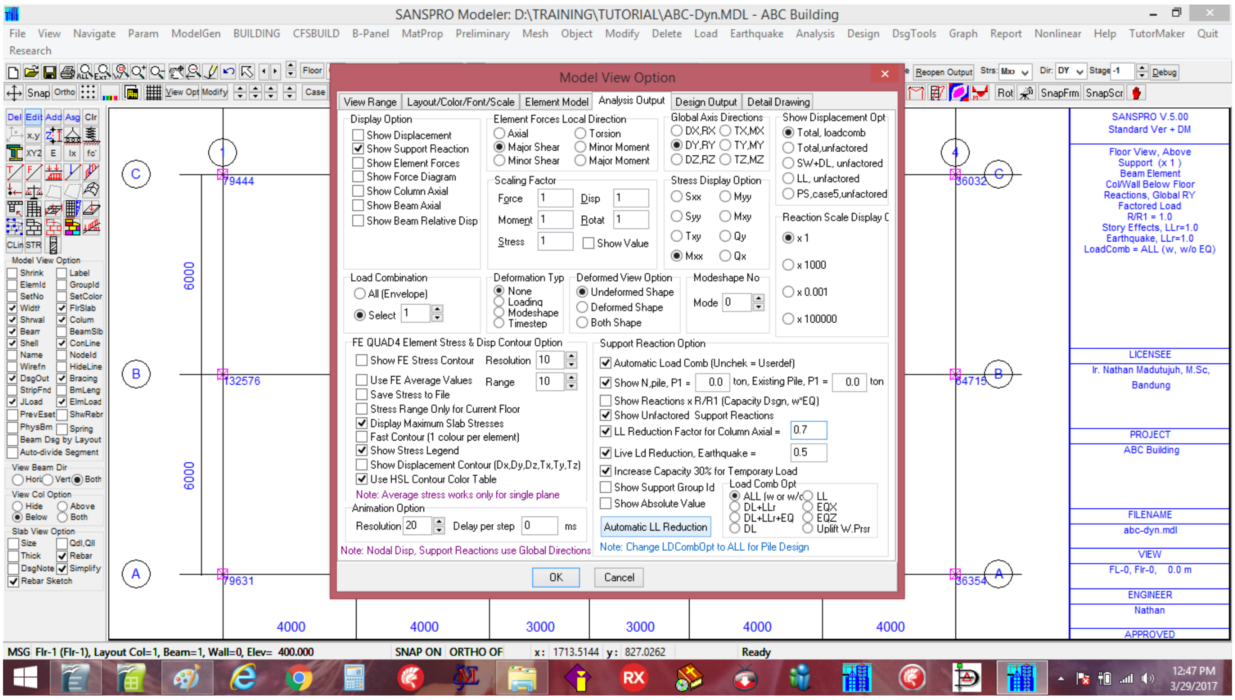
- Volume and weight of pilecap, tie beam, retaining wall, staircase are excluded and must be added manually
- Using this feature, we can conduct WHAT-IF study easily for various parameters:

– Concrete strength fc'
– Column size
– Beam size

5. Example 7: Simplified Foundation Design

Very simple and fast pile foundation design can be done using SANSPRO as follows:

- **Right-click**, select **View Support Reactions**
- **Right-Click**, select **Change View Option**
- Enter the allowable pile axial load capacity (ton)
- Enter the following parameters:
- For pilecap with more than 2 piles without shallow bedrock :
  - Pile axial load capacity may be reduced by pile group efficiency factor = 0.65 – 1.0
  - Group efficiency must be applied for friction type pile
  - In this case we will use : Pcap = 40 ton x 0.85 = **34 ton**
- Enter the following parameters:
  - [x] Show N,pile, P1 = **34** ton
  - [x] Show Unfactored Support Reactions
  - [x] LL Reduction for Column Axial = 0.7
  - [x] LL Reduction for Earthquake = 0.5
- click **[OK]**

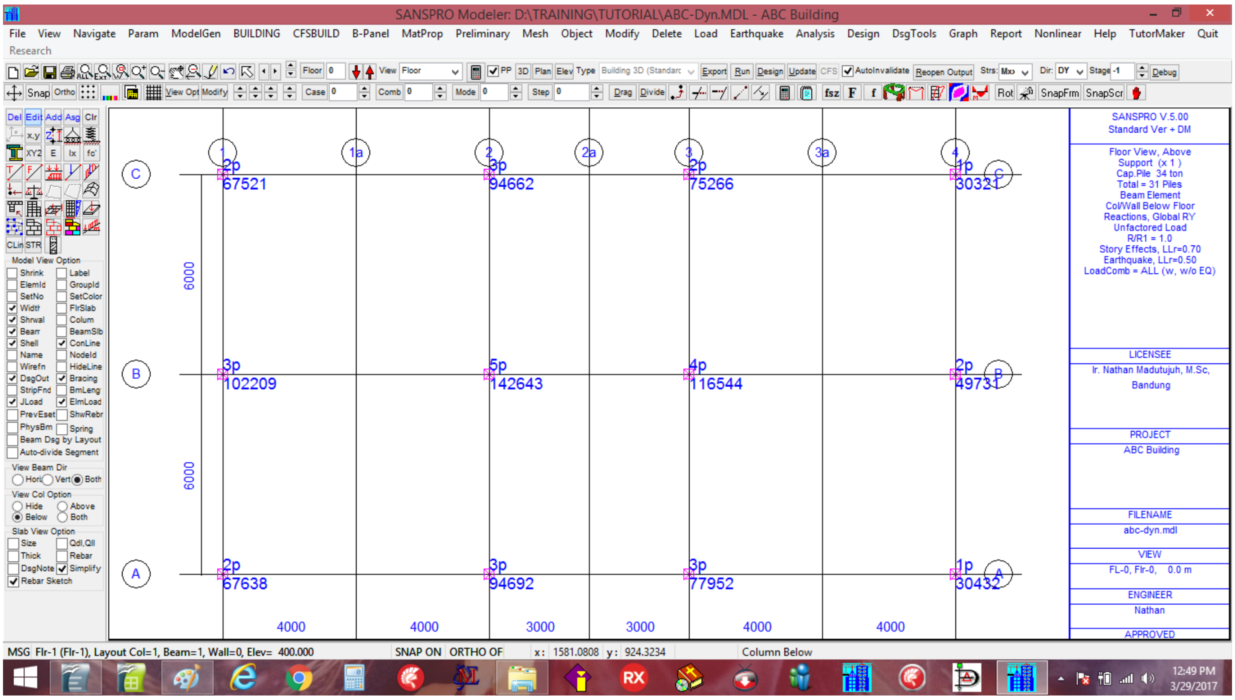


Estimation of number of piles needed will be appear :

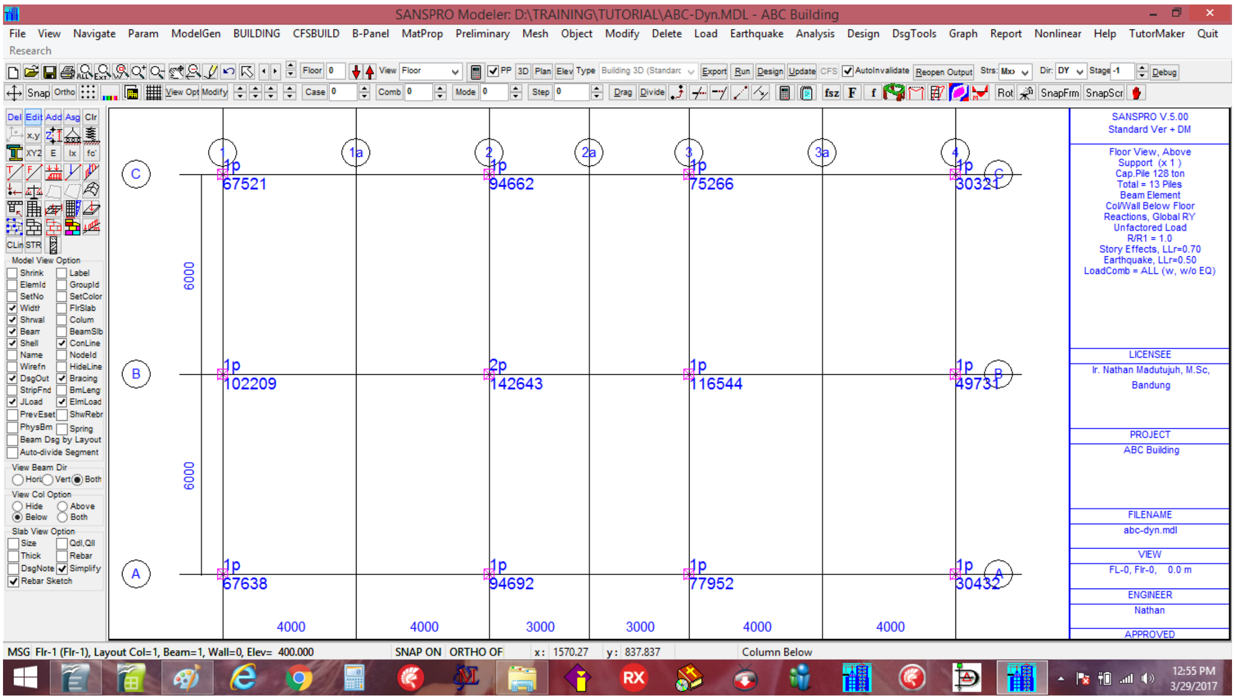
NOTE:

- Total number of piles needed is given at top-right of screen (Total = 31 piles)
- This simplified pile design only considers axial load and neglected shear and moment
- Total number of piles needed for shear is : Vbsh / P1, where P1 = 34 ton
- Special load combinations including SW,DL,LL,EQX,EQZ and Llrf will be used
- More detail foundation design considering Moment, Tension, Shear forces can be done in the other menu (Run Foundation Design)
- If more than 1 type of pile used, repeat the process again using different value of P1
- If using for foundation strengthening, enter the existing capacity of foundation. The number of piles shown will reflect the additional piles needed for foundation strengthening.

Pile number estimation, PC 25x25, 40 ton x 0.85 → Total 31 piles @ PC 25x25 (Precast pile)

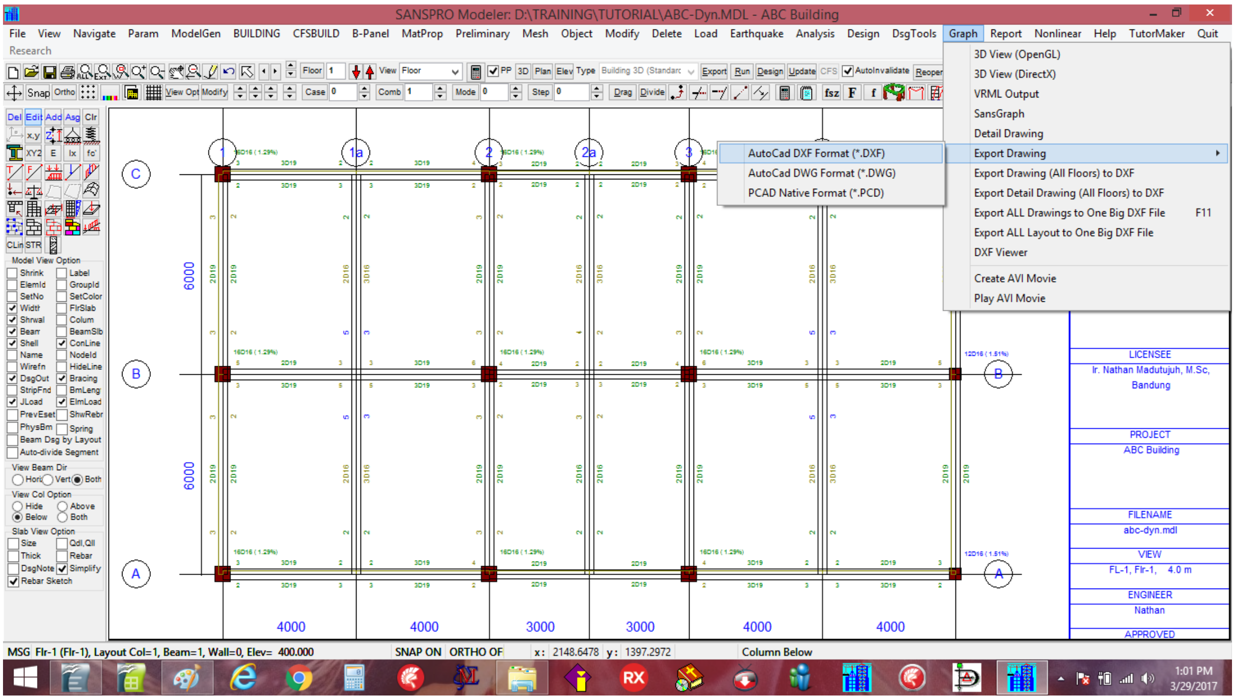


Pile number estimation, BP 60, 150 ton x 0.85 → Total 13 piles @ BP 60 (Bored Pile D600)

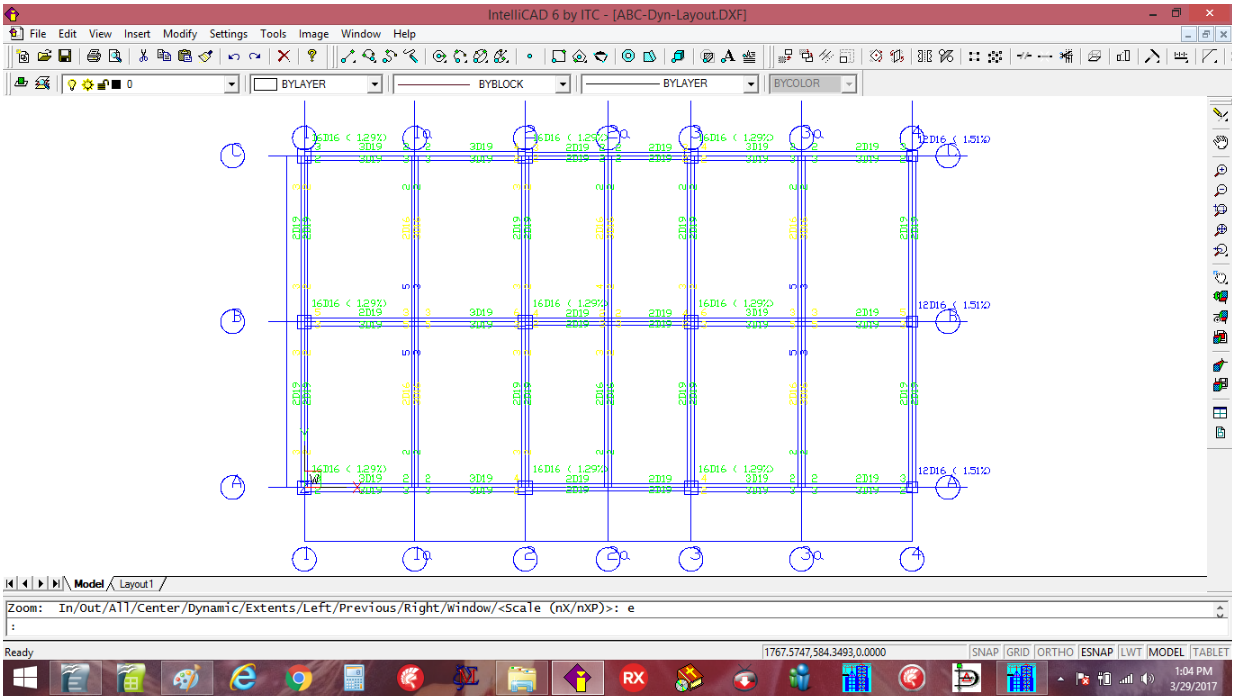


6. Example 8: Generate Detail Drawing

Most drawings that can be seen on screen can be exported to Autocad DXF file format by clicking menu : click **Graph – Export Drawing – Autocad DXF Format (*.DXF)**

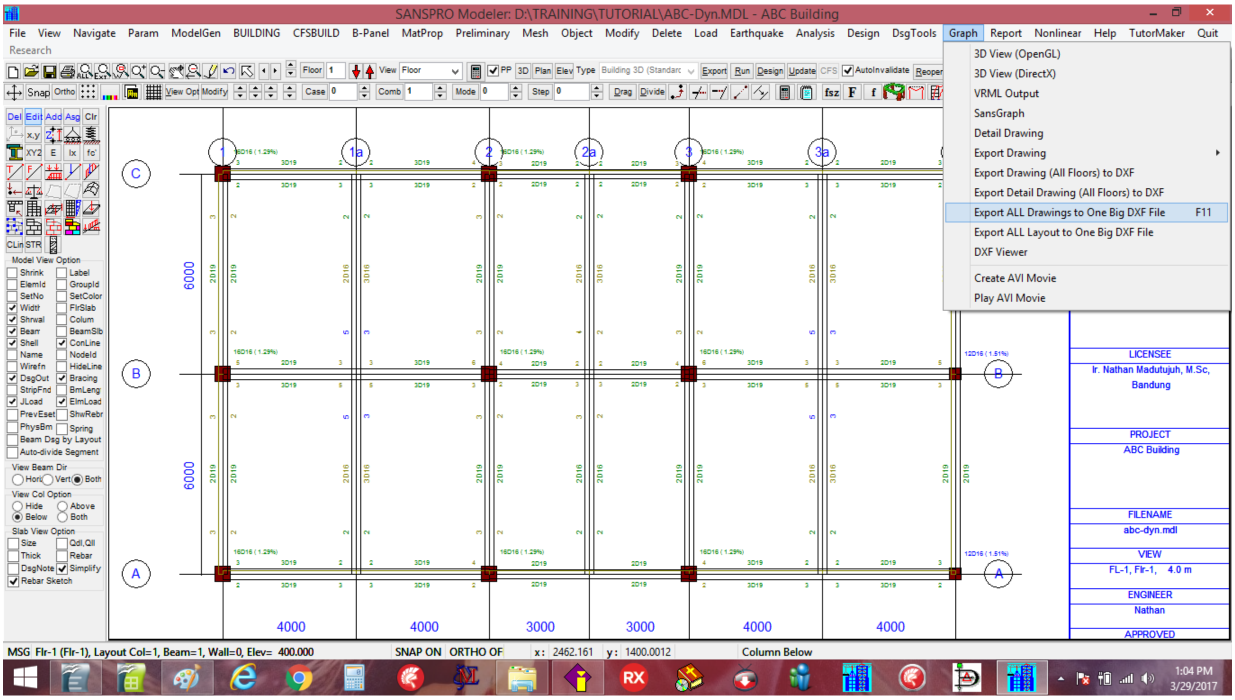


Enter a new DXF File for this drawing: ABC-Dyn-Layout.DXF
A new DXF file will be created and can be open by any CAD Program:

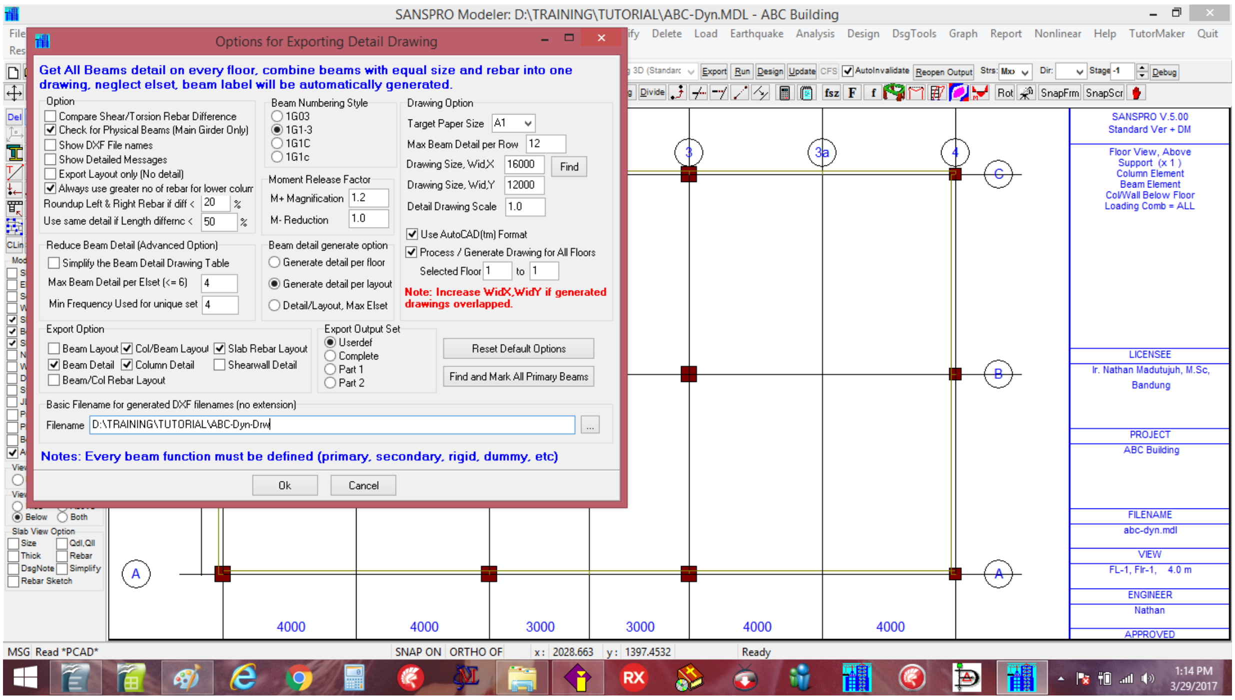


SANSPRO has another useful feature that enable a user to generate detail drawing from the current model.

- Click **Graph – Export All Drawings to One Big DXF File**



Option menu for the drawing generator will appear:
(Usually no changes will be needed except paper size)



- Enter a file name: **ABC-Dyn-Drw**

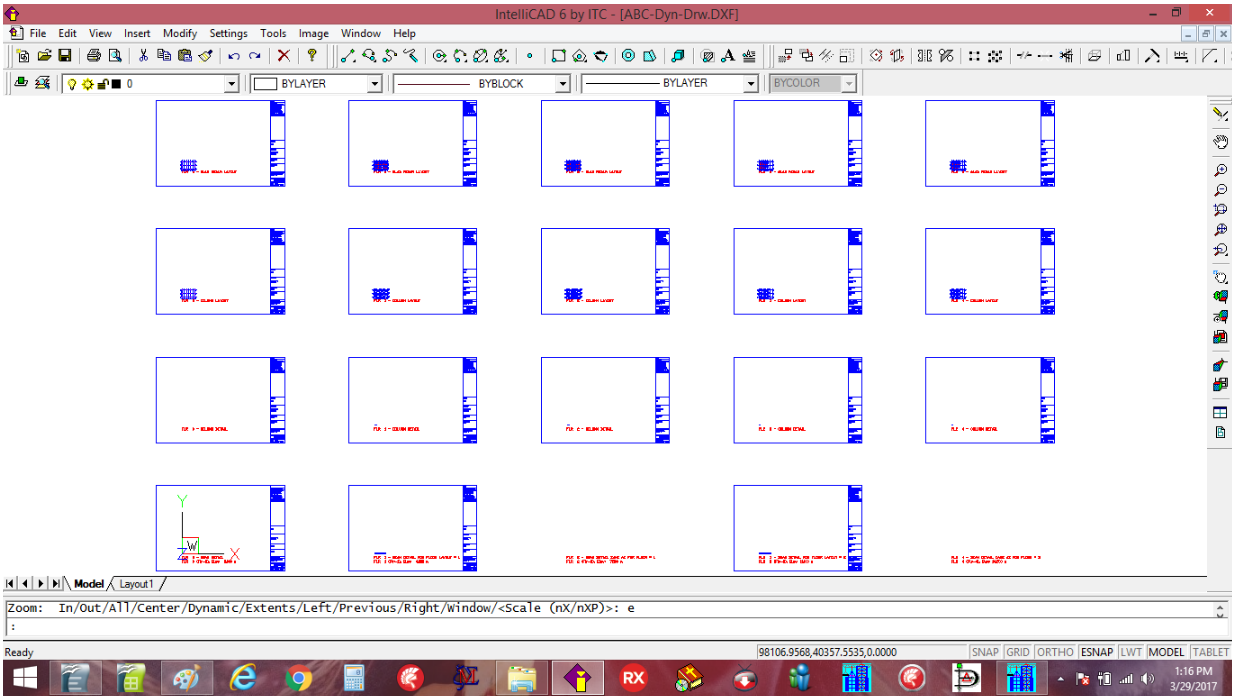
click **[Ok]** to generate the whole detail drawings in one big DXF file:

What is inside the drawing file:

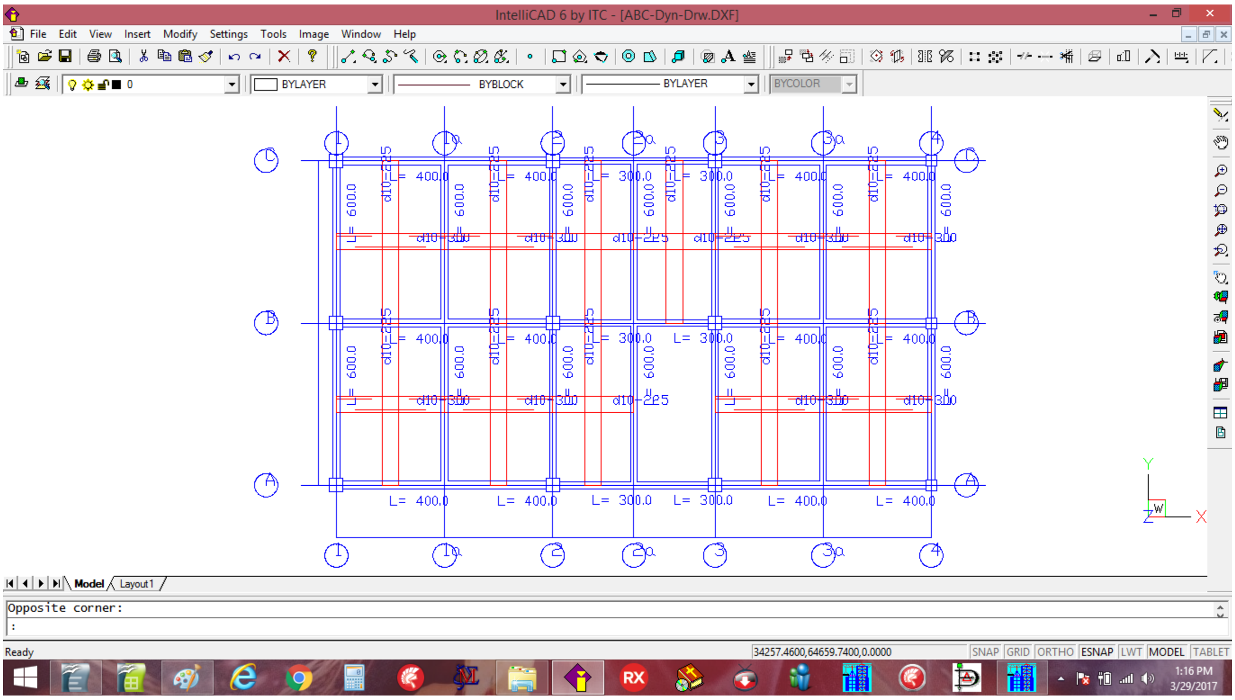
- Building floor layout
- Floor slab rebar skecth
- Shearwall rebar detail
- Column rebar layout and detail
- Beam rebar layout and detail

The whole detail drawing sets generated for this building :

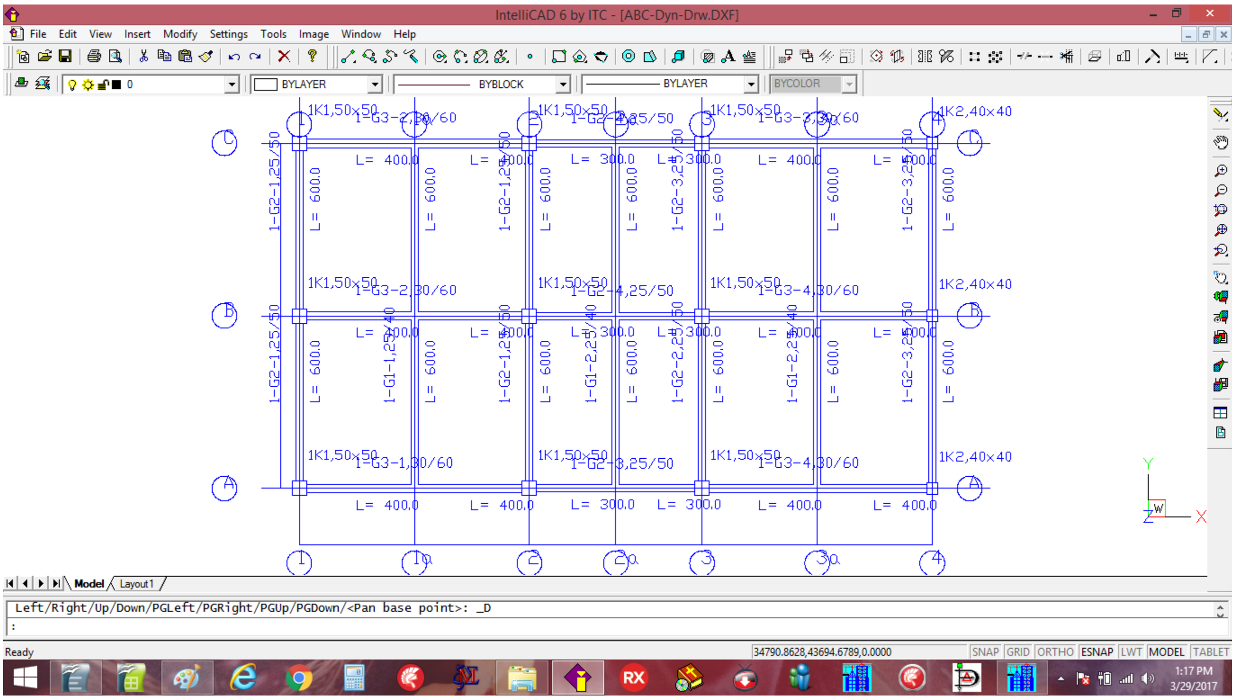
(Simple border and title is given for drawing notes, the generated drawing can be edited by drafter later)



Floor Slab Rebar Sketch:

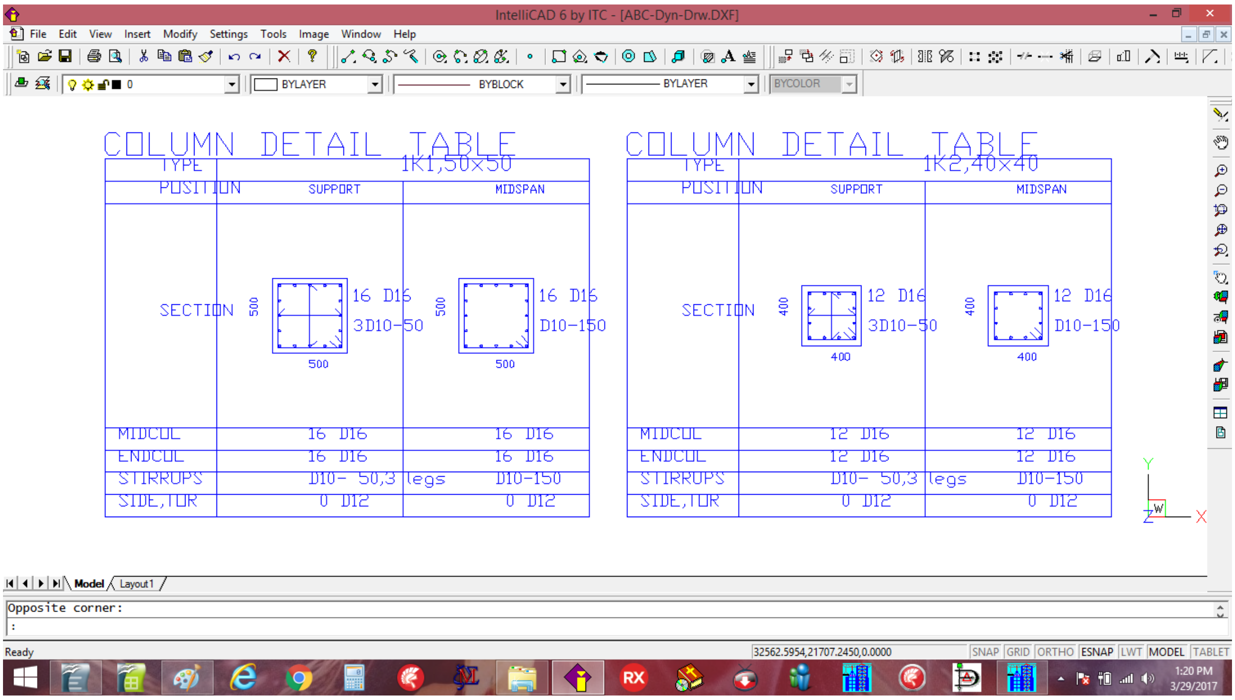


Column and Beam Rebar Layout:



This drawing will contain column and beam layout with index pointing to detail drawing below.
For large building and also if the building is not regularly shape, many beam detail can be generated and user must reduce it manually by selecting the needed ones.

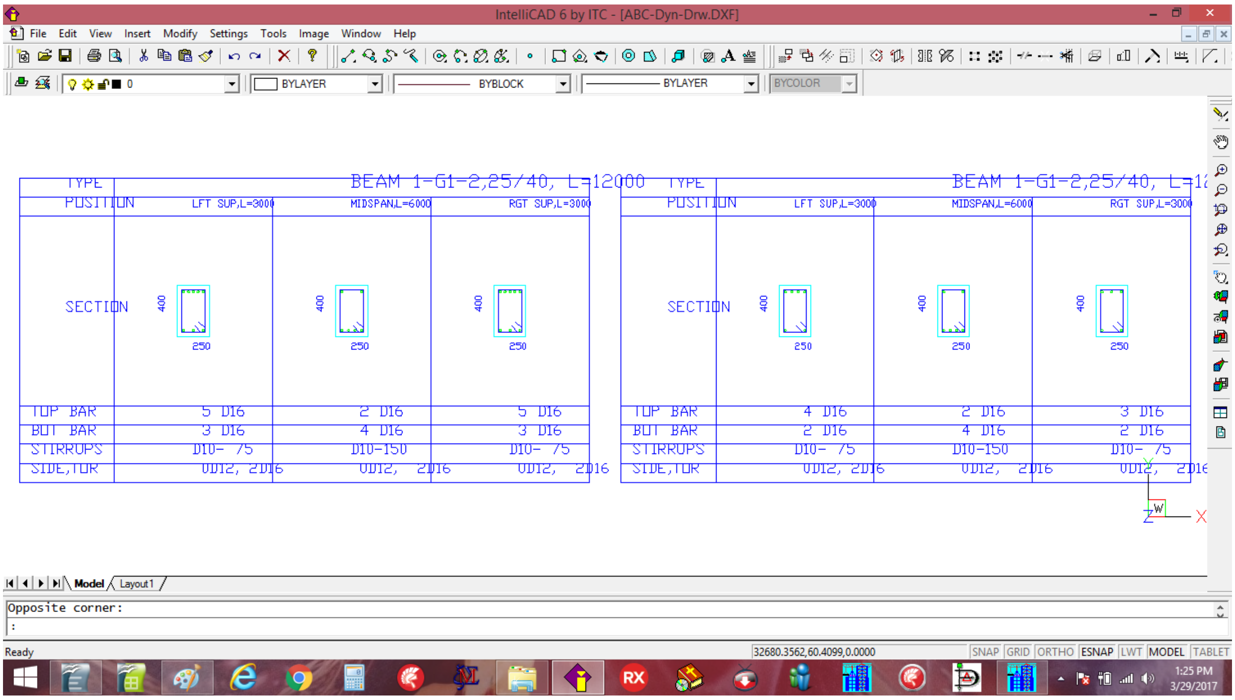
Column Rebar Detail:



NOTE:

- Column hoops and ties arrangement must be same along the columns. The drawing generator will be revised later to correct the drawing.
- For highly seismic zone, the maximum distance of stirrup legs is 150 to 200 mm.
- Maximum column stirrups spacing is 150 mm

Beam rebar drawing:



NOTE:

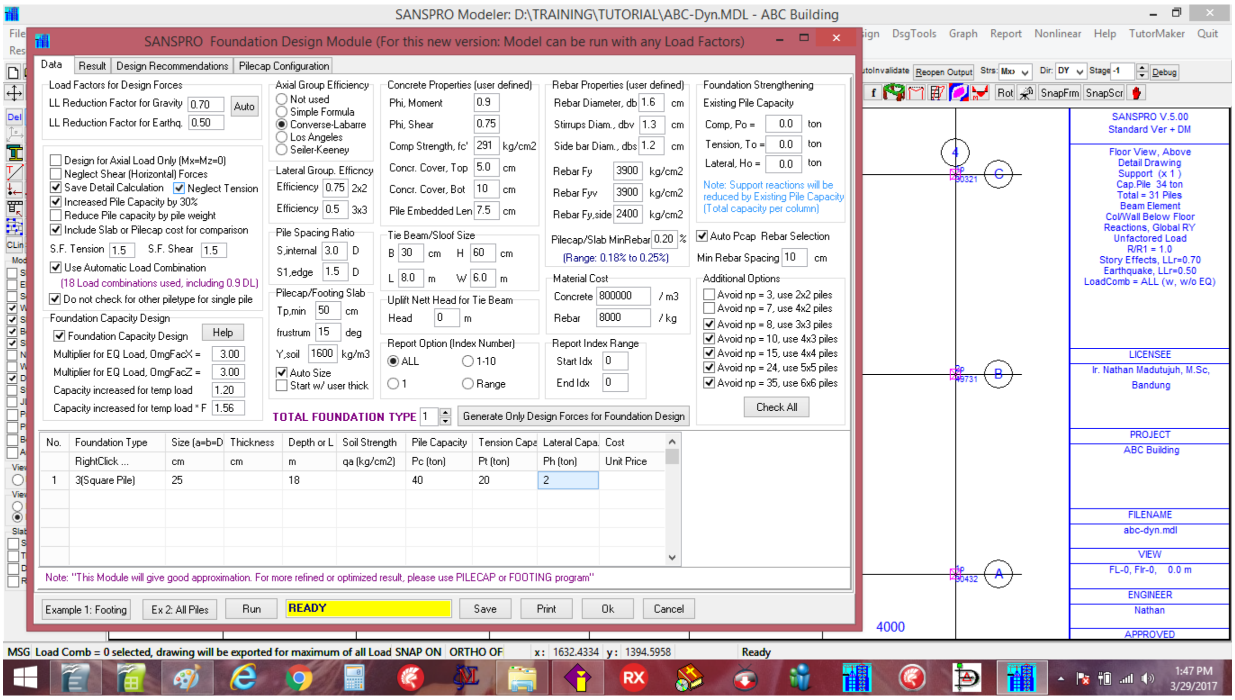
- User is advised to always re-check the beam rebar detail compared to the visual rebar checking per floor
- For highly seismic zone, the maximum distance of stirrup legs is 200 mm.
- Maximum beam stirrups spacing is 150 mm

9. Detail Foundation Design

1. Example 10: Detail Pile Foundation Design

More detail foundation design covering Axial compression, tension, shear and moment forces can be done using

- click menu Design – Run Foundation Design
- Enter the following parameters:



[ ] Neglect Shear forces
[x] Neglect Tension
Capacity increased for temporary load = 1.20 (for medium earthquake load combination)
Capacity increased for temporary load * F = 1.56 (For large earthquake load combination)

Axial Group Efficiency : (x) Converse-Labarre (higher coefficient than standard / simple method)
Tie Beam size : B=30, H=60
Uplift Head : 0 m (No uplift water pressure)
Rebar Dia, Db : 1.6 cm

Total Foundation Type = 1
Foundation Type = 3(Square Pile), Size = 25, Depth = 18m,
Pile capacity = 40, Tension = 20, Lateral = 2.0 ton

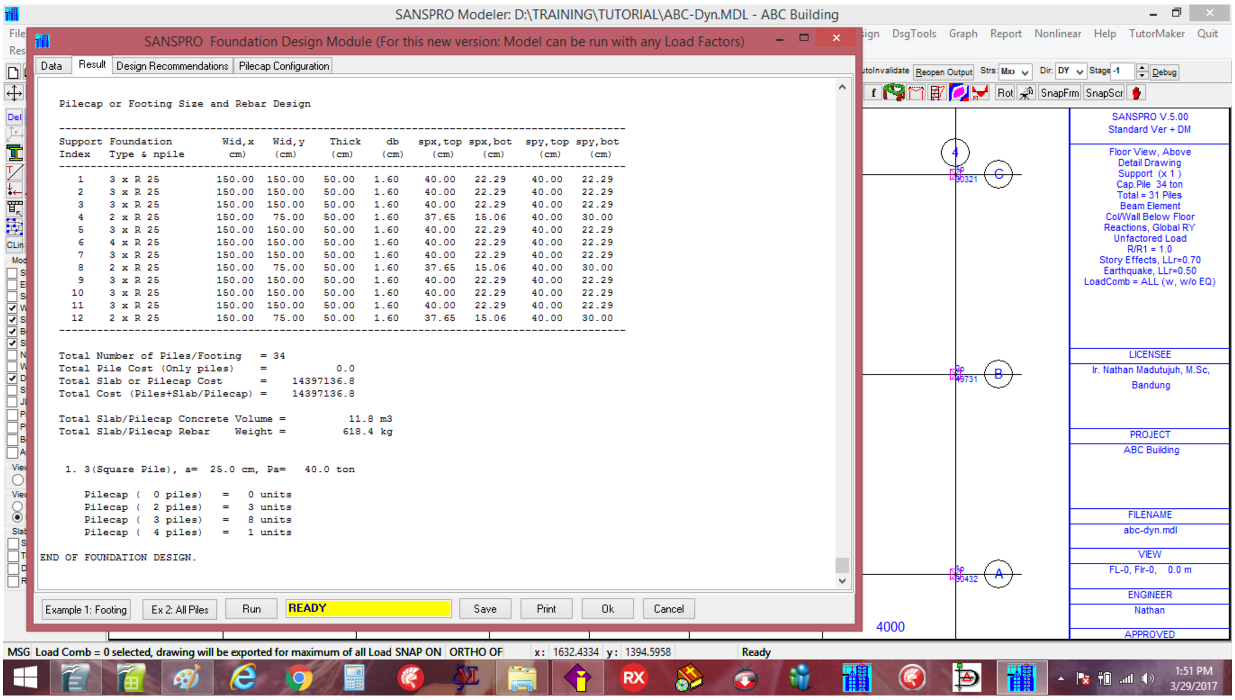
click **[Run]** to start the foundation design process.

(NOTE: Due to many iterations, it may requires some time to finish, usually couple of minutes).

Output of the detail foundation design are:

- Number of pile (from total load including weight of pile and pilecap, axial, shear, tension, moment force, group efficiency, gravity and earthquake load combinations)
- Recommended pilecap size and rebar diameter selection (checked for punching shear and bending moment)
- Detail calculation report for each support
- Foundation and Pilecap layout detail drawing

Detail Foundation Report:

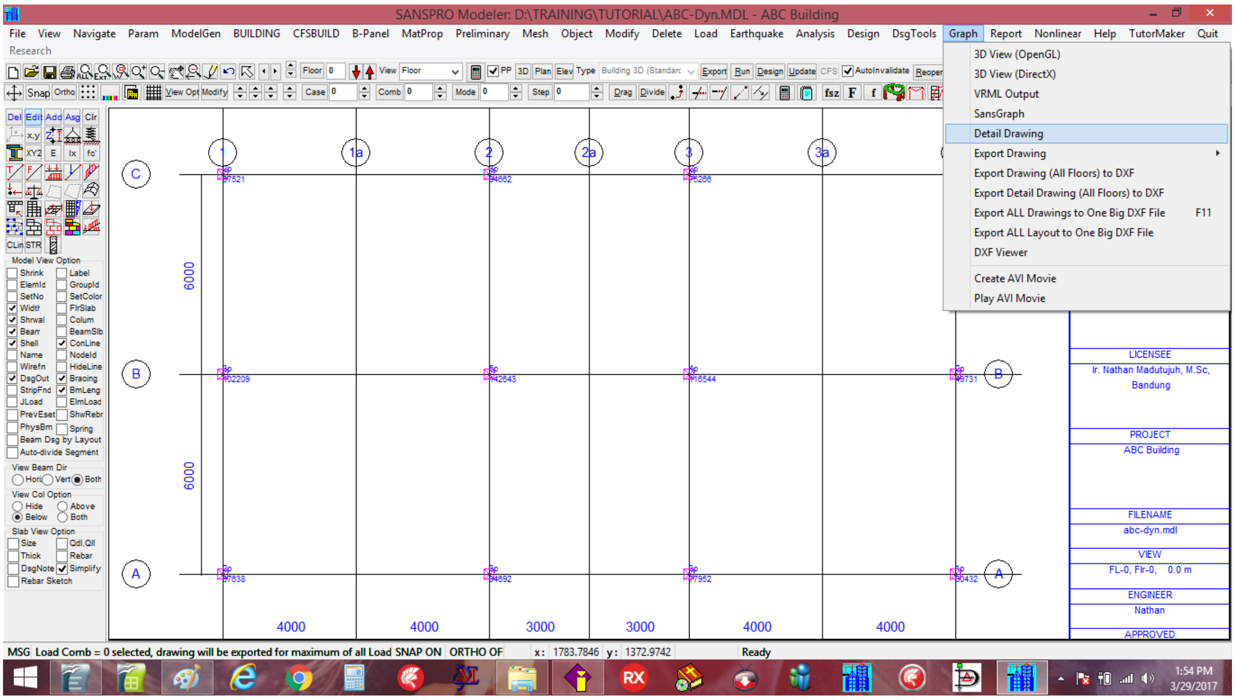


Total number of piles = 34 (a little bit larger than simplified design due to Shear forces considered in design) Pilecap size and recommended rebar diameter and spacing also given above.

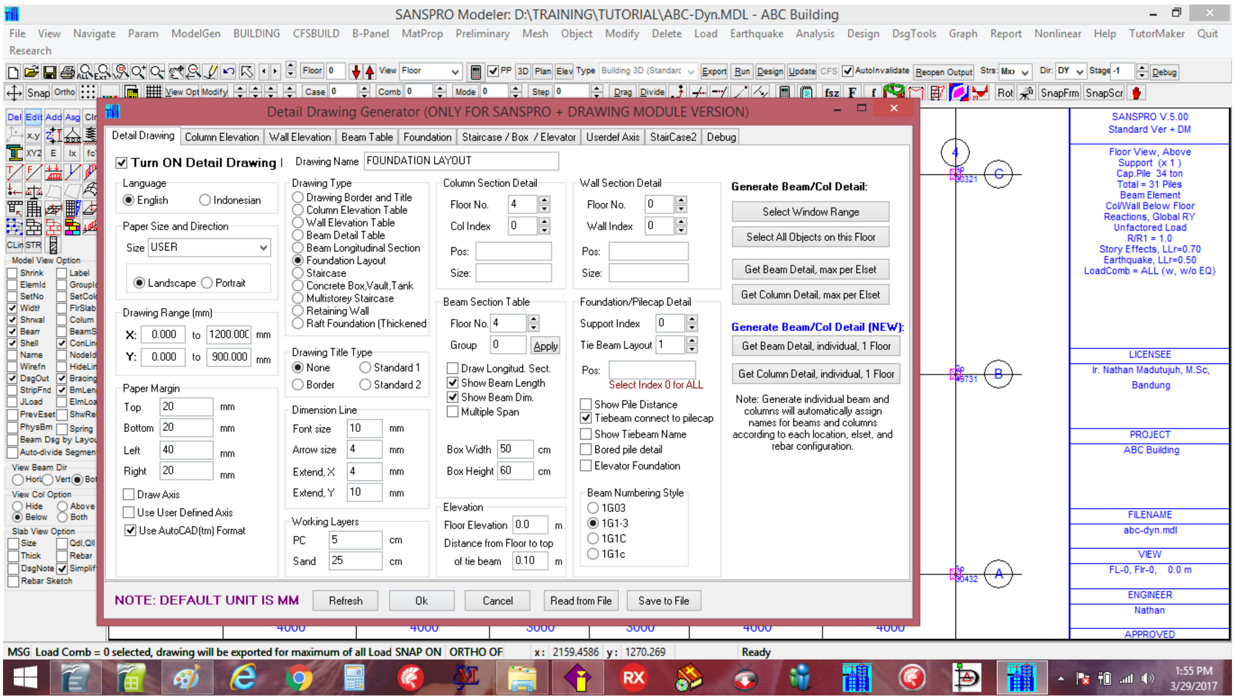
Generating Pile foundation detail drawing :

SANSPRO provides easy way to generate and visually check the foundation results above as follows:

- click menu Graph – Detail Drawing

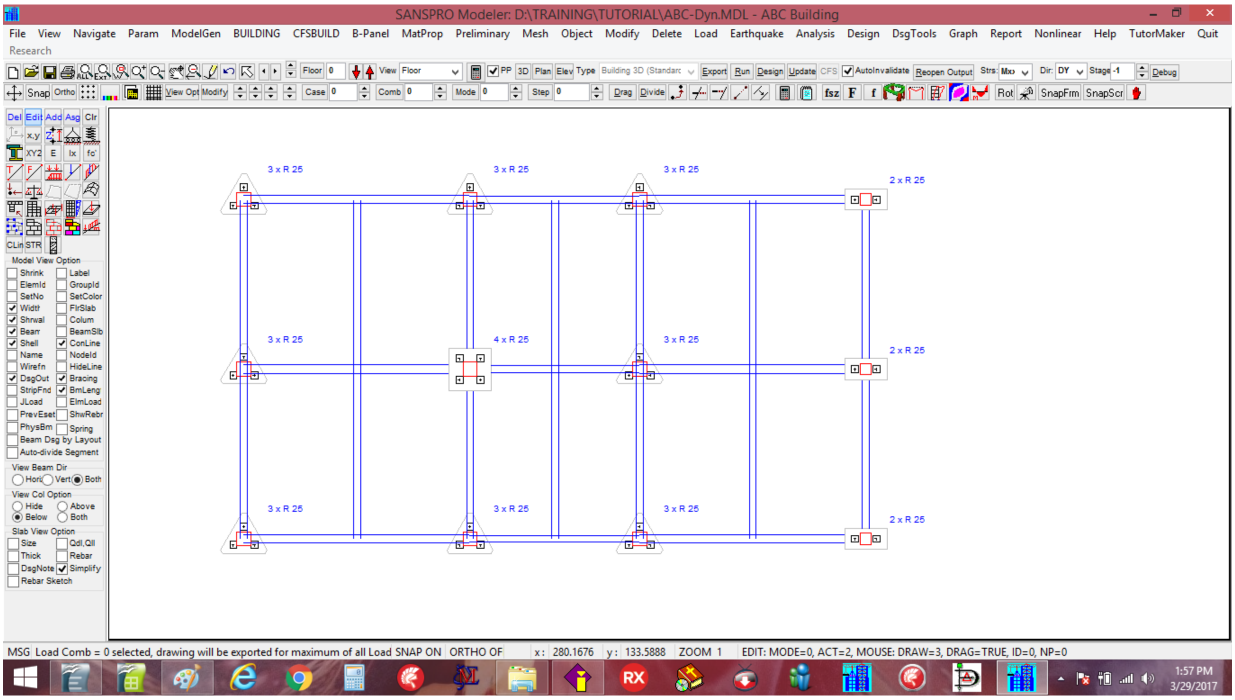


Enter the following parameters:



click **[x] Turn ON Detail Drawing**
click **(x) Foundation Layout**
Tie Beam Layout = **1**
[ ] **Show Pile Distance**
[ ] **Show Bored Pile detail**

click **[Ok]**
zoom to enlarge the drawing.



Using this visual checking we can easily check the foundation design result and revise it if necessary.

The shown drawing can be exported to Autocad DXF by clicking menu **Graph – Export Drawing**.

Foundation Design Report :

A very comprehensive detail calculation report for the foundation design is also given.
To save some paper, only some reports for certain columns or support needed to be included in final report.

Detail Foundation Report:

SANSPRO V.5.10 Foundation Design Utility
(C) Nathan Madutujuh, 1988-2017
Code: ACI-318-2011, PBI-91, PBI-2002, PBI-2013
Licensee: Ir. Nathan Madutujuh, M.Sc, Bandung

A. ASSUMPTIONS:

1. Reactions Forces

   - Reactions are taken Automatically from Load Cases and combined into design group
   - User Load Factors will not be used for combinations
   - Live Load Reduction Factors use user defined live load reductions:
   - Live Load Reduction Factor for Storey Number LLRF1 = 0.7
   - Live Load Reduction Factor for Temporer Load LLRF2 = 0.5
   - Load combinations for maximum tension/uplift use 0.9*DL
   - Load combinations selected = ALL
   - Use Vertical Earthquake, Av = 0.12

   - Forces Included: N,Vx,Vz,Mux,Muz (Axial and Biaxial Bending)
   - Axial forces used : Nmin, Nmax to accomodate compression and tension
   - Foundation capacity is increased by 120.0 % for temporary loading

2. Spread Footing

   - Three Conditions of Soil Stress Pattern (no tension allowed)
   - Uplift Force is resisted by slab weight and soil weight
   - Rebar Minimum uses value given by user (rho >= 0.15%)
   - Slab Thickness is checked for punching shear
   - Slab Rebar is calculated for compression and tension condition

3. Pile Foundation

   - Pile configuration is from standard/optimum configuration
   - Pile to pile/edge distance ratio are determined by user
   - Pile min/max force is determined using rigid slab assumption
   - Rebar Minimum uses value given by user (rho >= 0.15%)
   - Tension pile rebar is determined using max tension force

   - Capacity Design Method used
   - OmegaFacX = 3.00
   - OmegaFacZ = 3.00
   - RR1FAC = 0: Pile capacity is not increased, P = 1.0 x P
   - RR1FAC = 1: Pile capacity P = P x LFTEMP1 for temporary load (Default = 1.5)
     (moderate earthquake case)
   - RR1FAC > 0: Pile capacity P = P x LFTEMP2 for temporary load (Default = 2.5)
     (strong earthquake case, capacity design)

   - Pile capacity is NOT reduced by pile self-weight
   - Pile axial capacity is reduced by pile group efficiency
     theta = ArcTan(D/s)
     eff = 1.0 - theta * ((n-1)*m+(m-1)*n)/(0.5*Pi*m*n)
   - Pile lateral capacity reduction factor for 1x1 pile : 1.0
   - Pile lateral capacity reduction factor for 2x2 pile : 0.75
   - Pile lateral capacity reduction factor for > 3x3 pile : 0.5

4. Pilecap Design

   - Pilecap size is from standard/optimum configuration
   - Pilecap Thickness is checked for punching shear from column and group block
   - Punching shear from column is neglected if d > 1.1*(2*dp-bcol)
   - Mininum Thickness from punching shear of column is 1.1*(2*dp-bcol)
   - Pilecap Thickness is checked for punching shear from pile
   - Pilecap rebar is designed for nett bending (beam action)
   - Pilecap rebar is designed differently for top and bottom rebar
   - Minimum Pilecap rebar ratio is 0.18% to 0.25% depends on Fy or user defined
   - Top and Bottom Concrete Cover can be different values
   - Segment of pile embedded to pilecap is included in calculation
   - Bending Moment Mx = cmbx * P1, My = cmby * P1
   - Where P1 = Single Pile Compression Capacity
   - Where cmbx, cmby are properties of pilecap configuration

5. Tie Beam/Sloof Design

   - Longitudinal rebar is designed for tension and bending
   - Tension is calculated from 10% of maximum column compression
   - Bending is calculated from self-weight and uplift force (LxW area)

6. Cost Calculation

   - Pile cost is unit cost per pile (for total length of pile)
   - Cost includes pilecap, excludes tie beam/sloof and basement slab
   - Only detail calculation for foundations with minimum cost are displayed
   - Unit price of concrete = 800000 / m3
   - Unit price of rebar = 8000 / kg

B. FOUNDATION DESIGN CALCULATION:

```
*************************************************************************
1. SUPPORT NO.   1, Node=  1, Location: x=     0.00000, y=     0.00000

  phi,m   =  0.80       phi,v =  0.60
  fc1     = 291.0 kg/cm2  fy  = 3900.0 kg/cm2
  fyv     = 3900.0 kg/cm2  fys = 2400.0 kg/cm2
  s,ratio =  3.00    s1,ratio =  1.50
  col,bx  = 50.00 cm   col,bz = 50.00 cm
  sloof   db=1.60 cm, dbv=1.30 cm, dbs=1.20 cm
  pilecap db=1.60 cm, dbv=1.30 cm, dbs=1.20 cm
  foot    db=1.60 cm, pile embeded=7.5 cm

      Unfactored forces: all(f1*f2), static, temporary (f1*f2), temporary (f1*f2=1)
     Maximum Axial,  Pu =    76829.8,    59124.4,    76829.8,    67637.6 kg
     Minimum Axial,  Pu =    26024.8,    39813.1,    26024.8,    35217.0 kg.cm
     Moment, X-Dir, Mux =  1755788.9,   175691.4,  1755788.9,   708541.1 kg.cm
     Moment, Y-Dir, Muy =  2152478.2,   124274.1,  2152478.2,   803633.5 kg.cm
     Horiz Force,   Vux =     7250.5,     1503.8,     7250.5,     3469.9 kg
     Horiz Force,   Vuy =     7571.6,      911.3,     7571.6,     3156.2 kg

      Factored forces: all(f1*f2), static, temporary (f1*f2), temporary (f1*f2=1)
     Single Axial,   Pu =        0.0,        0.0                   kg
     Maximum Axial,  Pu =    88295.7,    74148.0,    88295.7,    79103.4 kg
     Minimum Axial,  Pu =    26024.8,    39813.1,    26024.8,    35217.0 kg.cm
     Moment, X-Dir, Mux =  1788530.4,   222706.9,  1788530.4,   741282.6 kg.cm
     Moment, Y-Dir, Muy =  2174762.3,   159091.6,  2174762.3,   825917.7 kg.cm
     Horiz Force,   Vux =     7529.0,     1909.3,     7529.0,     3748.4 kg
     Horiz Force,   Vuy =     7735.6,     1165.7,     7735.6,     3320.2 kg

  PILE FOUNDATION DESIGN:

  1. Pilecap Thickness:

     a. Given Pilecap Thickness, Tp =   30.70 cm

     b. From Punching of Single Pile:
        Factored Punching Force, 1 pile,    Pu =   60000.00 kg
        Allowable Punching Stress,          vc =   18.09 kg/cm2, fc1 =  291.00 kg/cm2
        Perimeter Length of Punching Area, Kp1 =  200.00 cm
        Tpmin from Punch Shear of One Pile     =   40.00 cm

     c. From Punching of Single Column:
        Punching of Single Column Status         = Skipped
        Factored Punching Force, Column,    Pu =   79103.45 kg
        Allowable Punching Stress,          vc =   18.09 kg/cm2
        Perimeter Length of Punching Area, Kp2 =  310.53 cm

     d. Minimum Thickness required by user, Tpmin =   50.00 cm

     e. Selected Pilecap Thickness, Tp =   50.00 cm
        Allowable Punching Stress,        vc =   18.09 kg/cm2
        Shear Stress, Punching of Pile,   vc =   15.77 kg/cm2  -> OK
        Shear Stress, Punching of Column, vc =   14.95 kg/cm2 Neglected

  2. Pile Number Calculation:

     a. First Trial (pilecap weight = 0, + for compression)

        Unfactored Max Force, Static Load           Pu1 =      59.12 ton, Pcap1 =      40.00 ton, np1=2
        Unfactored Max Force, Temp. Load, F=f1*f2,   Pu1 =      76.83 ton, Pcap1 =      62.40 ton, np1=2
        Unfactored Max Force, Temp. Load, F=1.0,     Pu1 =      67.64 ton, Pcap1 =      48.00 ton, np1=2
        Pilecap Weight                            Wpcap =    0.00 ton
        Weight of One Pile,                          Wp =    0.00 ton
        Gross Capacity of One Pile,                 P1 =      40.00 ton
        Nett  Capacity of One Pile,                 P1 =      40.00 ton
        Number of Piles needed for Compression Force, Np1 =   2 piles
        Total Compression Capacity (No Earthquake),   Pn =      80.00 ton  -> OK
        Total Compression Capacity (f1*f2=1.0),       Pn =      96.00 ton  -> OK
        Total Compression Capacity (Use f1*f2),       Pn =     124.80 ton  -> OK

        Unfactored Min Force, (Tension=negative), Pumin =      35.22 ton
        Pilecap Weight                           Wpcap =    0.00 ton
        Unfactored Tension reduced by Pilecap Wgt,   Tu =       0.00 ton (compression)
        No Tension Force Occured -> OK

        Unfactored Max Force, Static Load           Vu1 =       1.50 ton, Pcap1 =      20.00 ton, np1=1
        Unfactored Max Force, Temp. Load, F=f1*f2,   Vu1 =       7.57 ton, Pcap1 =      20.00 ton, np1=3
        Unfactored Max Force, Temp. Load, F=1.0,     Vu1 =       3.47 ton, Pcap1 =      20.00 ton, np1=2
        Unfactored Lateral Force,                   Vu =       0.00 ton
        Lateral Capacity of One Pile,               P3 =       2.00 ton
        Number of Piles needed for Lateral Force,   Np3 =   3 piles
        Total Lateral Capacity (No Earthquake),      Vn =       2.00 ton  -> OK
        Total Lateral Capacity (f1*f2 = 1.0),        Vn =       4.80 ton  -> OK
        Total Lateral Capacity (f1*f2 > 0),          Vn =       9.36 ton  -> OK

        Number of Piles needed, Np =   3 piles

     b. Second Trial (with Pilecap Weight)

        Unfactored Max Force, Static Load           Pu1 =      59.12 ton, Pcap1 =      40.00 ton, np1=2
        Unfactored Max Force, Temp. Load, F=f1*f2,   Pu1 =      76.83 ton, Pcap1 =      62.40 ton, np1=2
        Unfactored Max Force, Temp. Load, F=1.0,     Pu1 =      67.64 ton, Pcap1 =      48.00 ton, np1=2
        Pilecap Weight                            Wpcap =    0.00 ton
        Unfactored Force + Pilecap Weight,          Pu1 =      59.12 ton
        Weight of One Pile,                          Wp =    0.00 ton
```

```
      Gross Capacity of One Pile,                   P1 =      40.00 ton
      Nett  Capacity of One Pile,                   P1 =      40.00 ton
      Number of Piles needed for Compression Force,Np1 =   2 piles
      Total Compression Capacity (No Earthquake),   Pn =      80.00 ton  -> OK
      Total Compression Capacity (f1*f2=1.0),       Pn =      96.00 ton  -> OK
      Total Compression Capacity (Use f1*f2),       Pn =     124.80 ton  -> OK

      Unfactored Min Force, (Tension=negative), Pumin =      35.22 ton
      Pilecap Weight                            Wpcap =    0.00 ton
      Unfactored Tension reduced by Pilecap Wgt,   Tu =       0.00 ton (compression)
      No Tension Force Occured -> OK
      Number of Pile needed, Np =   3 piles
        Compres: P1 = (Nmax+Wpcap-Po)/np =     23445.85 kg, dPMx =      4723.61 kg, dPMy =      5357.56 kg
        Tension: P1 = (Nmin+Wpcap-To)/np =     12639.00 kg, dPMx =      4723.61 kg, dPMy =      5357.56 kg
        Pcomp=    52000.00  Ptens=    26000.00, P1max =     33527.01, P1min =      2557.84

   c. Third Trial (with Group Efficiency and Bending Moment)

      Number of Pile needed,                        Np =   3 piles
      Group Efficiency Method                          = Converse-Labarre
      Group Efficiency,                             e  = 0.795
      Unfactored Max Force, (+ -> compression),  Pumax =      76.83 ton
      Unfactored Min Force, (Tension=negative),  Pumin =      26.02 ton
      Pilecap Weight                             Wpcap =    2.70 ton
      Unfactored Max Force + Pilecap Weight,       Pu1 =      59.12 ton
      Unfactored Min Force + Pilecap Weight,       Pu2 =      37.92 ton
      Weight of One Pile,                           Wp =    0.00 ton
      Gross Compression Capacity of One Pile,       P1 =      40.00 ton
      Nett  Compression Capacity of One Pile,       P1 =      40.00 ton
      Tension Capacity of One Pile,                 P2 =      20.00 ton
      Tension Capacity of One Pile + Pile weight,  P22 =      20.00 ton
      Maximum Compression on Pile,               P1max =      33.53 ton  -> OK
      Minimum Compression on Pile,               P1min =       2.56 ton  -> OK
      Concrete Slab Design Status, X-Direction = OK
      Concrete Slab Design Status, Y-Direction = OK

Optimum Foundation Selected, Index = 1

Pile, Rect, a= 25 cm

  Pile Size Parameter:

       a =     25.00000 cm
       b =     25.00000 cm
      sp =     75.00000 cm
     sp1 =     37.50000 cm
     spx =     75.00000 cm
     spy =     75.00000 cm
      Ap =    625.00000 cm2
      dp =     25.00000 cm
     Apw =      0.00000 cm2
      Kp =    100.00000 cm
     Kp1 =    200.00000 cm

PILE FOUNDATION DESIGN:

1. Pilecap Thickness:

   a. Given Pilecap Thickness, Tp =   30.70 cm

   b. From Punching of Single Pile:
      Factored Punching Force, 1 pile,    Pu =   60000.00 kg
      Allowable Punching Stress,          vc =   18.09 kg/cm2, fc1 =  291.00 kg/cm2
      Perimeter Length of Punching Area, Kp1 =  200.00 cm
      Tpmin from Punch Shear of One Pile     =   40.00 cm

   c. From Punching of Single Column:
      Punching of Single Column Status         = Skipped
      Factored Punching Force, Column,    Pu =   79103.45 kg
      Allowable Punching Stress,          vc =   18.09 kg/cm2
      Perimeter Length of Punching Area, Kp2 =  310.53 cm

   d. Minimum Thickness required by user, Tpmin =   50.00 cm

   e. Selected Pilecap Thickness, Tp =   50.00 cm
      Allowable Punching Stress,        vc =   18.09 kg/cm2
      Shear Stress, Punching of Pile,   vc =   15.77 kg/cm2  -> OK
      Shear Stress, Punching of Column, vc =   14.95 kg/cm2 Neglected

2. Pile Number Calculation:

   a. First Trial (pilecap weight = 0, + for compression)

      Unfactored Max Force, Static Load          Pu1 =      59.12 ton, Pcap1 =      40.00 ton, np1=2
      Unfactored Max Force, Temp. Load, F=f1*f2,   Pu1 =      76.83 ton, Pcap1 =      62.40 ton, np1=2
      Unfactored Max Force, Temp. Load, F=1.0,     Pu1 =      67.64 ton, Pcap1 =      48.00 ton, np1=2
      Pilecap Weight                             Wpcap =    0.00 ton
      Weight of One Pile,                           Wp =    0.00 ton
      Gross Capacity of One Pile,                   P1 =      40.00 ton
      Nett  Capacity of One Pile,                   P1 =      40.00 ton
      Number of Piles needed for Compression Force, Np1 =   2 piles
      Total Compression Capacity (No Earthquake),   Pn =      80.00 ton  -> OK
      Total Compression Capacity (f1*f2=1.0),       Pn =      96.00 ton  -> OK
      Total Compression Capacity (Use f1*f2),       Pn =     124.80 ton  -> OK

      Unfactored Min Force, (Tension=negative), Pumin =      35.22 ton
      Pilecap Weight                            Wpcap =    0.00 ton
      Unfactored Tension reduced by Pilecap Wgt,   Tu =       0.00 ton (compression)
```

```
      No Tension Force Occured -> OK

      Unfactored Max Force, Static Load           Vu1 =       1.50 ton, Pcap1 =      20.00 ton, np1=1
      Unfactored Max Force, Temp. Load, F=f1*f2,   Vu1 =       7.57 ton, Pcap1 =      20.00 ton, np1=3
      Unfactored Max Force, Temp. Load, F=1.0,     Vu1 =       3.47 ton, Pcap1 =      20.00 ton, np1=2
      Unfactored Lateral Force,                   Vu =       0.00 ton
      Lateral Capacity of One Pile,               P3 =       2.00 ton
      Number of Piles needed for Lateral Force,   Np3 =   3 piles
      Total Lateral Capacity (No Earthquake),      Vn =       2.00 ton  -> OK
      Total Lateral Capacity (f1*f2 = 1.0),        Vn =       4.80 ton  -> OK
      Total Lateral Capacity (f1*f2 > 0),          Vn =       9.36 ton  -> OK

      Number of Piles needed, Np =   3 piles

   b. Second Trial (with Pilecap Weight)

      Unfactored Max Force, Static Load           Pu1 =      59.12 ton, Pcap1 =      40.00 ton, np1=2
      Unfactored Max Force, Temp. Load, F=f1*f2,   Pu1 =      76.83 ton, Pcap1 =      62.40 ton, np1=2
      Unfactored Max Force, Temp. Load, F=1.0,     Pu1 =      67.64 ton, Pcap1 =      48.00 ton, np1=2
      Pilecap Weight                            Wpcap =    2.70 ton
      Unfactored Force + Pilecap Weight,          Pu1 =      59.12 ton
      Weight of One Pile,                          Wp =    0.00 ton
      Gross Capacity of One Pile,                 P1 =      40.00 ton
      Nett  Capacity of One Pile,                 P1 =      40.00 ton
      Number of Piles needed for Compression Force,Np1 =   2 piles
      Total Compression Capacity (No Earthquake),   Pn =      80.00 ton  -> OK
      Total Compression Capacity (f1*f2=1.0),       Pn =      96.00 ton  -> OK
      Total Compression Capacity (Use f1*f2),       Pn =     124.80 ton  -> OK

      Unfactored Min Force, (Tension=negative), Pumin =      35.22 ton
      Pilecap Weight                           Wpcap =    2.70 ton
      Unfactored Tension reduced by Pilecap Wgt,   Tu =       0.00 ton (compression)
      No Tension Force Occured -> OK
      Number of Pile needed, Np =   3 piles

      Pile Configuration:

        np, total =  3,  npx= 2, npy= 2
        Pilecap, bx=  150.0 cm, by=  150.0 cm
        Column Block size, cx=   50.0 cm, cy=   50.0 cm
        Furthest pile, xp,max =   37.50,  yp,max=  37.50
        Sigma dx^2 =         5625.00, Sigma dy^2 =         5625.00
        Bending Moment Coefficient, cmbx =     25.00000, cmby =     25.00000
        Bending Moment (Factored) : Mx = 1400000.00000 kg.cm, My = 1400000.00000 kg.cm
        Compres: P1 = (Nmax+Wpcap-Po)/np =     23445.85 kg, dPMx =      4723.61 kg, dPMy =      5357.56 kg
        Tension: P1 = (Nmin+Wpcap-To)/np =     12639.00 kg, dPMx =      4723.61 kg, dPMy =      5357.56 kg
        Pcomp=    52000.00  Ptens=    26000.00, P1max =     33527.01, P1min =      2557.84

   c. Third Trial (with Group Efficiency and Bending Moment)

      Pile Configuration:

        np, total =  3,  npx= 2, npy= 2
        Pilecap, bx=  150.0 cm, by=  150.0 cm
        Column Block size, cx=   50.0 cm, cy=   50.0 cm
        Furthest pile, xp,max =   37.50,  yp,max=  37.50
        Sigma dx^2 =         5625.00, Sigma dy^2 =         5625.00
        Bending Moment Coefficient, cmbx =     25.00000, cmby =     25.00000
        Bending Moment (Factored) : Mx = 1400000.00000 kg.cm, My = 1400000.00000 kg.cm

      Number of Pile needed,                       Np =   3 piles
      Group Efficiency Method                         = Converse-Labarre
      Group Efficiency,                            e  = 0.795
      Unfactored Max Force, (+ -> compression),  Pumax =      76.83 ton
      Unfactored Min Force, (Tension=negative),  Pumin =      26.02 ton
      Pilecap Weight                            Wpcap =    2.70 ton
      Unfactored Max Force + Pilecap Weight,       Pu1 =      59.12 ton
      Unfactored Min Force + Pilecap Weight,       Pu2 =      37.92 ton
      Weight of One Pile,                          Wp =    0.00 ton
      Gross Compression Capacity of One Pile,       P1 =      40.00 ton
      Nett  Compression Capacity of One Pile,       P1 =      40.00 ton
      Tension Capacity of One Pile,                P2 =      20.00 ton
      Tension Capacity of One Pile + Pile weight,  P22 =      20.00 ton
      Maximum Compression on Pile,              P1max =      33.53 ton  -> OK
      Minimum Compression on Pile,              P1min =       2.56 ton  -> OK
      Concrete Slab Design Status, X-Direction = OK
      Concrete Slab Design Status, Y-Direction = OK

3. Pilecap Rebar Design:

   Rebar pct min         =  0.20 %
   Minimum Rebar Spacing = 10.00 cm
   Bx,By,Tp  =  150.00 x  150.00 x 50.00
   Bending Section in X-direction, b =  150.00 cm, h =   50.00 cm
   Bending Section in Y-direction, b =  150.00 cm, h =   50.00 cm
   Bending Moment  in X-direction, Mpx =   1400000.00 kg.cm
   Bending Moment  in Y-direction, Mpy =   1400000.00 kg.cm
   Rebar Spacing, X-Dir, Bottom = d16- 22.3 cm ( 0.28%)
   Rebar Spacing, X-Dir, Top    = d16- 40.0 cm ( 0.16%)
   Rebar Spacing, Y-Dir, Bottom = d16- 22.3 cm ( 0.28%)
   Rebar Spacing, Y-Dir, Top    = d16- 40.0 cm ( 0.16%)
```

TIE BEAM DESIGN:

```
Tie Beam / Sloof Width,               B =   30.00 cm
Tie Beam / Sloof Width,               H =   60.00 cm
Factored Maximum Column Axial Load,  Pu =   79103.45 ton
10% of Factored Axial Load,          Tu =    7910.34 ton
Required Rebar for Tension,         Ast =    2.54 cm2

Nett Uplift Height,                  Hw =    0.00 m
Tie Beam / Sloof Length,              L =    8.00 m
Tie Beam / Sloof Tributary Width,     W =    6.00 m
Distributed Load on Tie Beam,        qL =    0.00 kg/m
Distributed Weight on Tie Beam,     qsw =  432.00 kg/m
Bending Moment,                     Mql =  276480.00 kg.cm
Shear Force,                        Vql =    1728.00 kg
Req. Rebar for Bending Moment,   Bottom =    4.02 cm2
Req. Rebar for Bending Moment,      Top =    1.87 cm2

Longitudinal Rebar, at Support =  3 d16 /  2 d16
Longitudinal Rebar, at Midspan =  2 d16 /  3 d16
Shear Reinforcement Spacing at Support = d13 -    0.00
```

2. Example 11: Simple Slope Stability Analysis
3. Example 12: Soldier Pile Design using Linear Spring

10. Advanced Building Design
    1. Example 12: Stage Construction Sequential Loading Analysis
    2. Example 13: Concrete Shearwall Design
    3. Example 14: Drift, P-Delta, Story Shear Checking
    4. Example 15: Dual System Design Checking
    5. Example 16: Diaphragm Chord and Collector Design
    6. Example 17: Pile-Raft Foundation Design

11. Steel Building Design
    1. Example 18: Basic Steel Building Design
    2. Example 19: Bracing Design using Stage Construction Sequential Loading
    3. Example 20: Beam-Column Steel Connection Design
    4. Example 21: Steel Capacity Design
    5. Example 22: Beam with Tapered Section

12. Nonlinear Analysis and Direct Integration Analysis
    1. Example 23: Soldier Pile with Nonlinear Spring
    2. Example 24: Geometrically and Materially Nonlinear Truss Analysis
    3. Example 25: Dynamic Direct Impact Load Analysis

13. Special Structures Design (Optional)
    1. Example 26: B-Panel Building Design
    2. Example 27: Cold-formed Steel Building Design
    3. Example 28: Blast Door Design

14. Additional Design Tools (Optional)
    1. Example 29: Concrete Section Design
    2. Example 30: Concrete Beam-Column Joint Design
    3. Example 31: Concrete Rebar and Cutting Plan
    4. Example 32: Concrete Shearwall Design
    5. Example 33: Concrete Retaining Wall Design
    6. Example 34: Concrete Footing Design
    7. Example 35: Concrete Pilecap Design
    8. Example 36: Steel Database and Design Search
    9. Example 37: Steel Purlin Design
    10. Example 38: Steel Connection Design