3.4. My First Building Project

This is your first tutorial on how to use SANSPRO effectively to design a building. This tutorial will highlight most useful commands and options.

In this tutorial we assume that you have already finished **My First Frame Project** tutorial. Please do it if you have not done it because it contains many basic tasks that will be used here and not be explained again in detail.

You can finish this tutorial in 30 minutes to 1 hour.
Please follow the step-by-step instruction below carefully, remember that you have plenty of time, do not be in hurry.

Static Equivalent Analysis:

1. Create New Model
2. Define Structural Analysis Option
3. Define Working Area and Axis
4. Define Building Parameters
5. Adding Load combinations
6. Adding Element Properties
7. Define Floor Slab Data
8. Adding Nodes
9. Adding Beam Elements
10. Adding Column Elements
11. Adding Shearwall Elements
12. Adding Support
13. Adding Floor Regions
14. Adding Beam Load
15. Compute Floor Weight
16. Generating Equivalent Load
17. Run Analysis
18. Output View
19. Element Design
20. Design View
21. Element Design View
22. Design Color Code
23. Invalidate Design
24. Revise Design
25. Printing Report
26. Foundation Design
27. Compute Volume and Cost
28. View Rebar Layout
29. Generating Drawing

Dynamic Analysis:

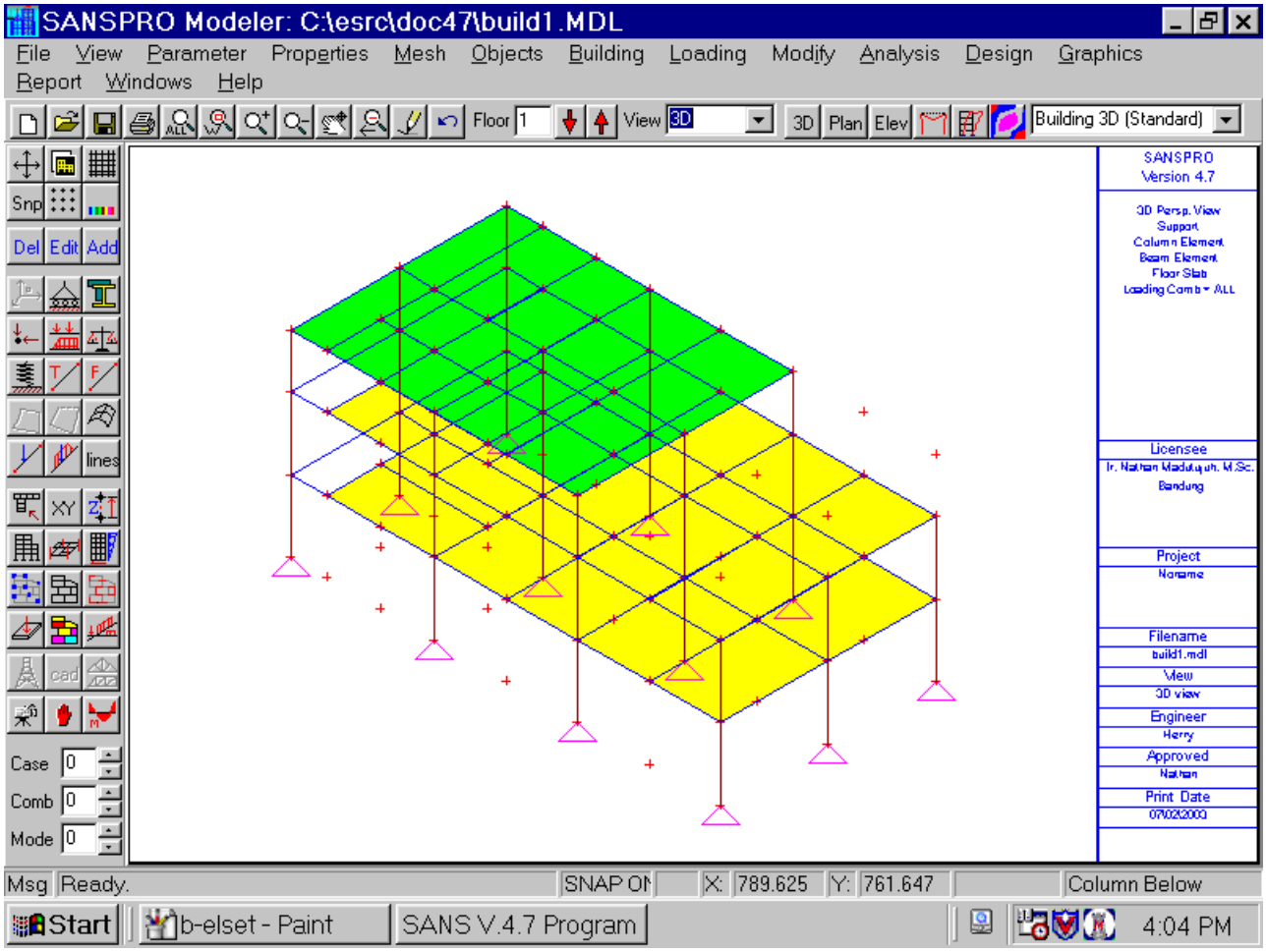
1. Define Mass Factor
2. Define Spectrum Response
3. Dynamic Analysis Option
    - Number of Eigen
    - Earthquake Direction
    - Static Condensation
4. Run Dynamic Analysis
5. Check Dynamic Output
    - Natural Period
    - First Mode/Dominant Mode
    - Mode Shape
    - Mass Participation Factor
    - Effective Mass Factor
    - Modal Direction Factor
    - Base Shear
    - Eccentricity
    - Displacements/Drift
6. Eccentricity
    - Eccentricity
    - Design Eccentricity
7. Base Shear
    - Base Shear
    - Base Shear Correction
    - Story Shear
8. Design for Dynamic Analysis
9. Visual Design Report

Capacity Design

1. Basic Concepts
2. Define Parameters
3. Run Capacity Design
4. Visual Design Report
5. Capacity Design Report: Beam Design Forces
6. Capacity Design Report: Beam Rebar
7. Capacity Design Report: Column Design Forces
8. Capacity Design Report: Column Rebar

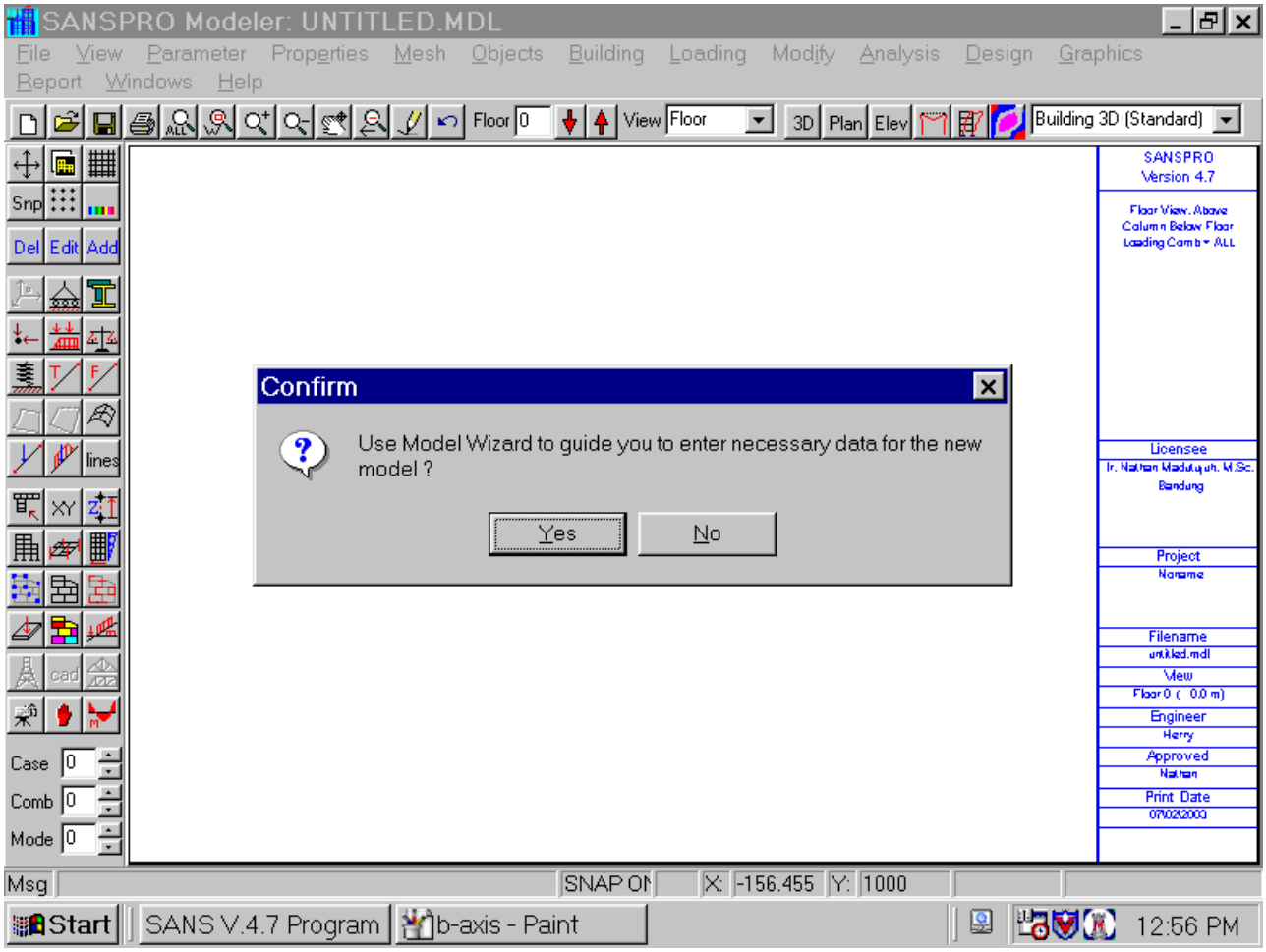
Problem Definition:

- Design a three storeys concrete building for residential purpose

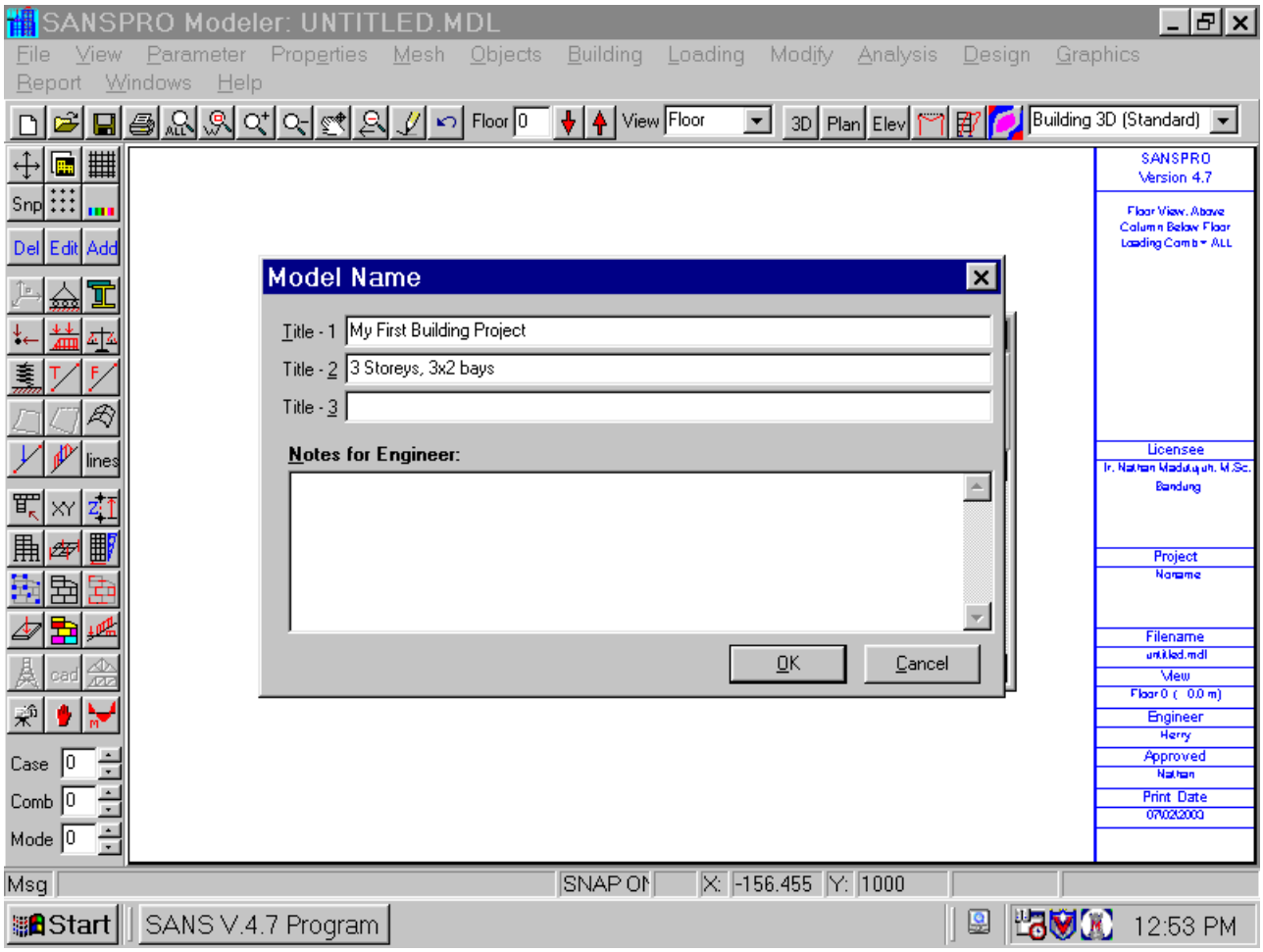


1. Create New Model

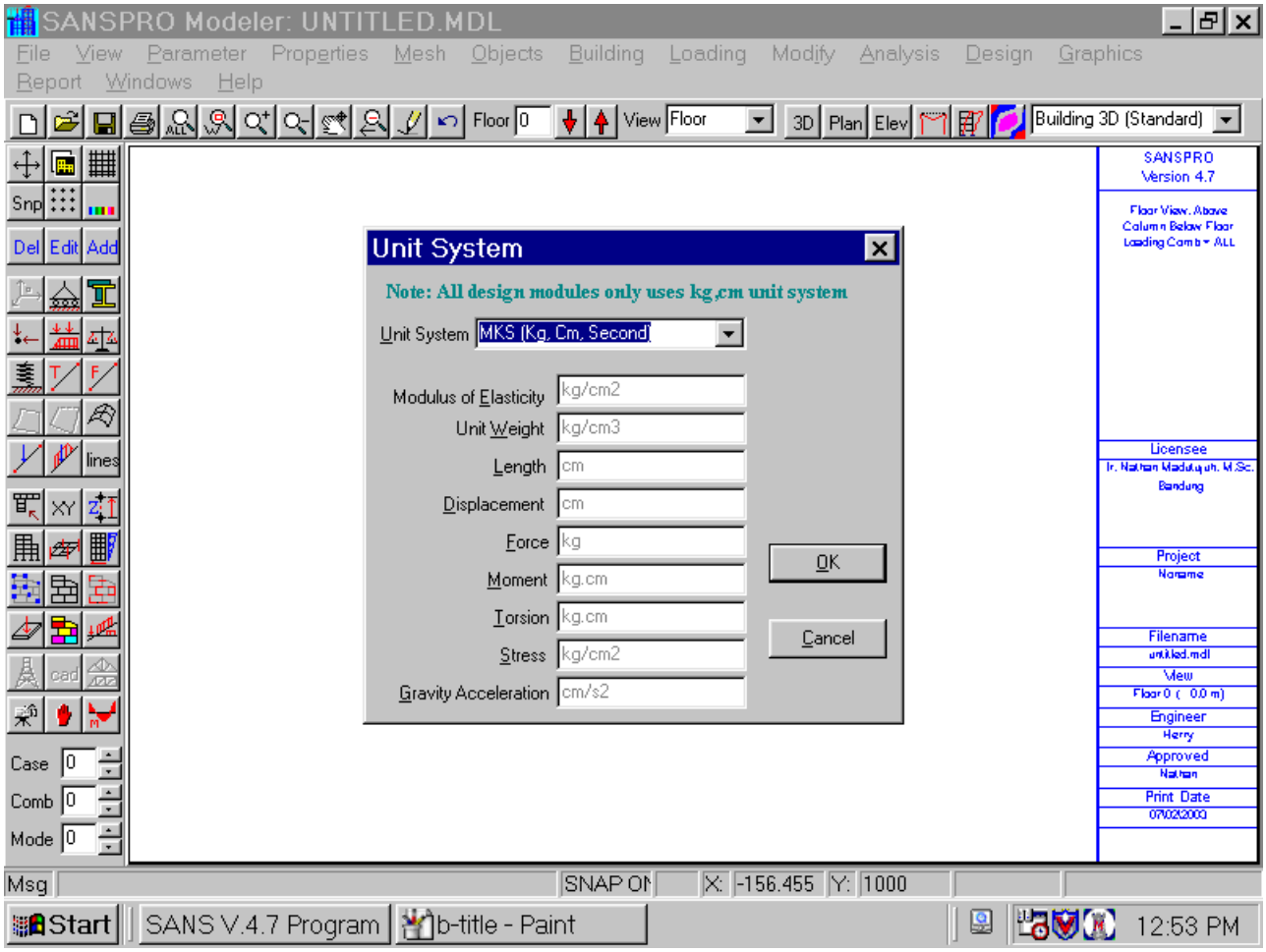
- Use Wizard to guide you step-by-step



- Click File-New
- Entering Model Name and Titles
- Title 1 can be used for Project Title
- Title 2 can be used for Job Title
- Title 3 can be used for Project Location
- Notes for Engineer can be used for notes to be shared with other engineers.

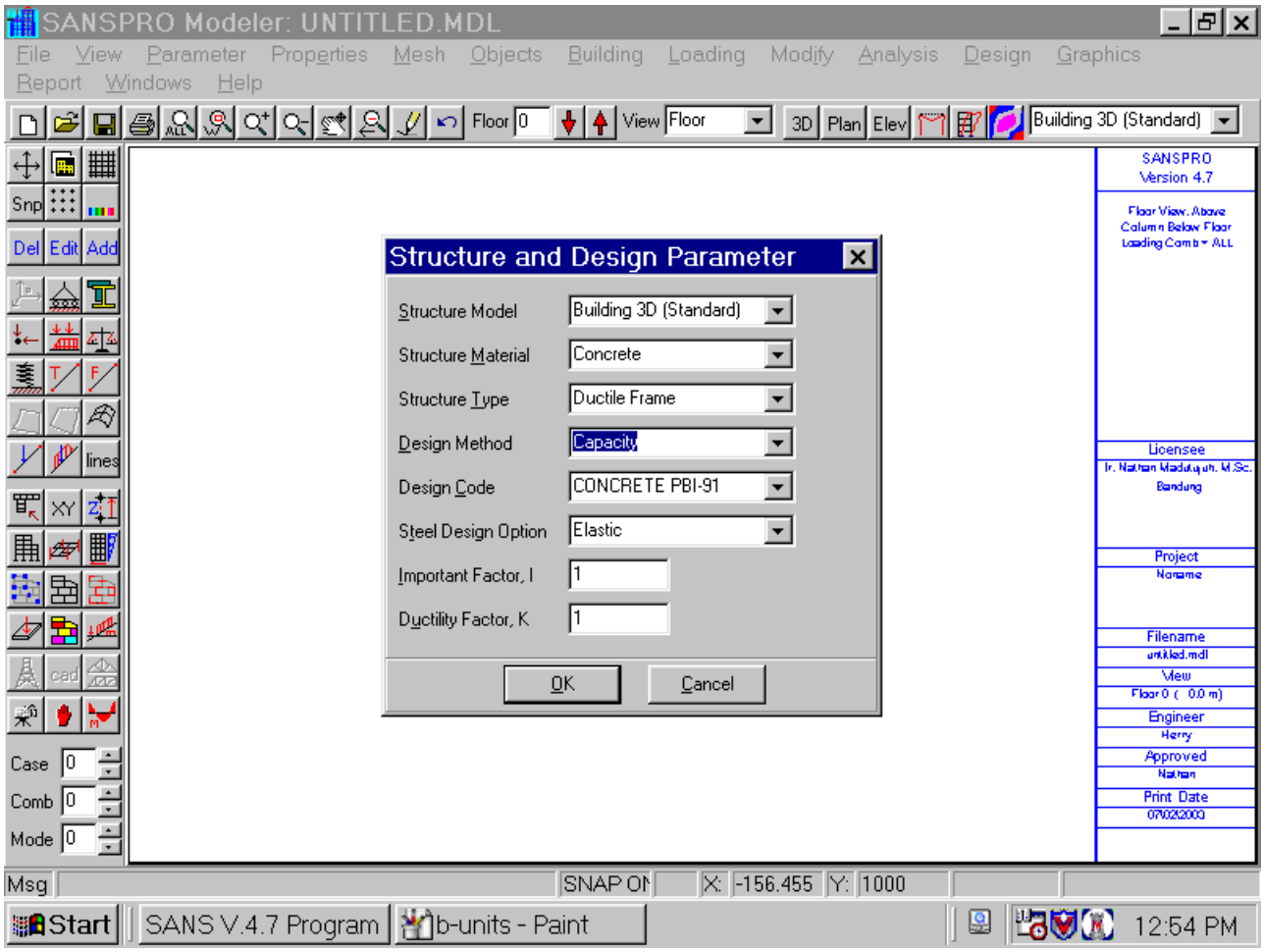


- Select Unit System = kg,cm



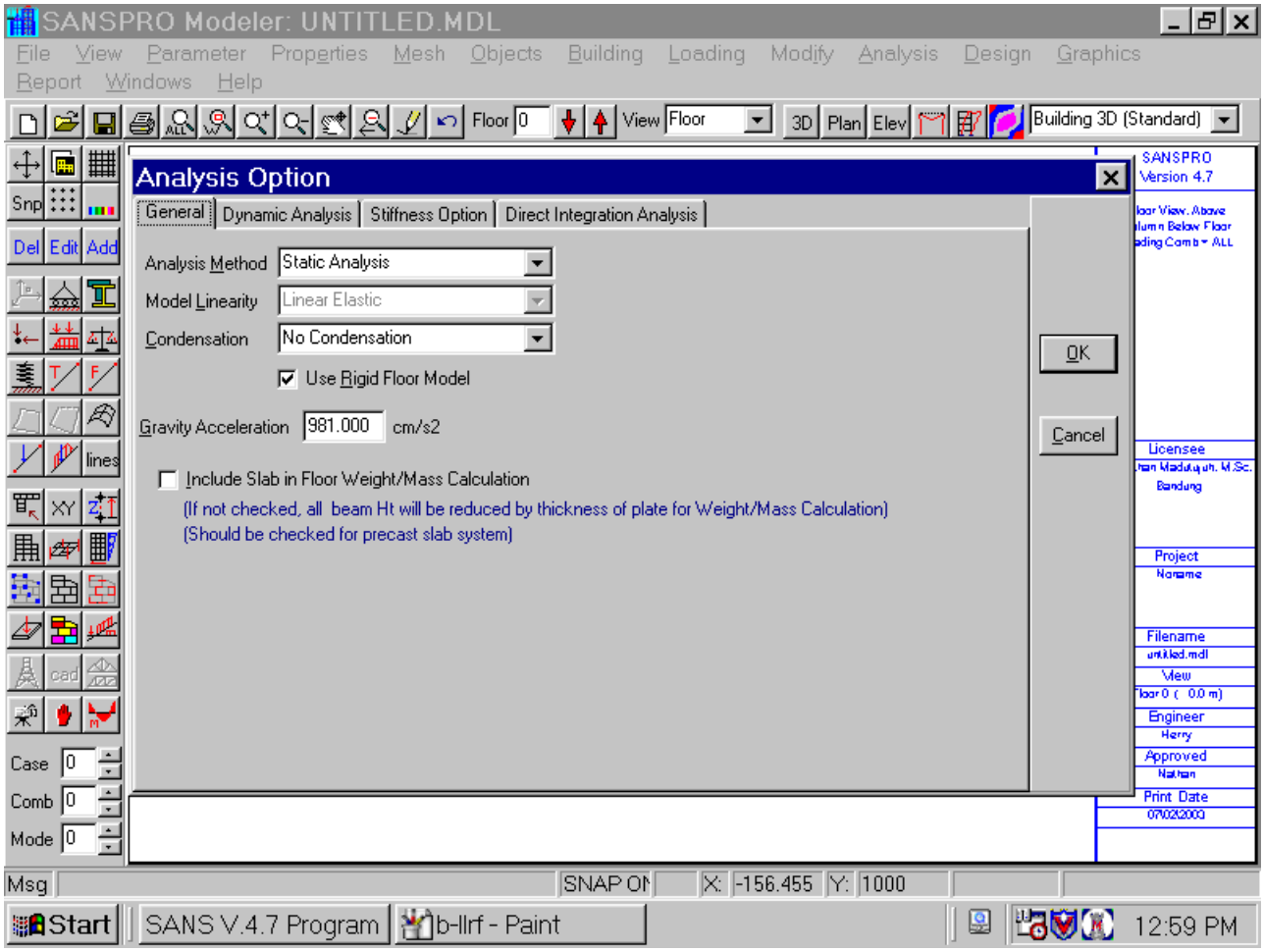
2. Define Structural Analysis Option

- Structural Model = Building 3D (Standard)
  Select Building 3D (Extended) if you want to add 3D Trusses/Frames/Domes.
- Structure Material = Concrete
- Structure Type = Ductile Frame
- Design Method = Capacity
- Design Code = CONCRETE PBI-91 (Indonesian Code)
- Steel Design Option = Elastic (Not Applicable for this concrete building)
- Importance Factor of the building = 1.0 (Standard)
- Ductility Factor K = 1 (K is used for Cd = C.I.K)



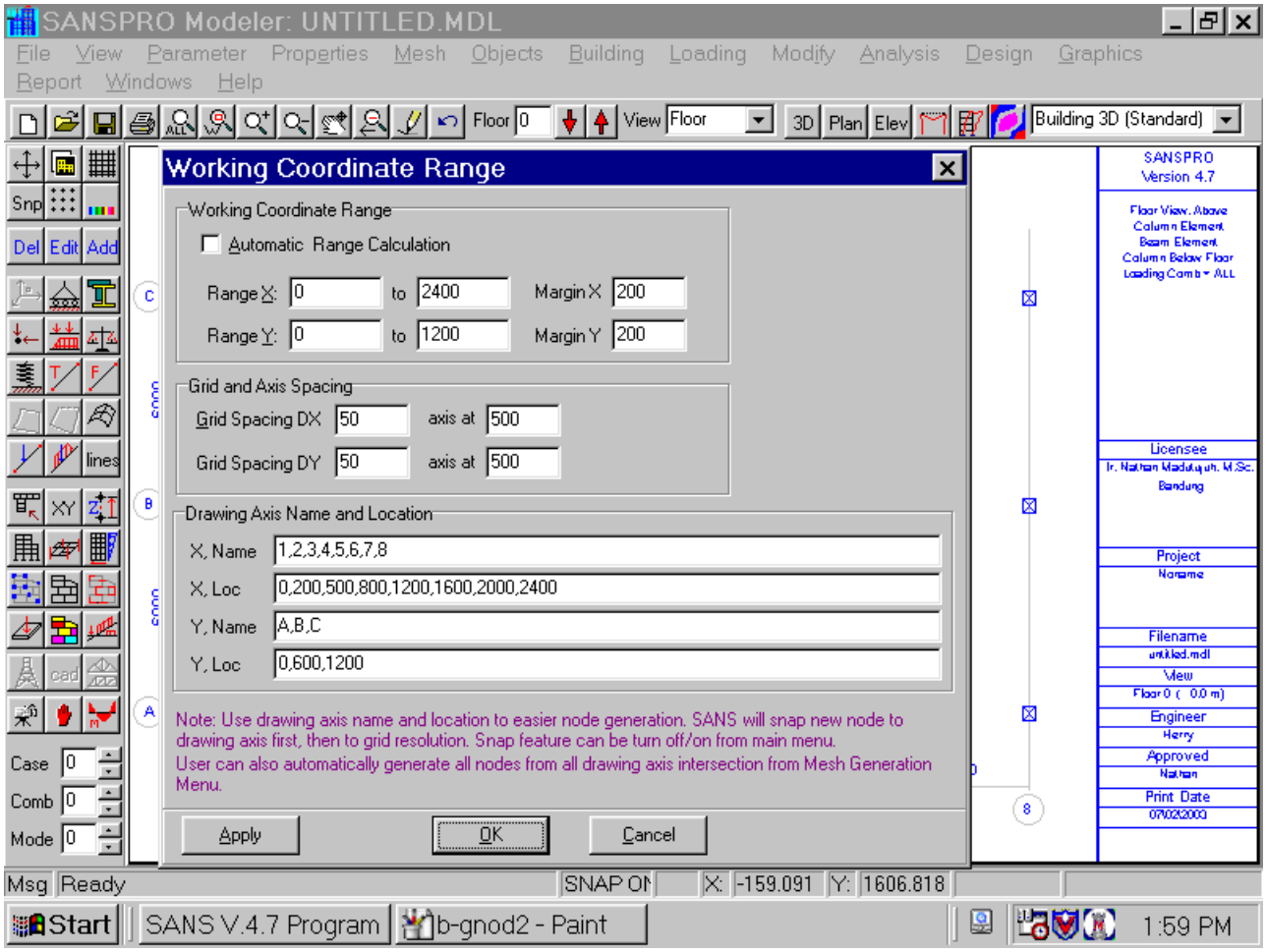
- Select Analysis Type = Static
- Model Type = Linear Elastic
- Condensation = No
    - Select Condensation = No, Rigid Floor = No for Open Frame
    - Select Condensation = No, Rigid Floor = Yes for Open Frame with Rigid Floor
    - Select Condensation = 3 Dof/Floor, Rigid Floor = Yes for 3 Dof/Floor Model

- Rigid Floor Model = YES
- Gravity Acceleration = 981 cm/second^2
- Uncheck Include Slab in Beam Contribution for Floor Weight/Mass Calculation
    - If checked, beam weight will be calculated using total beam height (should be selected for Precast slab system)
    - If not checked, beam height will be reduced by slab thickness for weight calculation (use this for cast-in-situ beam-slab system)

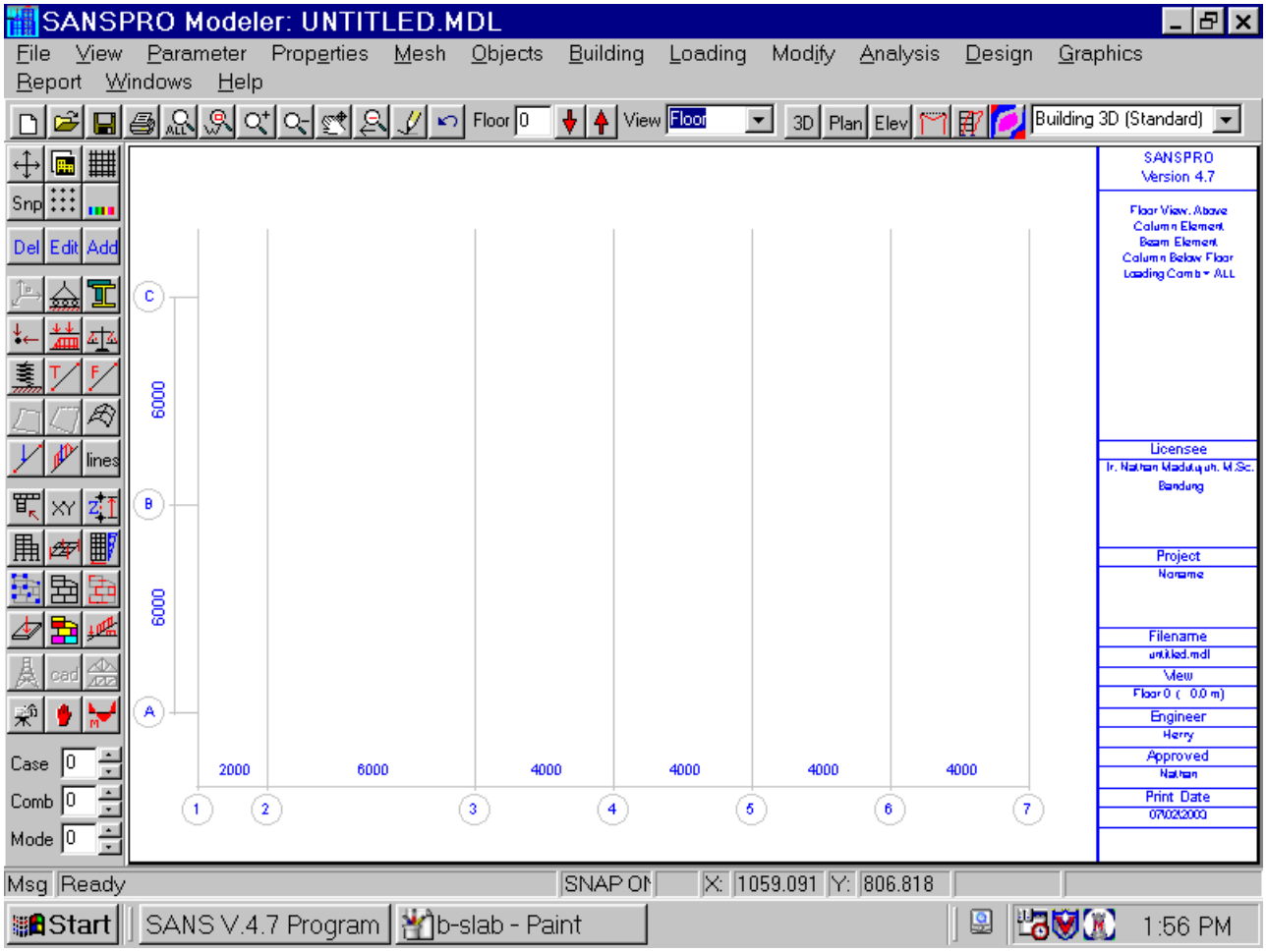


3. Define Working Area and Axis

- Define working area as X and Y coordinate ranges of building layout view from top.
- Here our X range is from 0 to 2400cm, Positive (+) direction is from left to right
- Here our Y range is from 0 to 1200cm , Positive (+) direction is from bottom-up
- Grid axis spacing is 500 (Coordinate will be snapped to grid axis system if Snap = on)
- We have 8 user define axis names and location for X axis and 3 for Y axis.
- If Snap on then every coordinate will be snap to user axis intersection, then to grid spacing, automatically.

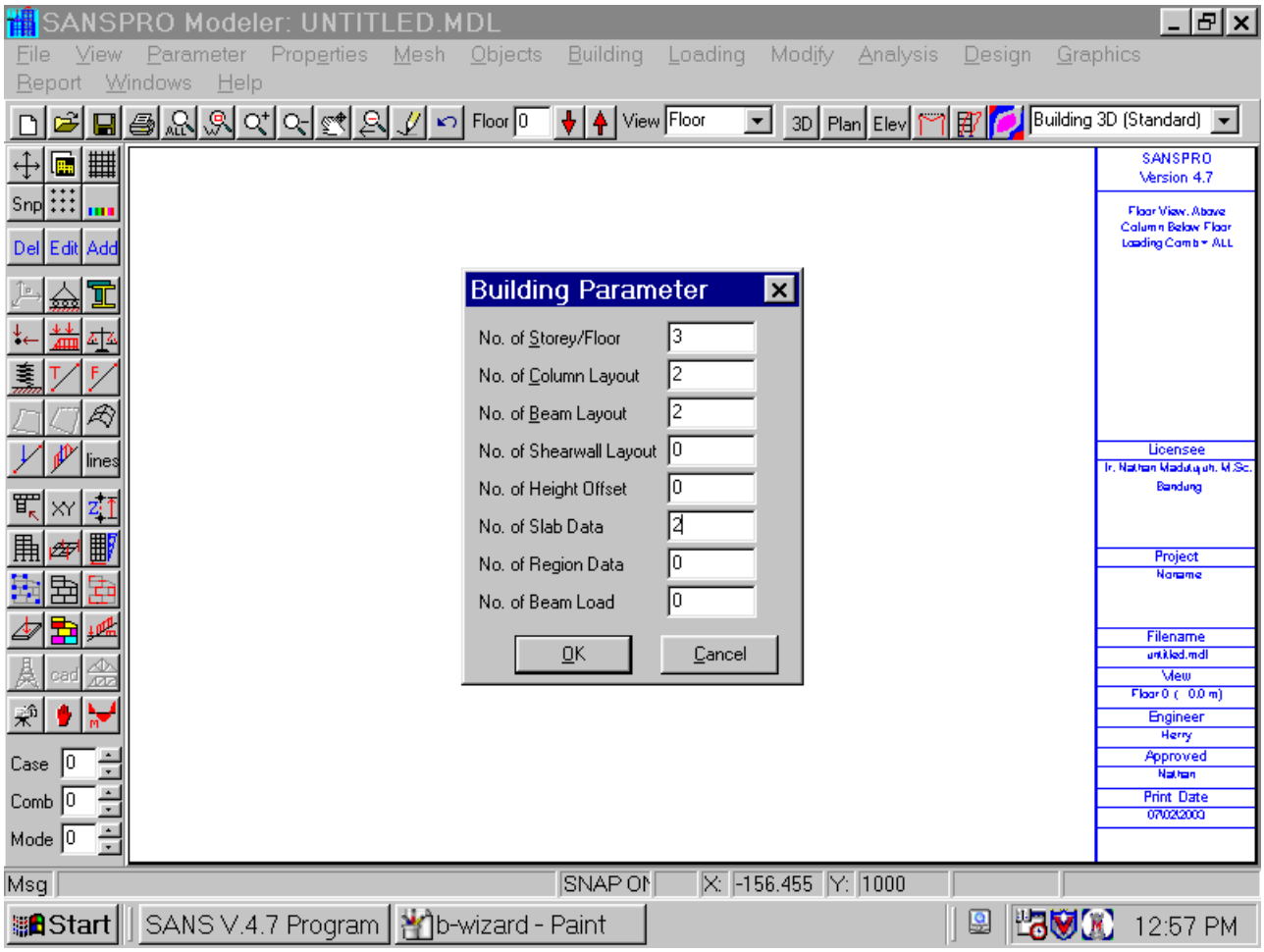


- User axis can be shown/hidden by clicking Icon User Axis.

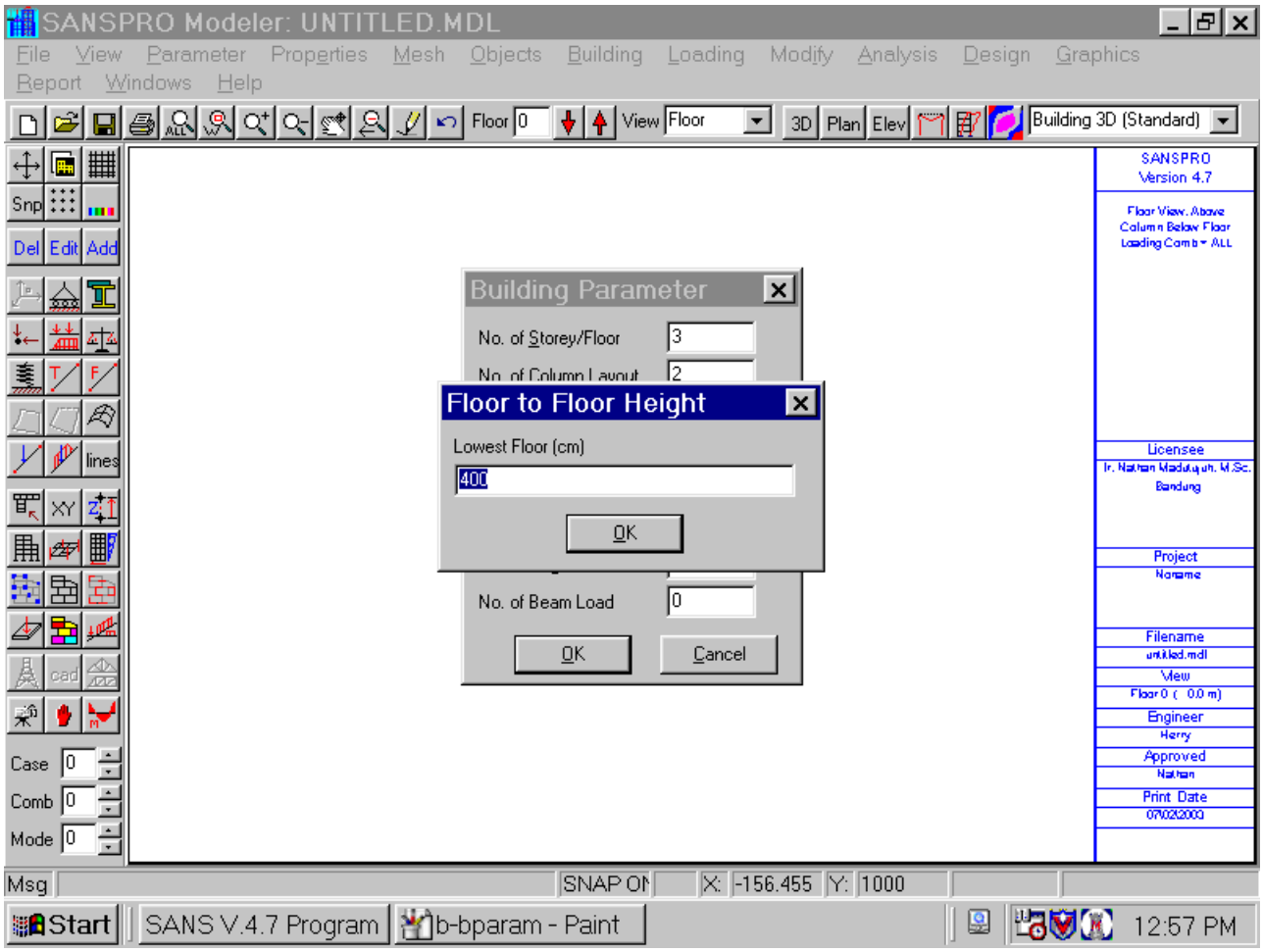


4. Define Building Parameters

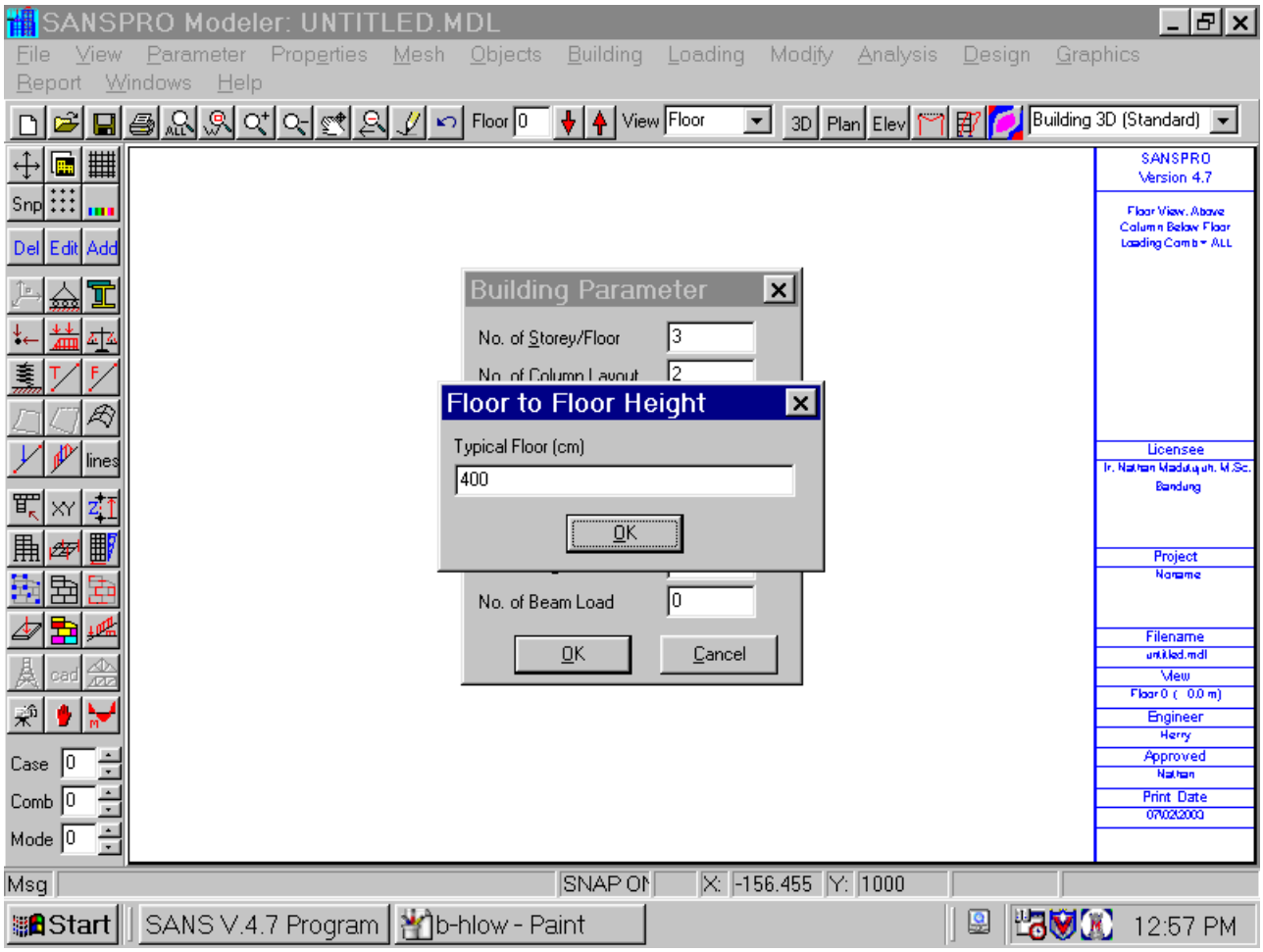
- Define Buiding Parameters
    - Number of Floor = 3
      (Start from Floor 0 at ground to Floor 3 at top)
    - Number of Column Layout = 2
        - number of different types of column size and layout changes for a floor
        - Each column layout can contain more than 1 column elset type
    - Number of Beam Layout = 2
        - number of different types of beam size and layout changes for a floor
        - Each column layout can contain more than 1 column elset type
    - Number of floor slab data = 2
        - We will use two slab thickness: 12cm, 10cm
    - Leave other fields blank, they will be filled by SANSPRO



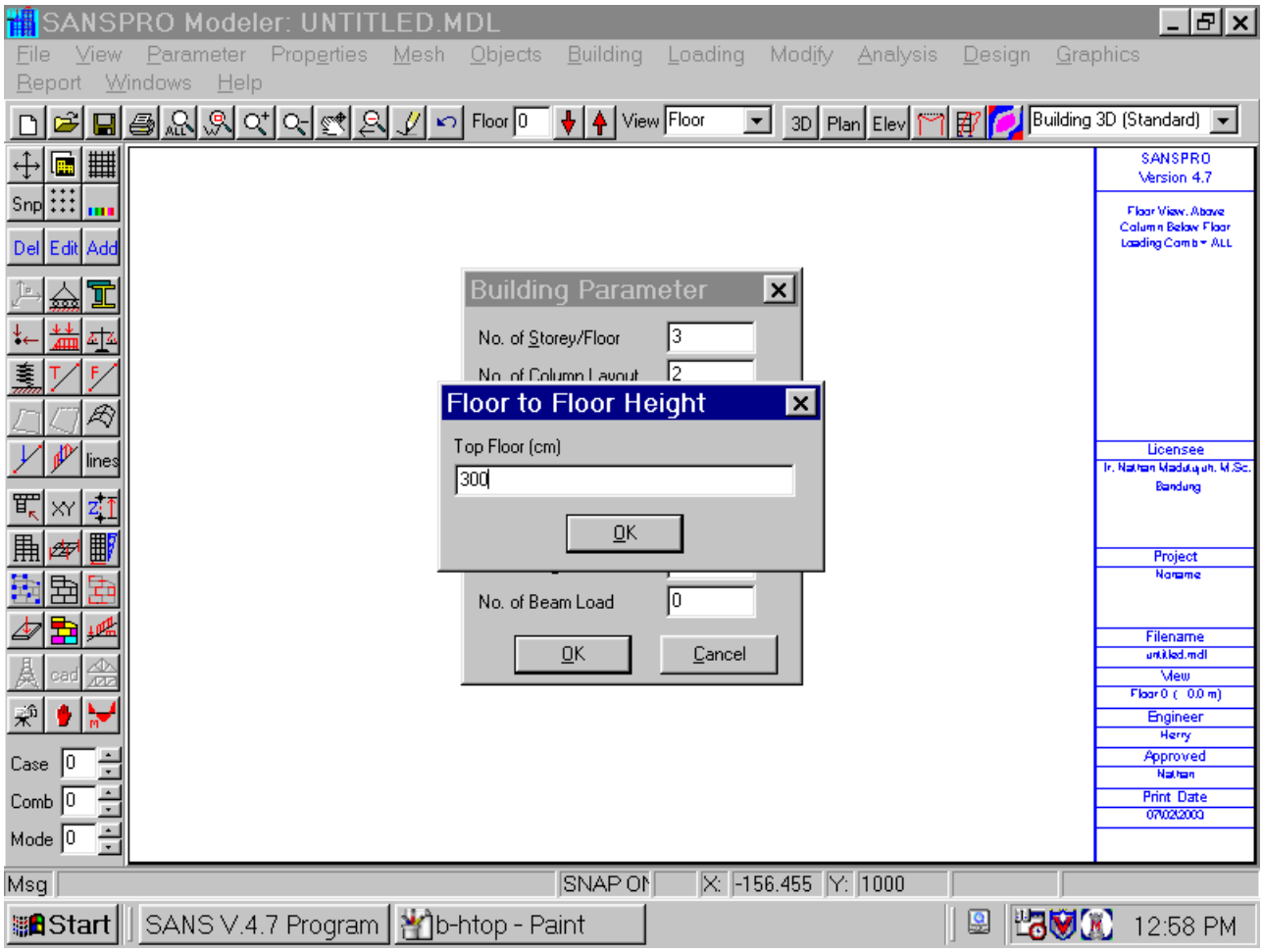
- Enter floor height for lowest floor = 400cm
- You can change each floor height later from STOREY DATA table



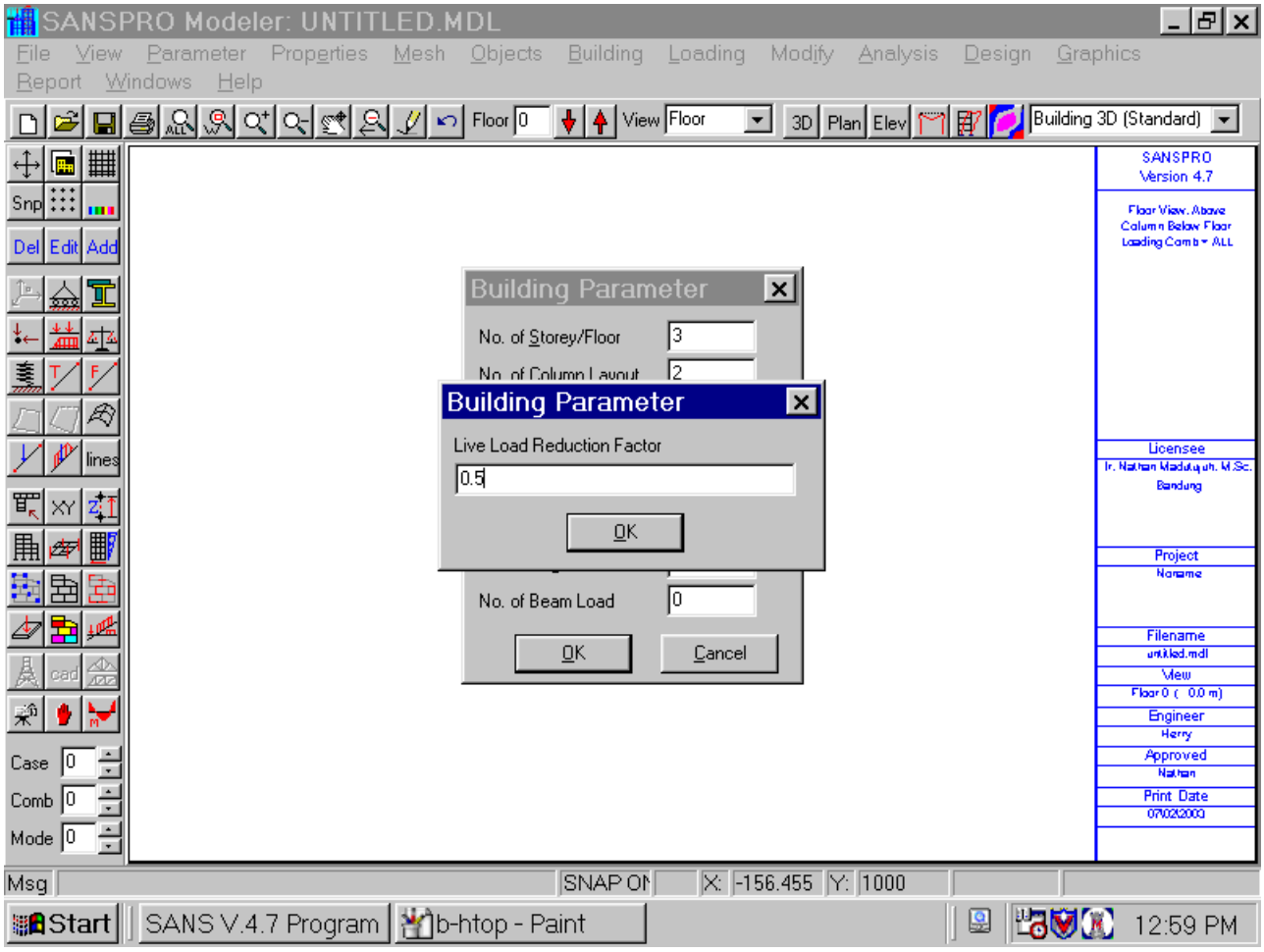
- Enter floor height for typical floor = 400cm
- You can change each floor height later from STOREY DATA table



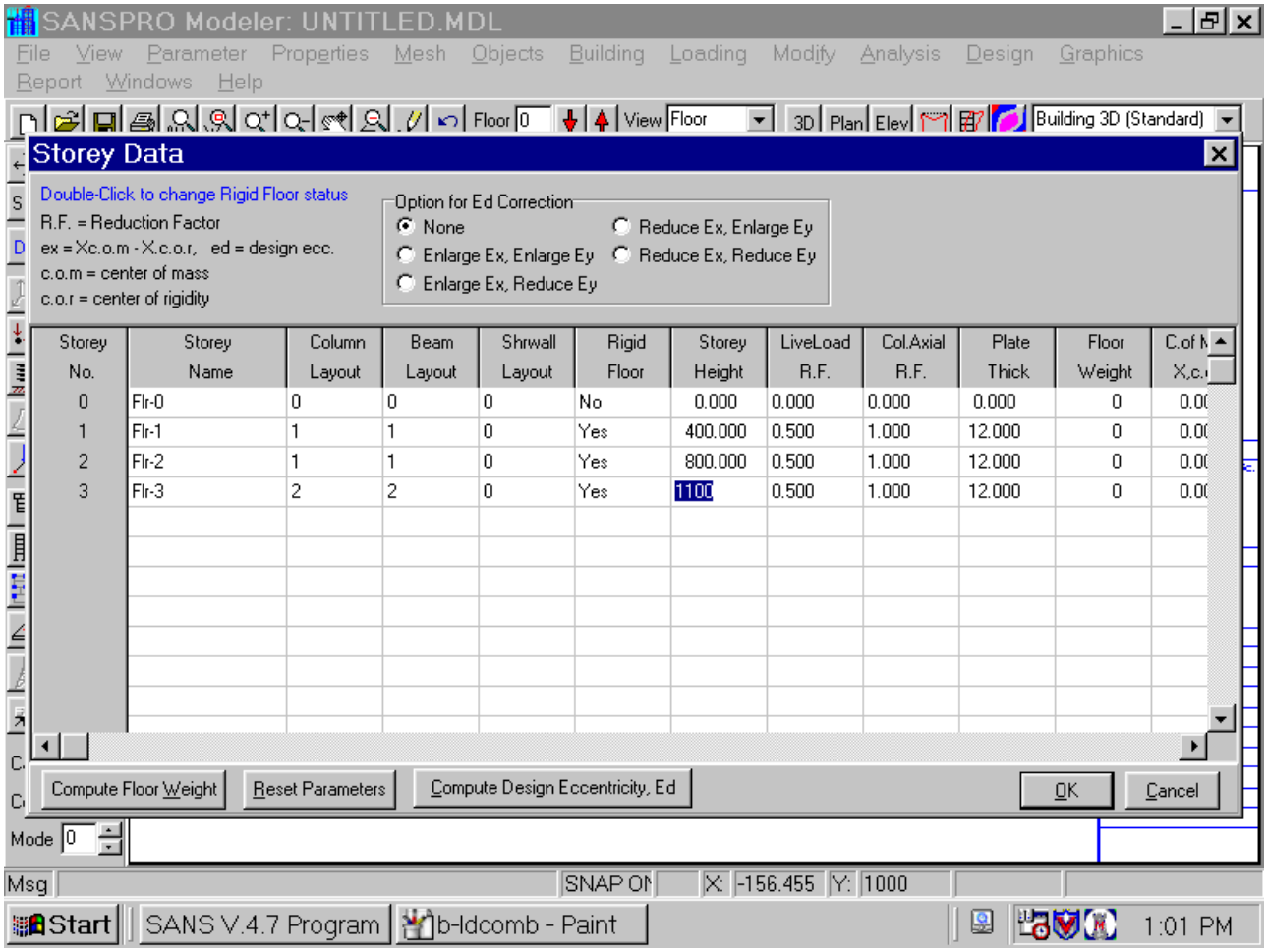
- Enter floor height for top most floor = 300cm
- You can change each floor height later from STOREY DATA table



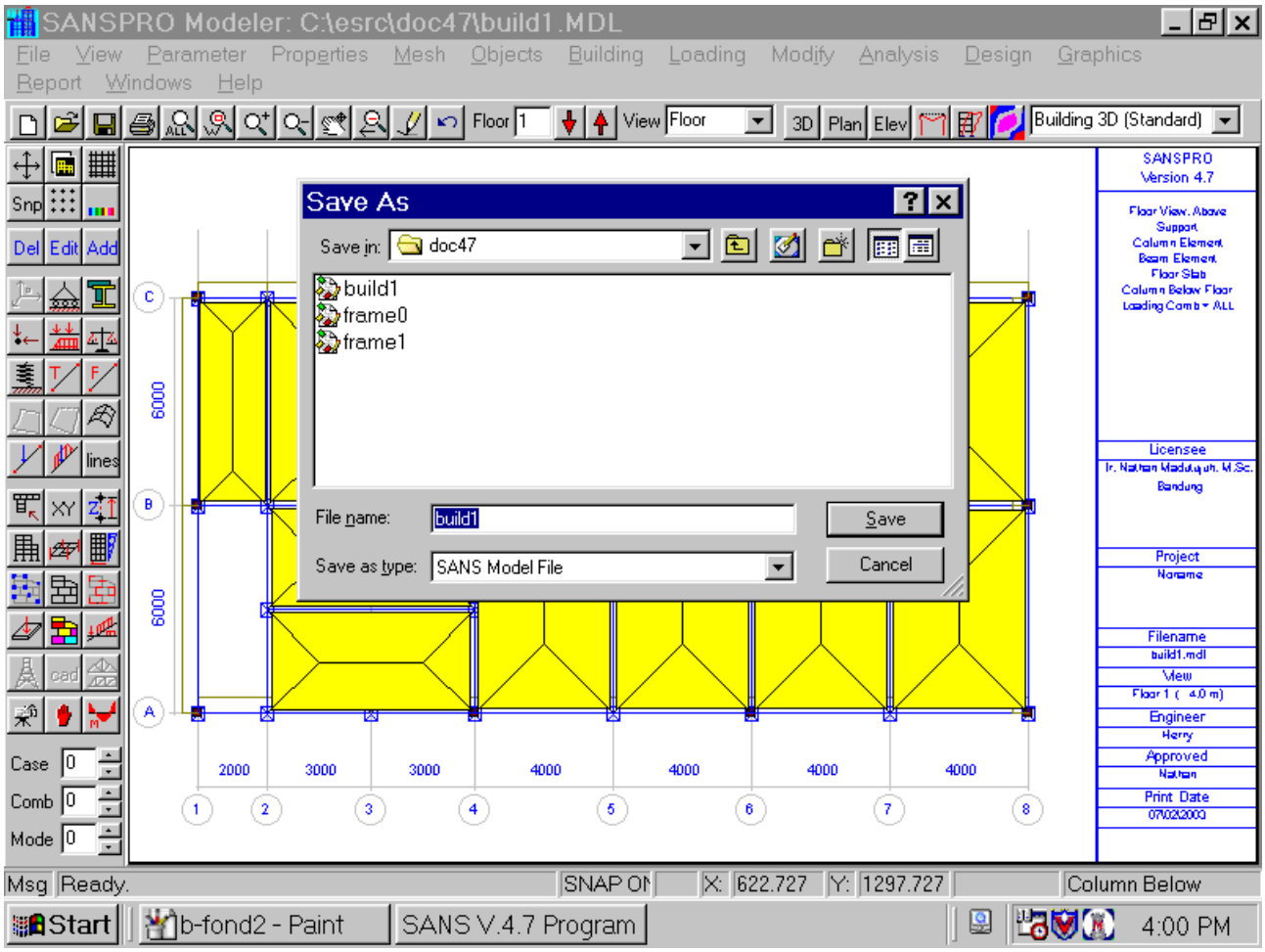
- Enter Live Load Reduction Factor for all floors, LLRF = 0.5 (means 50% LL reduction)
- You can change each floor LLRF later from STOREY DATA table
- This LLRF will be used to reduce Live Load for Floor Weight Calculation for Static Equivalent Load Analysis



- Storey Data Table will be displayed
- You can change each floor parameters from this STOREY DATA table
- For this example, just click OK to accept the default values

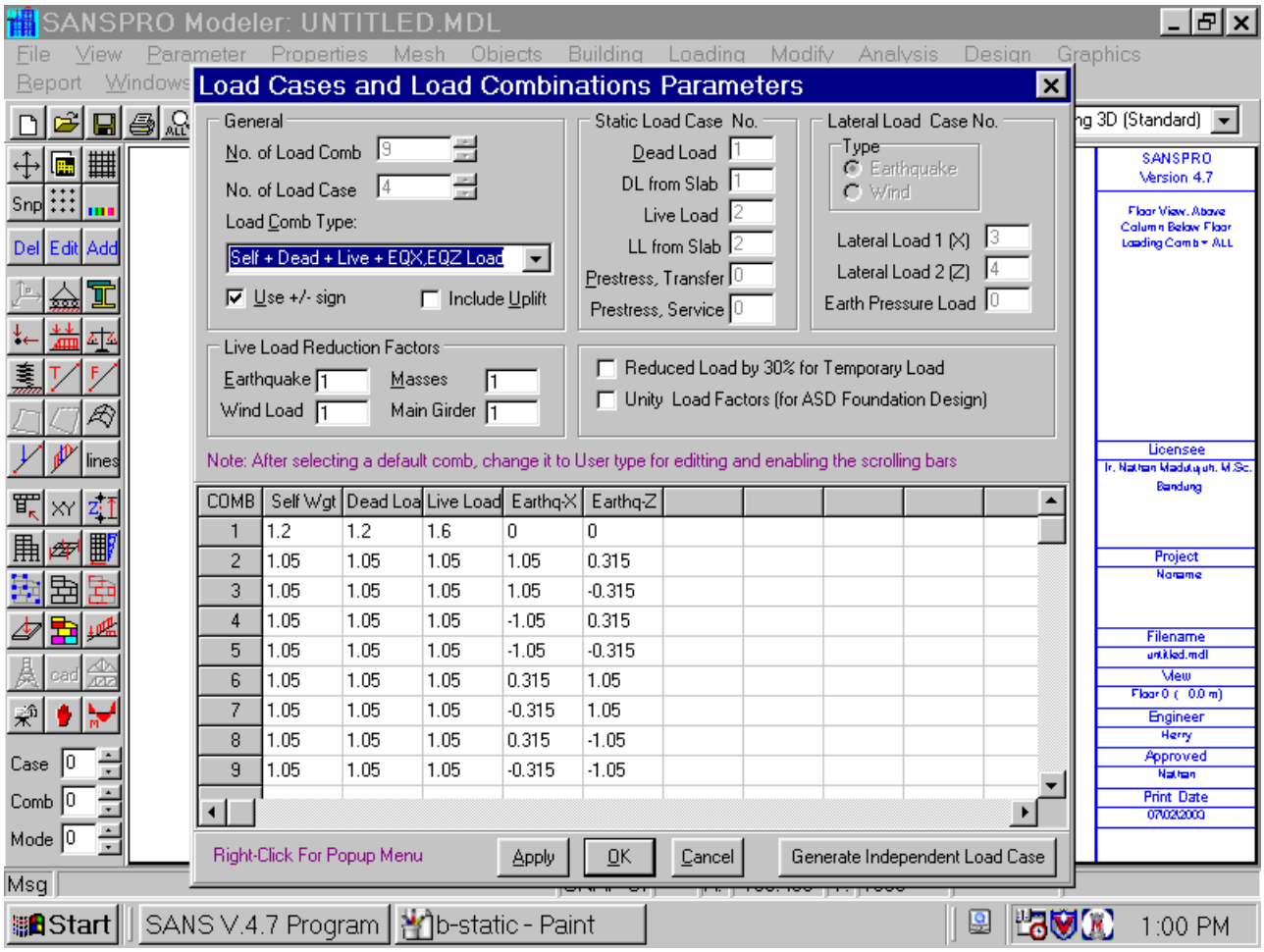


- Save the model and name it BUILD1.MDL
- After first saving, we can save the model by pressing F2
- It is recommended to save the model after major changes
- SANSPRO provides two backup files:
  - BUILD1.~MD - Created each time you save the model
  - BUILD1.SBK - Created automatically every certain duration (user defined)
- Automatic Save should be turn on (recommended) for your protection
- Click File-Preferences to see the status
- If something wrong with your model you can rename the backup file to BUILD1.MDL and open it again



5. Adding Load combinations

- SANSPRO provides predefined load combination sets for concrete and steel structures
- User can select **+/-** option to get earthquake in all directions
- User can use **Uplift** option to get extreme uplift combination (reduced DL)
- Select Load Comb Type = Self + Dead + Live + EQX, EQZ Load with +/- sign on
- Here we will have Load comb = 9, and Load case = 4



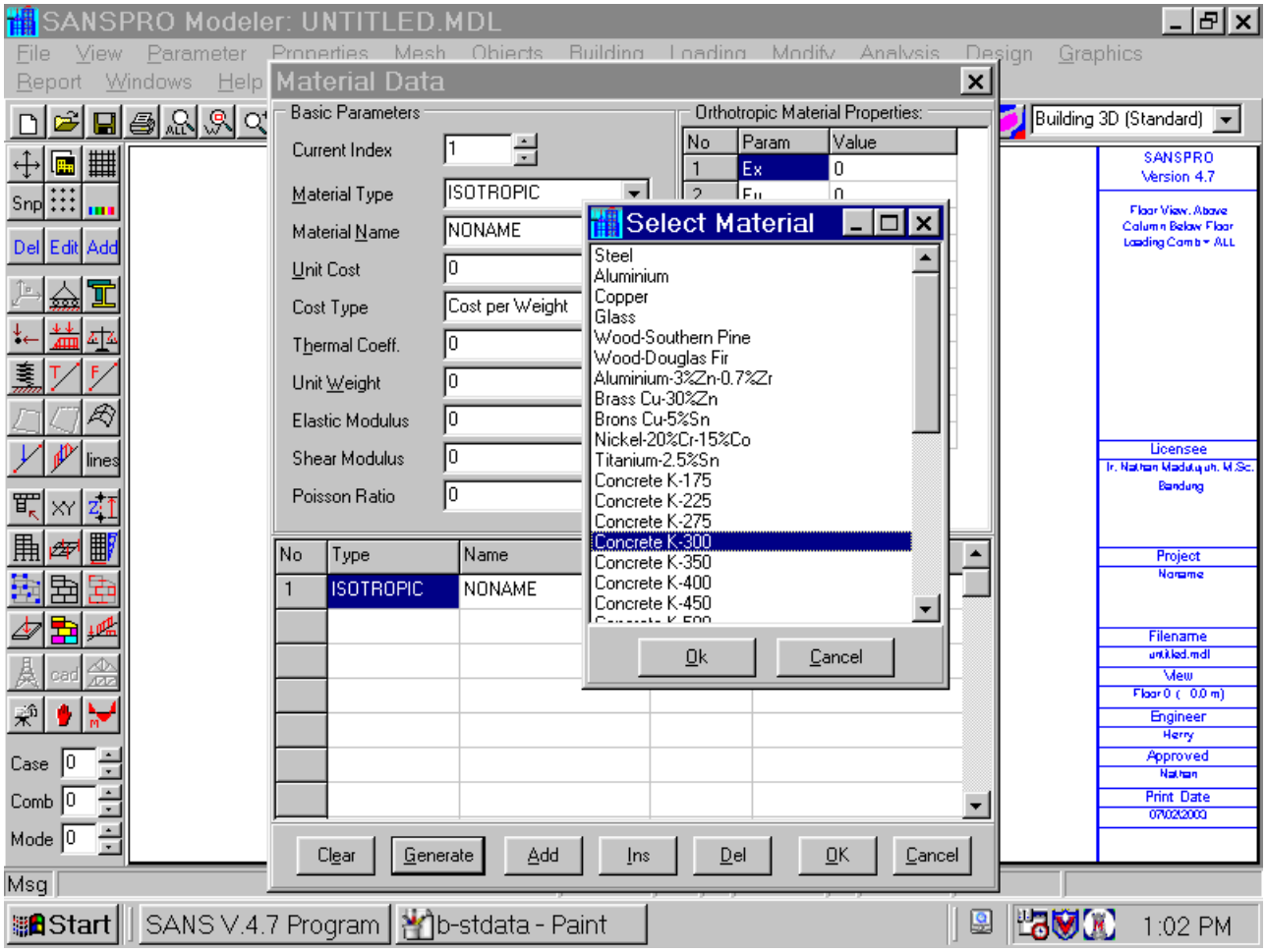
- Dynamic Load cases will not be counted as load case but will be included in load combination
- Use predefined load combinations carefully. If you want to modify it, please select the load comb type first then click User Defined.
- For Allowable Stress Design (ASD) as for steel or foundation design, user can reduced all load factors by 30% for temporary loadings and remove all load factors (change to unity).
- Live load reduction factors can be given according to local code. Please use this carefully.
- Usually live load reduction factor is used to reduce building mass for eathquake load calculation
- For live load factor at load combination, we usually use reduction factor for main girder only

6. Adding Element Properties

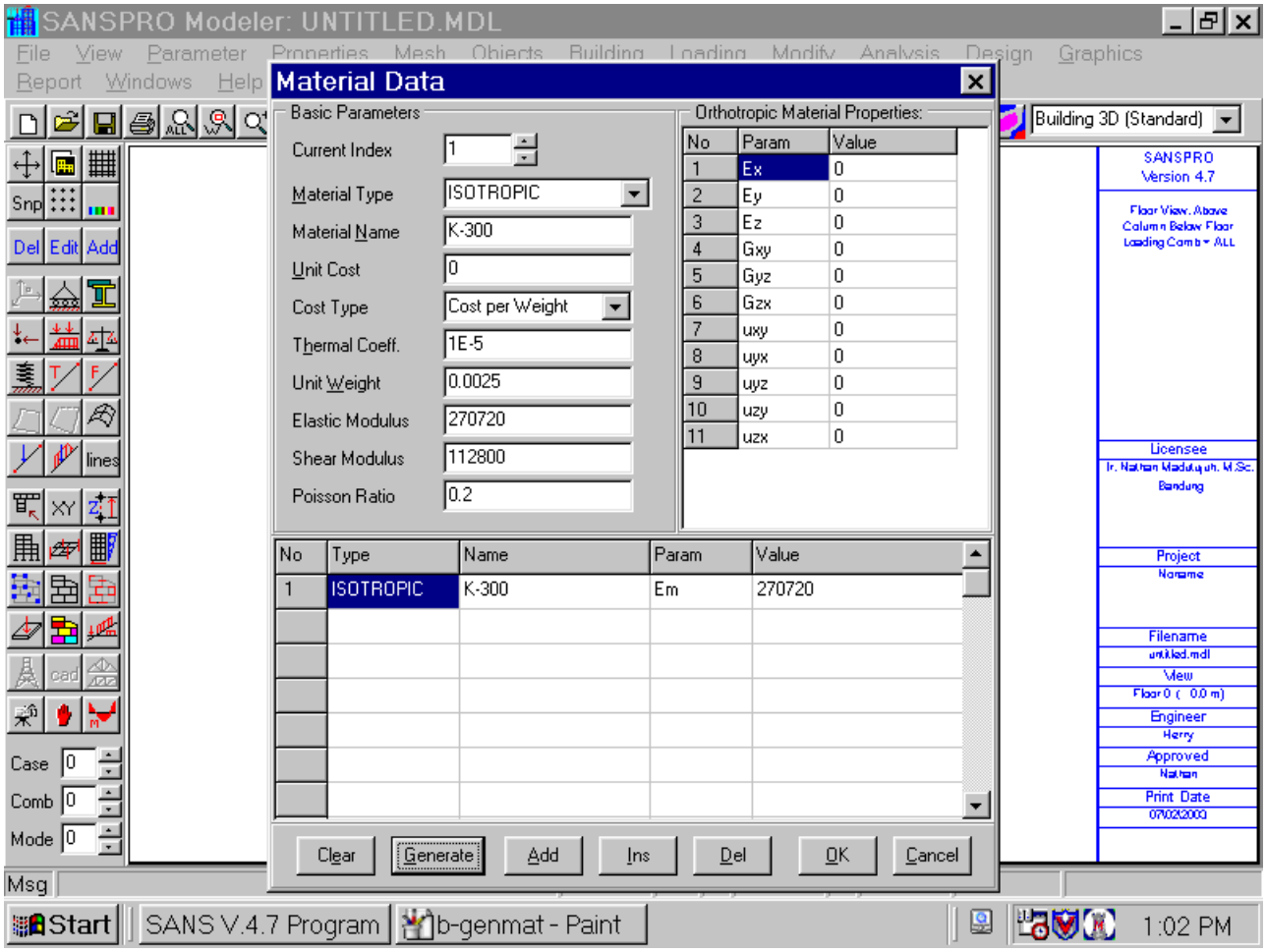
- We will define material, section, design properties and elset data

6.1. Generate Material Data

- Generate material data for Concrete K-300

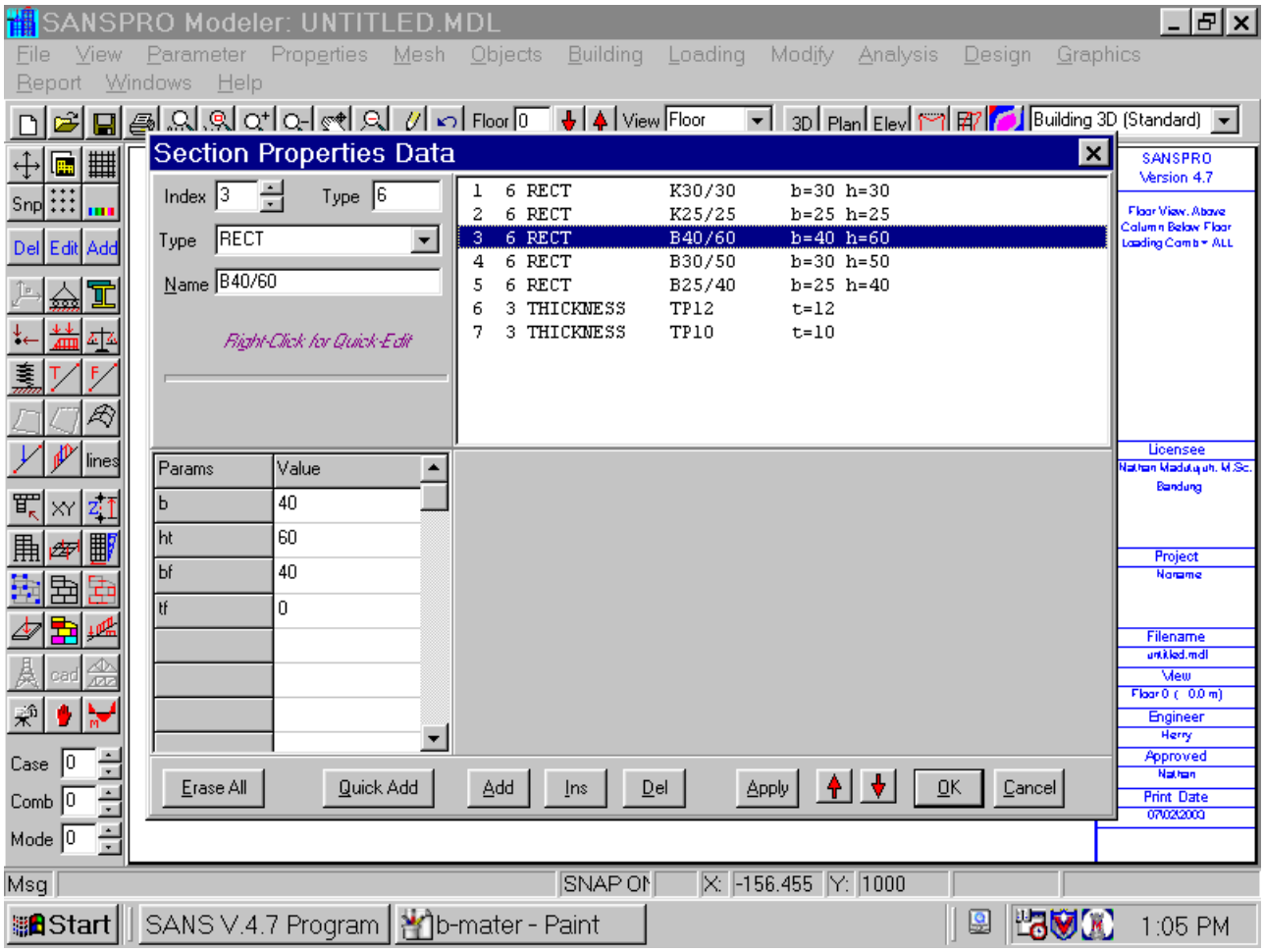


- Material Table will be as follows



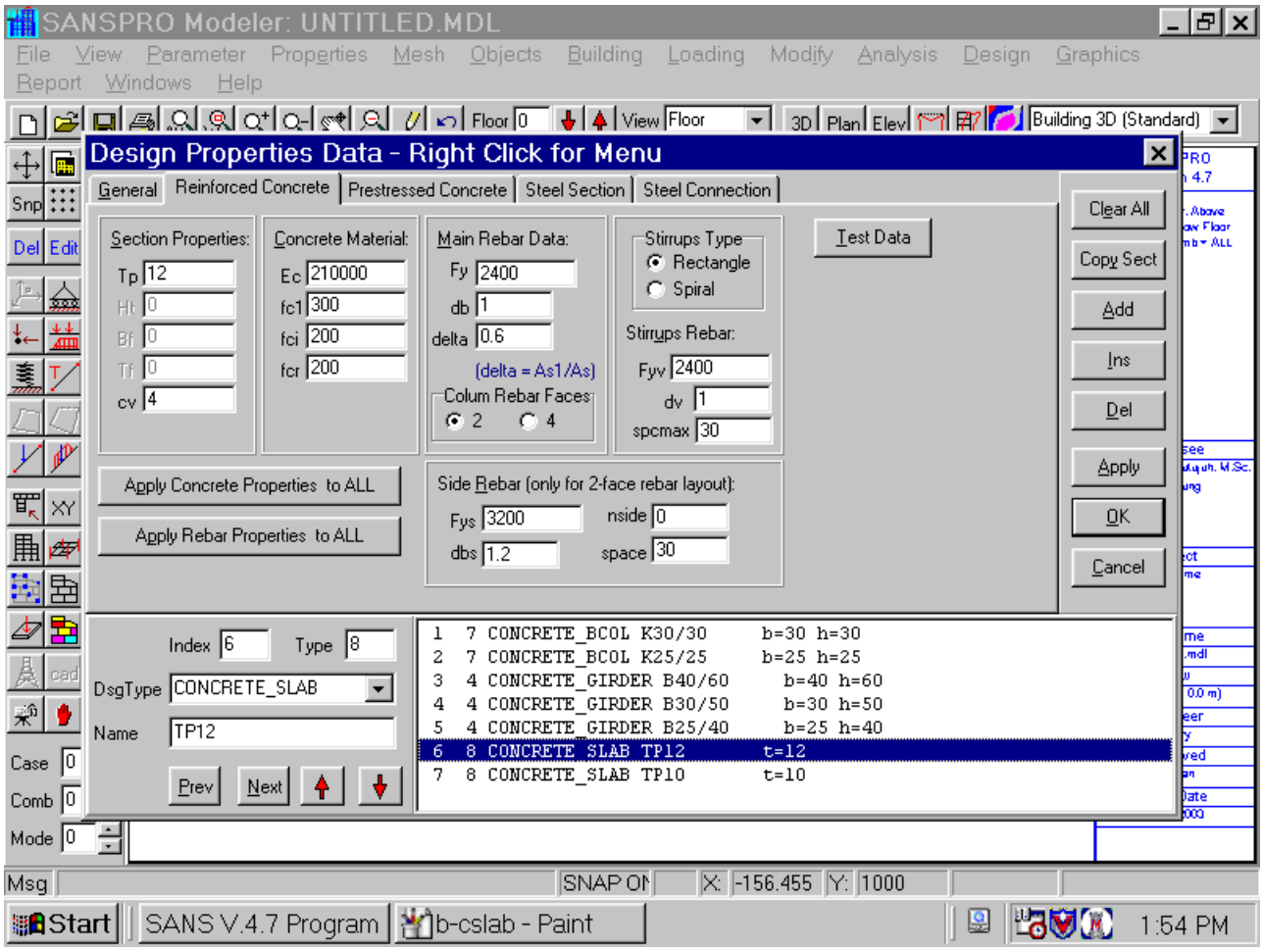
6.2. Define Section Data

- Define SECTION table for this building
- Click **Add** to add new section
- Here we use 7 sections, 2 for columns, 3 for beams, and 2 for floor slabs
- Section data at Section table will be used for Analysis phase only
- For beam sections, parameter bf (flange width) and tf (flange thickness) must be entered.
- If not a flanged beam, bf should be equal to b and tf = 0
- Avoid to use space for section names
- Beam and column sections are type RECT and Slab sections are type THICKNESS
- Click Apply or Dark bar to apply the changes made to table



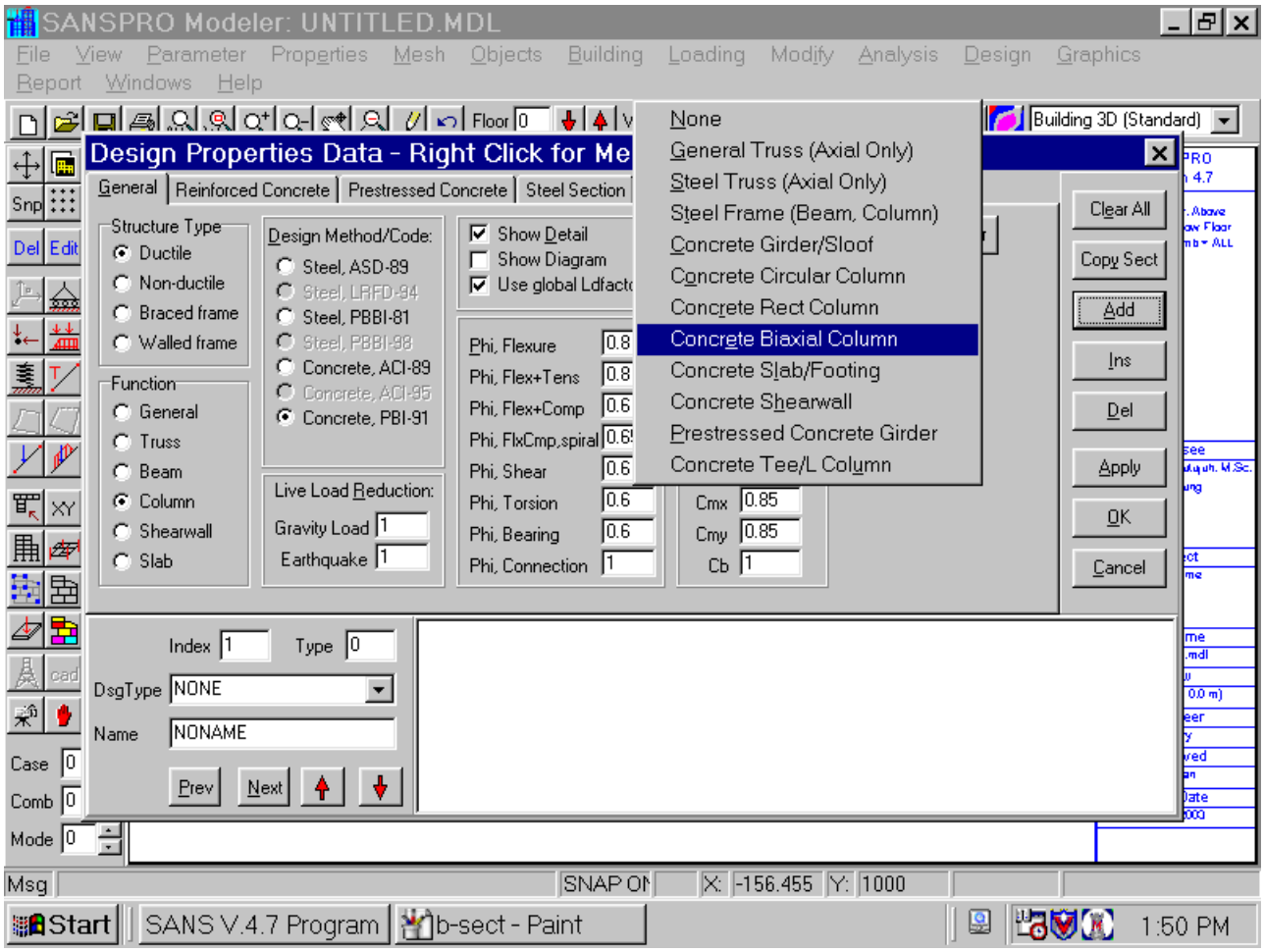
6.3. Define Design Data

- Define DESIGN properties data for this building
- Click **Add** to add new section
- Here we use 7 designs, 2 for columns, 3 for beams, and 2 for floor slabs
- Section data can be copied from Section Table by clicking button **Copy Section**
- Section data at Design table will be used for Design phase only
- For beam sections, parameter bf (flange width) and tf (flange thickness) must be entered.
- If not a flanged beam, bf should be equal to b and tf = 0
- Avoid to use space for design names
- Column designs are type Concrete Biaxial Column
- Beam designs are type Concrete Girder
- Slab designs are type Concrete Slab
- Click Apply or Dark bar to apply the changes made to table

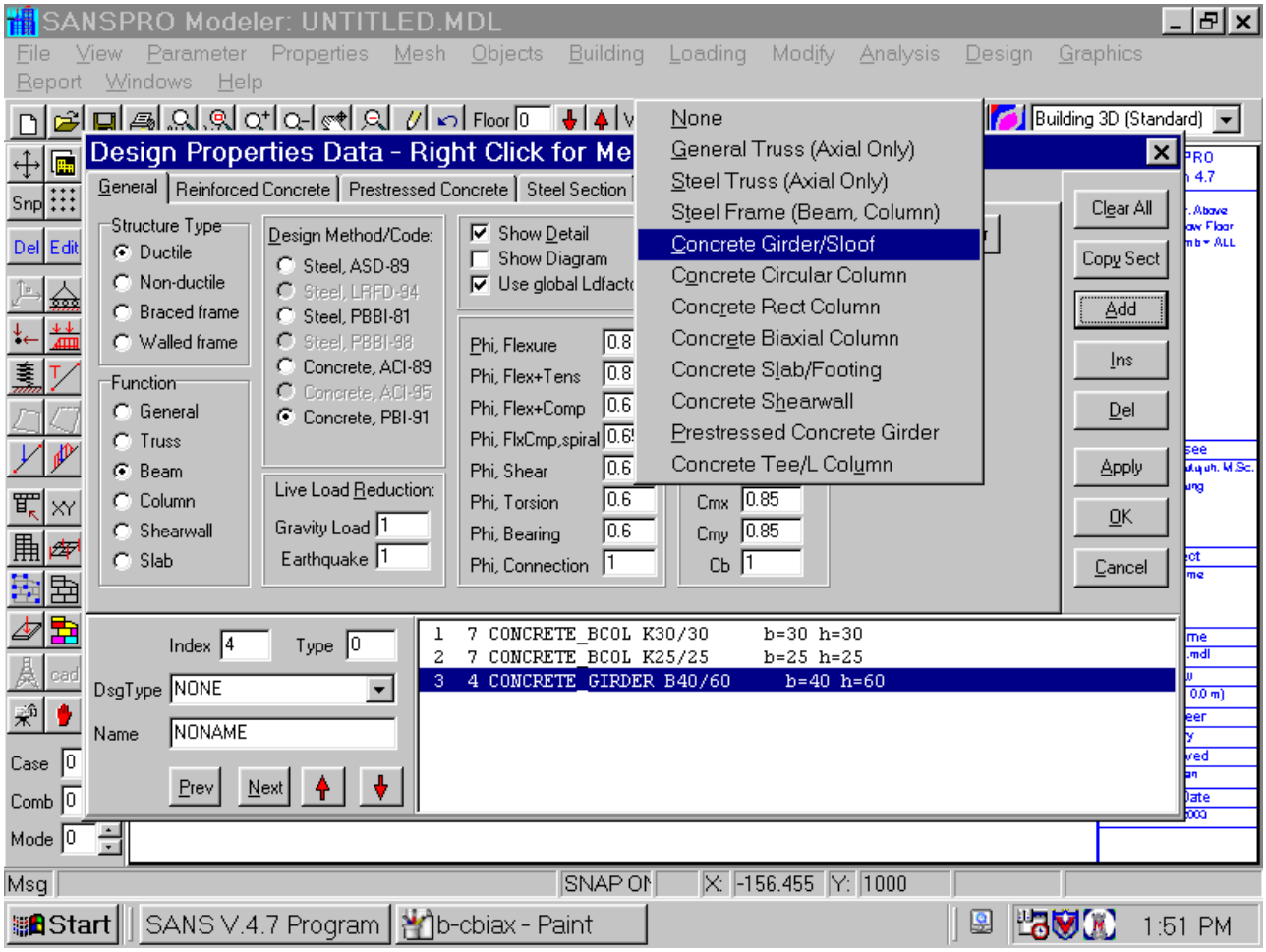


- Column Design Type = Concrete Biaxial Column
    - Columns will be designed using Biaxial Bending Moment and Normal forces
    - User can select rebar placement: two faces or four faces
    - If two faces rebar selected, user can enter total number of side bar
    - If four faces rebar selected, main rebars will be distributed equally for all four column sides

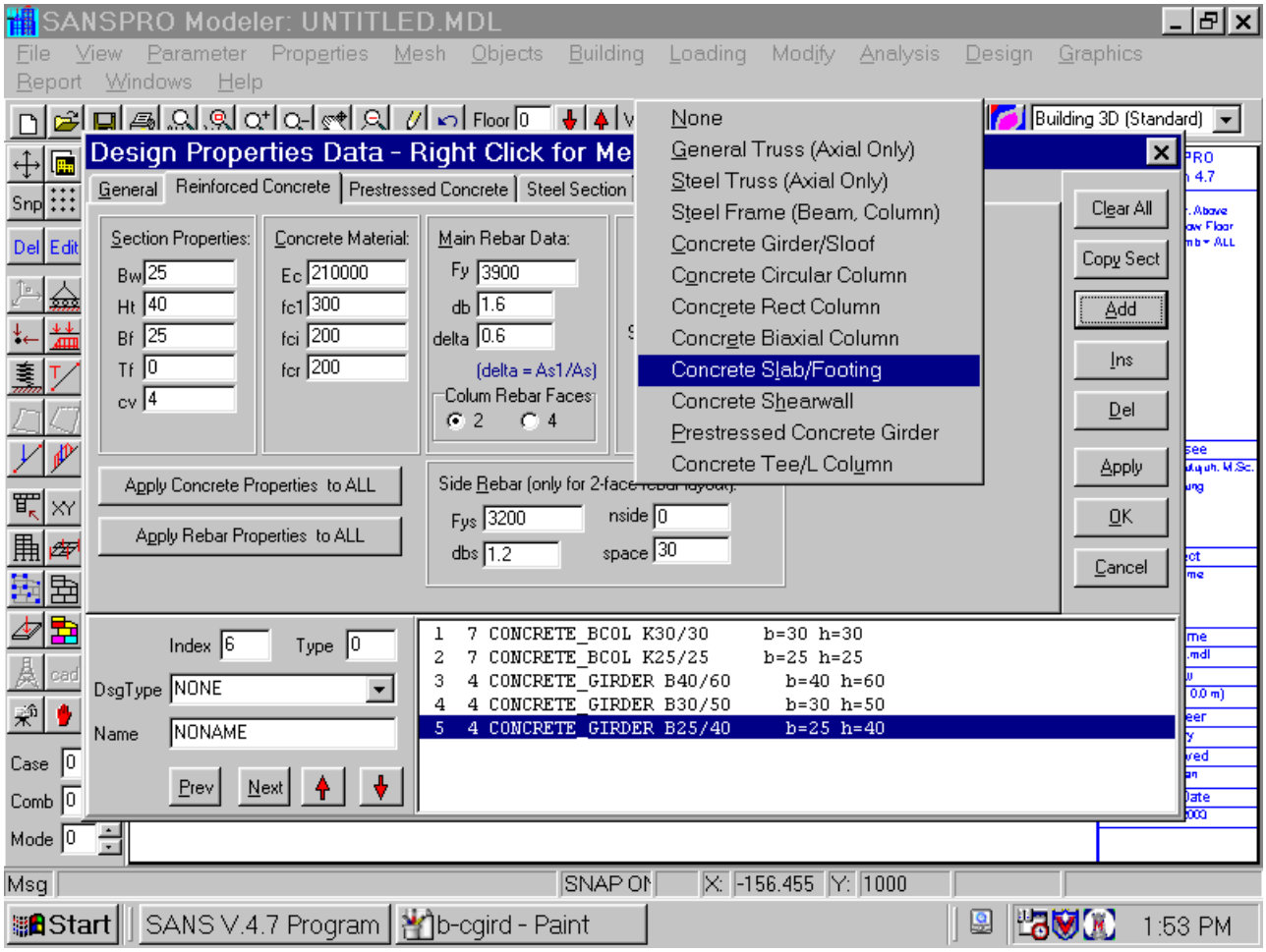
- If Design Type = Rect Column selected:
    - Column will be designed using resultant of Mx,Mz and Normal
    - Resultant Moment will be applied to H direction of column.
    - H direction of column should be entered as column size parallel to dominant moment direction
    - User can select rebar placement: two faces or four faces
    - If two faces rebar selected, user can enter total number of side bar
    - If four faces rebar selected, main rebars will be distributed equally for all four column sides



- Beam Design Type = Concrete Girder
- For beam sections, parameter bf (flange width) and tf (flange thickness) must be entered.
- If not a flanged beam, bf should be equal to b and tf = 0
- Enter delta factor as ratio of secondary to primary rebar area = 0.6
- For deep beam, one can enter total number of side rebar

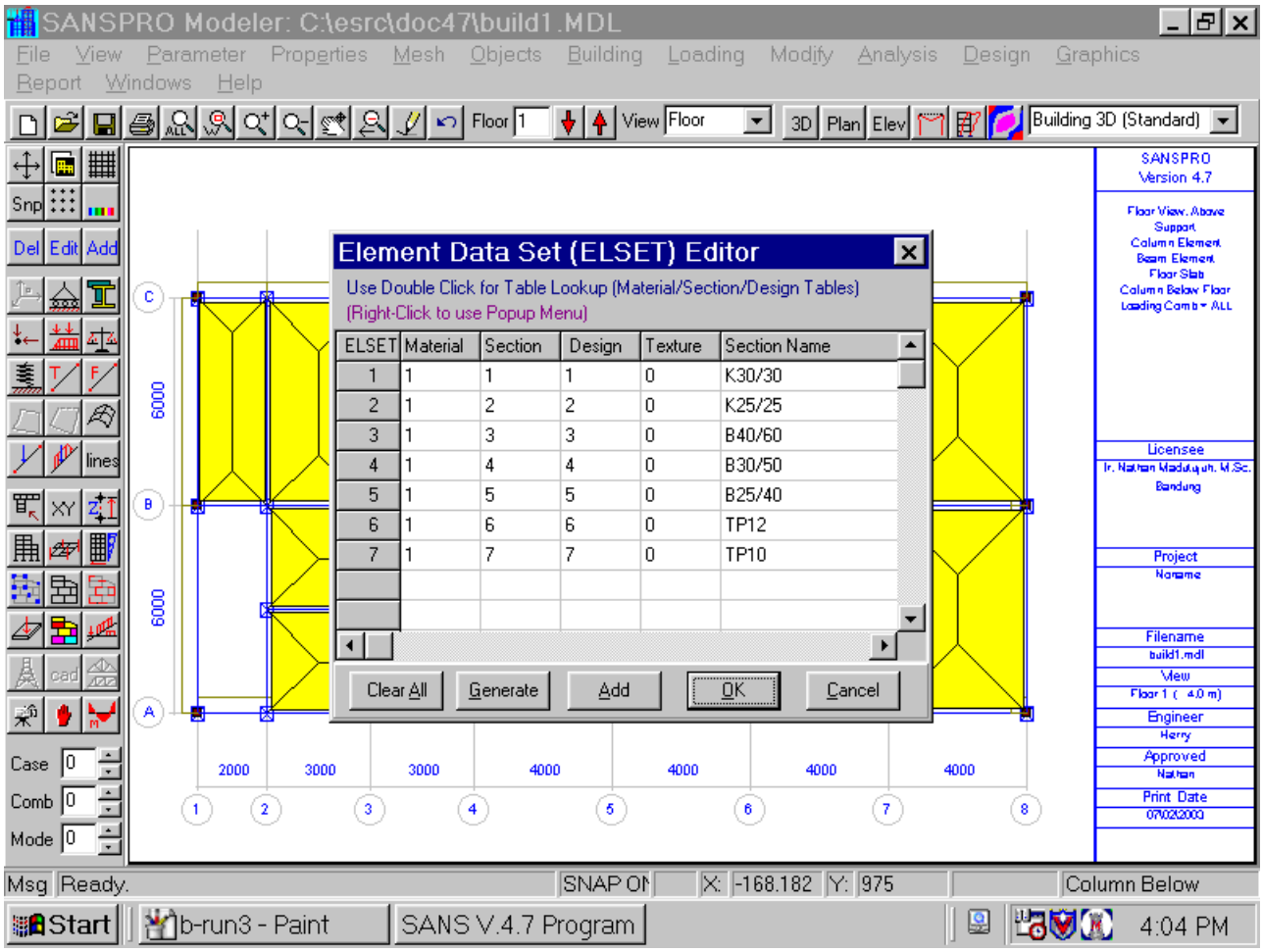


- Slab Design Type = Concrete Slab
- Slab will be design as concrete girder with equal top and bottom rebar
- Slab will be designed based on floor regions, so you do not need to analyze the model before viewing slab design.



6.4. Define ELSET Data

- Generate elset data for this building



- Press **F2** to save the model

7. Define Floor Slab Data

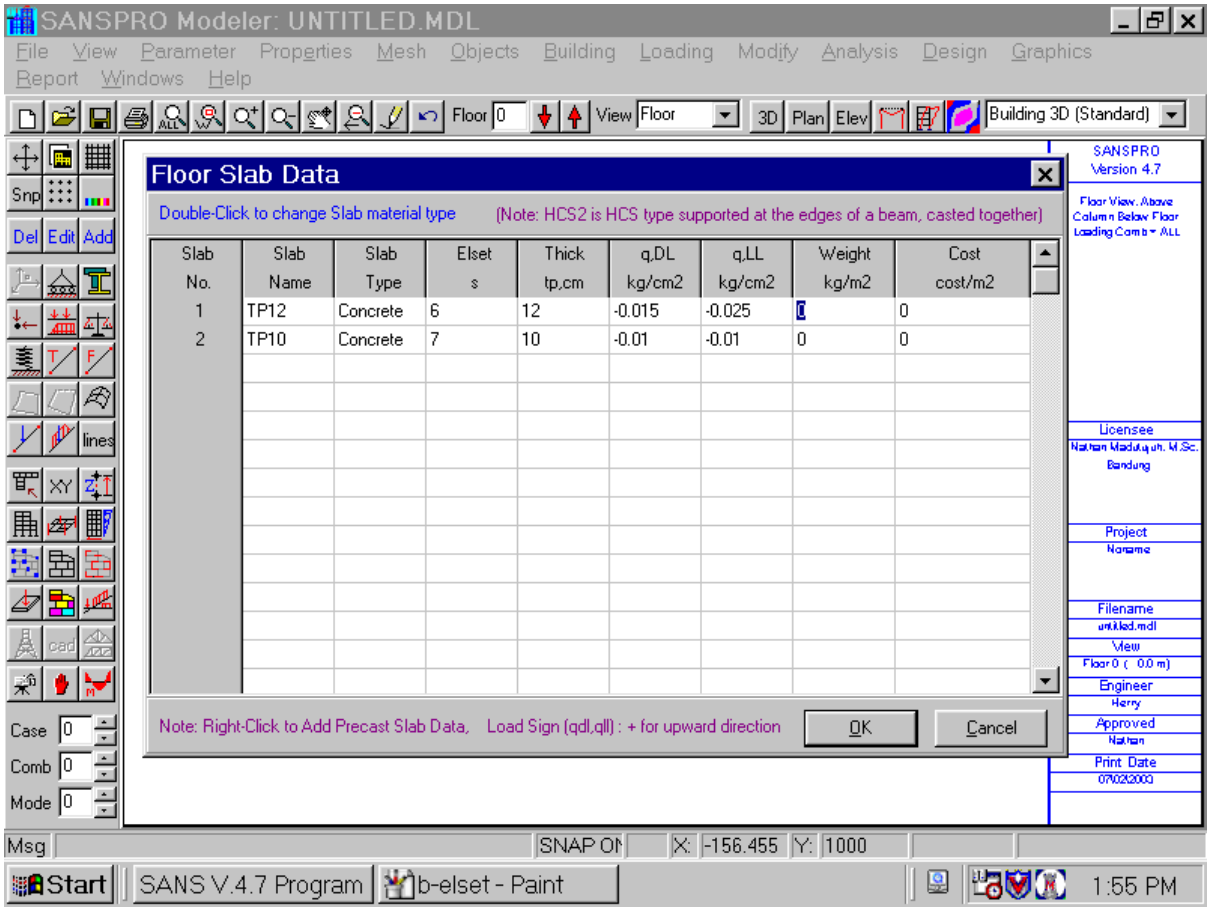
- Double-Click table column named Elset, select slab elset index
- Double-Click table column name Slab Type, select Concrete

Typical Slab:
- Enter q,DL as –0.015 kg/cm2 (equal to 150 kg/m2) for superimposed DL excluded self-weight of concrete slab that will be calculated automatically by SANSPRO
- Enter q,LL as –0.025 kg/cm2 (equal to 250 kg/m2) for superimposed LL

Roof top slab:
- Enter q,DL as –0.01 kg/cm2 (equal to 100 kg/m2) for superimposed DL excluded self-weight of concrete slab that will be calculated automatically by SANSPRO
- Enter q,LL as –0.01 kg/cm2 (equal to 100 kg/m2) for superimposed LL

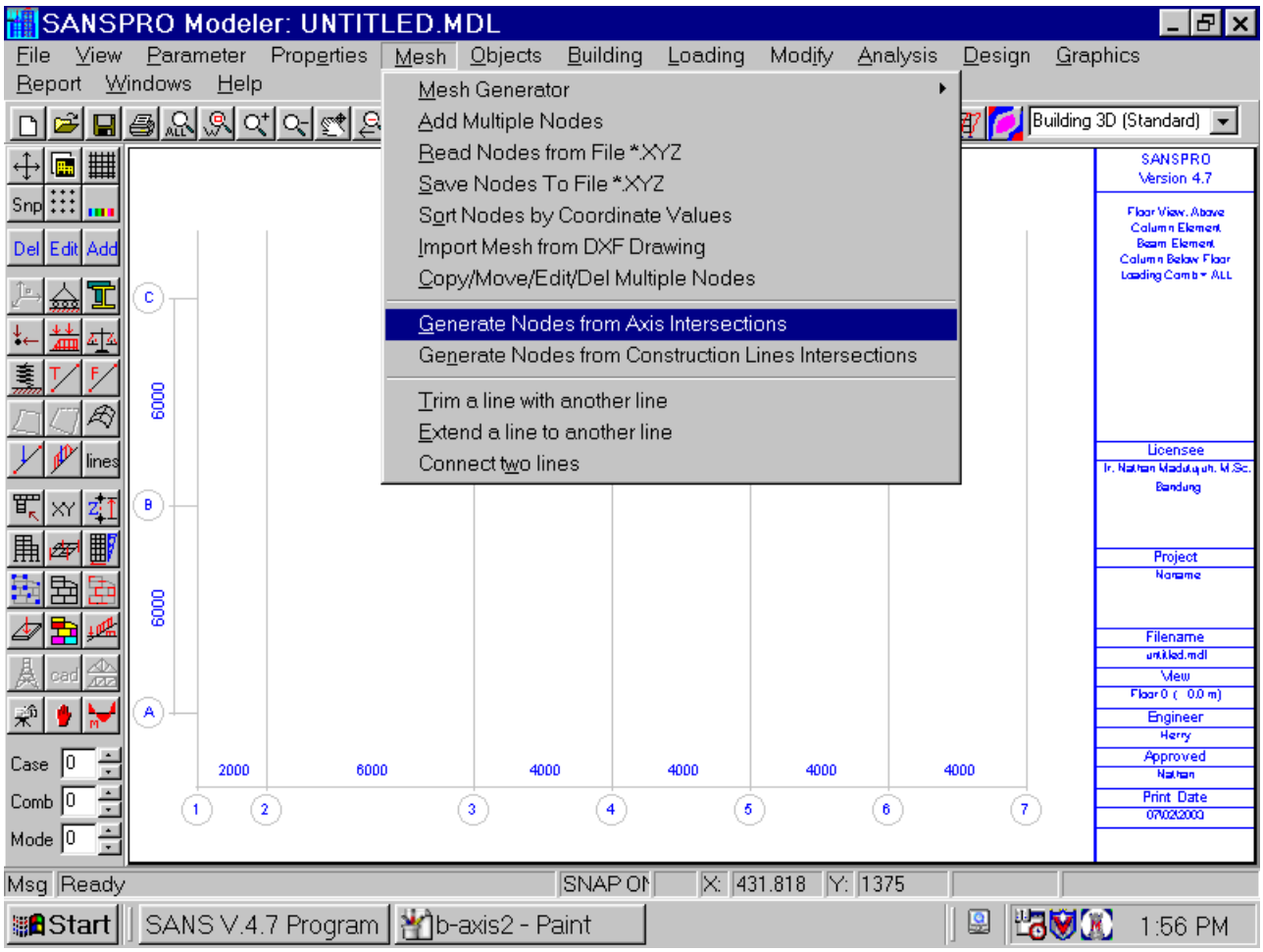
Other parameters:
- If Weight parameter filled, SANSPRO will use this value as weight of slab instead calculating it automatically using Unit weight x Thickness. This is useful for Hollow core precast slab or Cold-formed steel deck.

- If Cost parameter filled, SANSPRO will use this as unit price for calculating slab cost. This cost may include concrete, rebar, and additional rebar cost, placement cost, etc. If you entered formwork cost, exclude the cost from this slab unit cost.

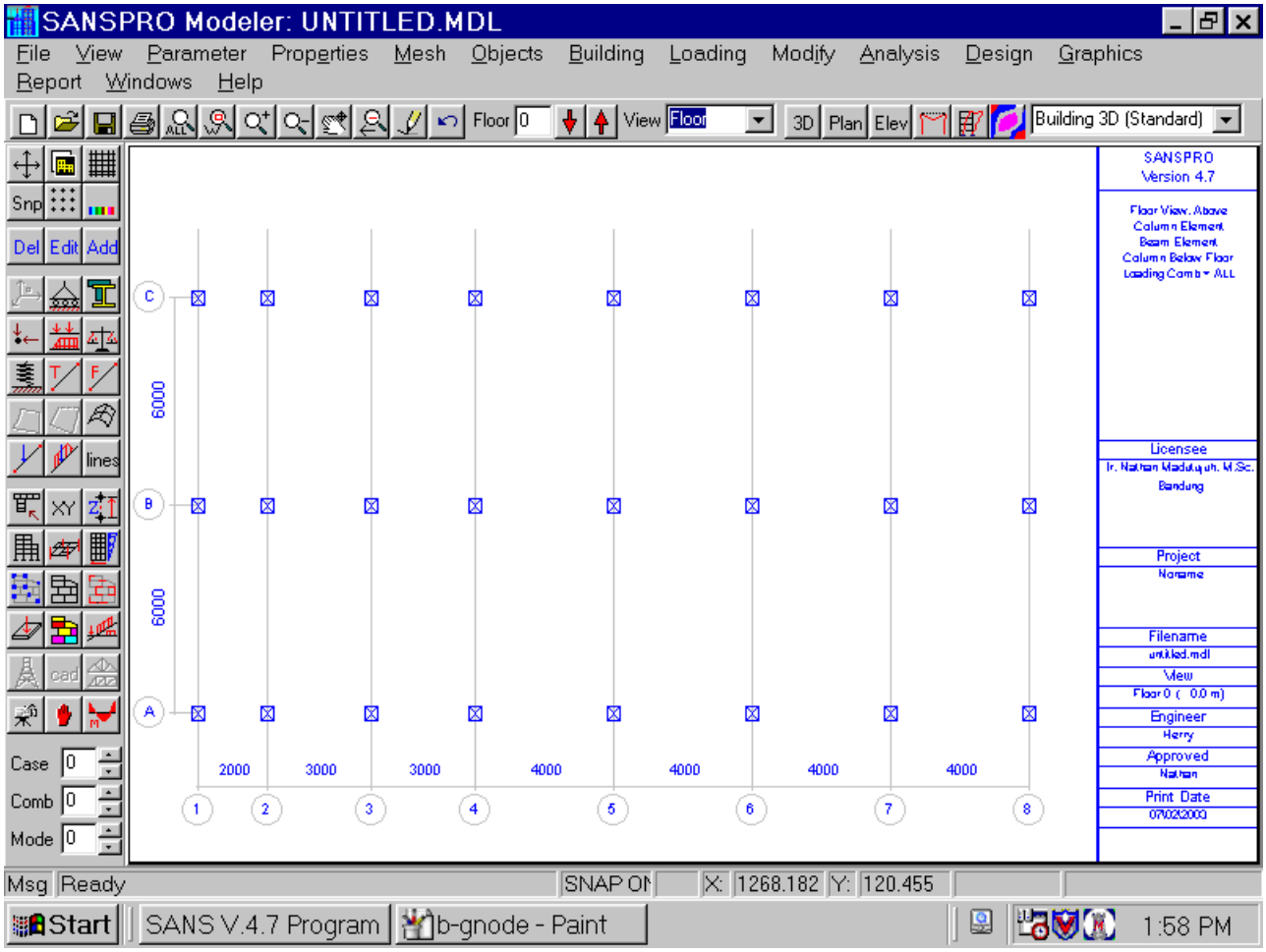


8. Adding Nodes

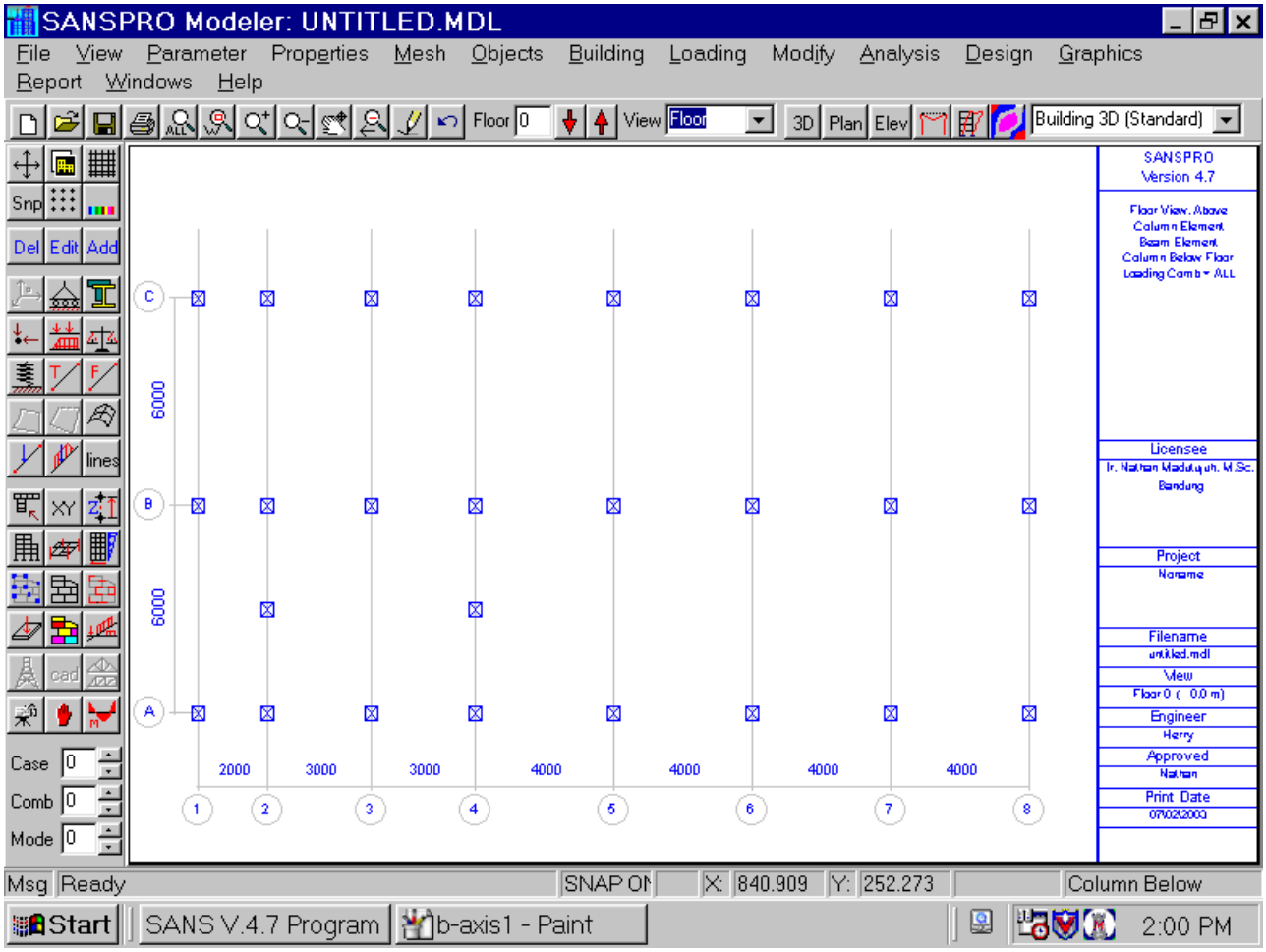
- Nodes will be added automatically by generating them from all axis intersections
- Additional nodes can be added by Clicking Icon XY and Icon Add then click any location



- The generated nodes will be as follows:



- Additional nodes can be added by Clicking Icon **XY** and Icon **Add** then click any location
- Here we will add two nodes at axis 2 and 4 for staircase void of the building
- After we add additional nodes, layout will be:



- Press **F2** to save the model

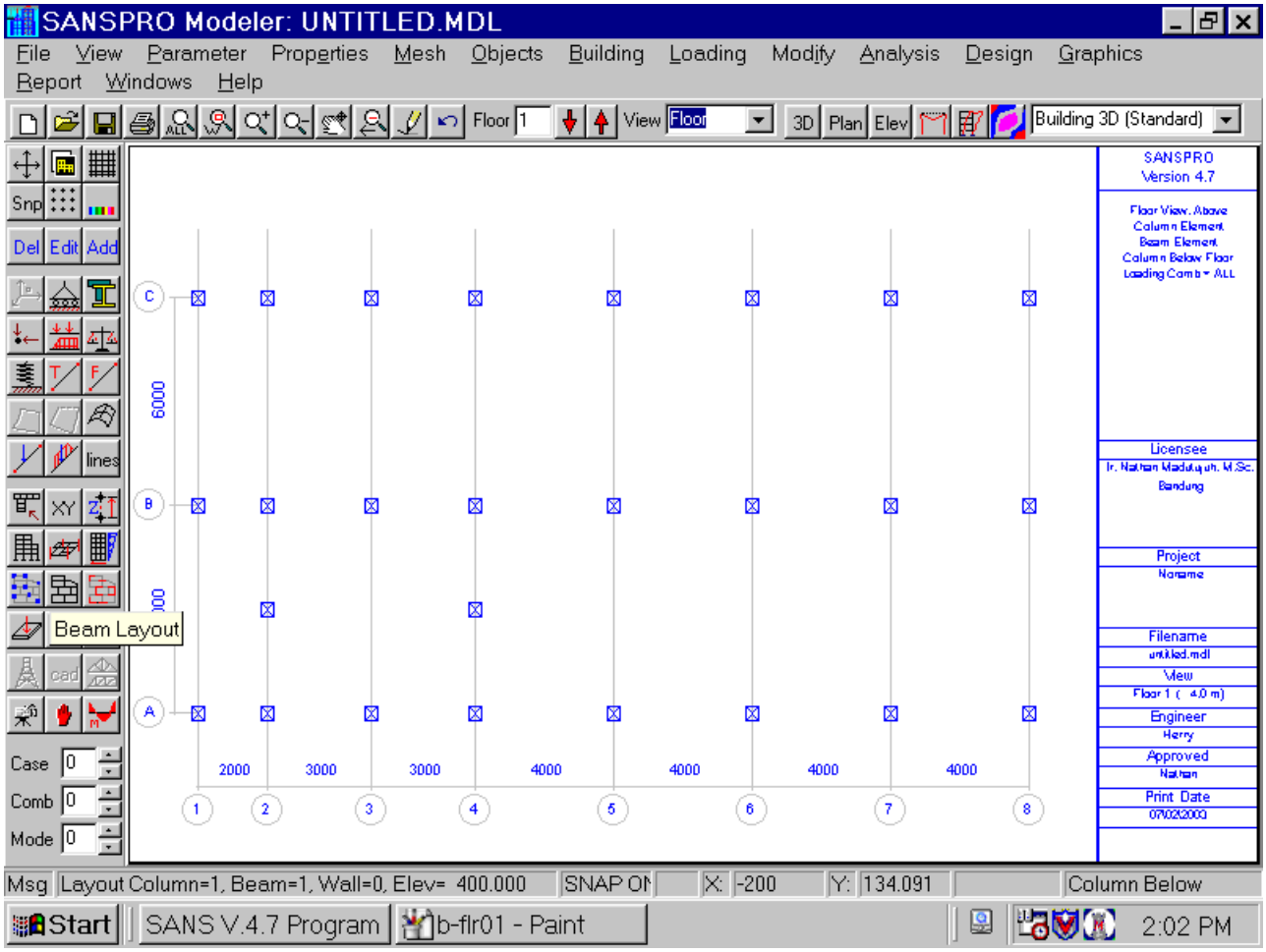
9. Adding Beam Elements

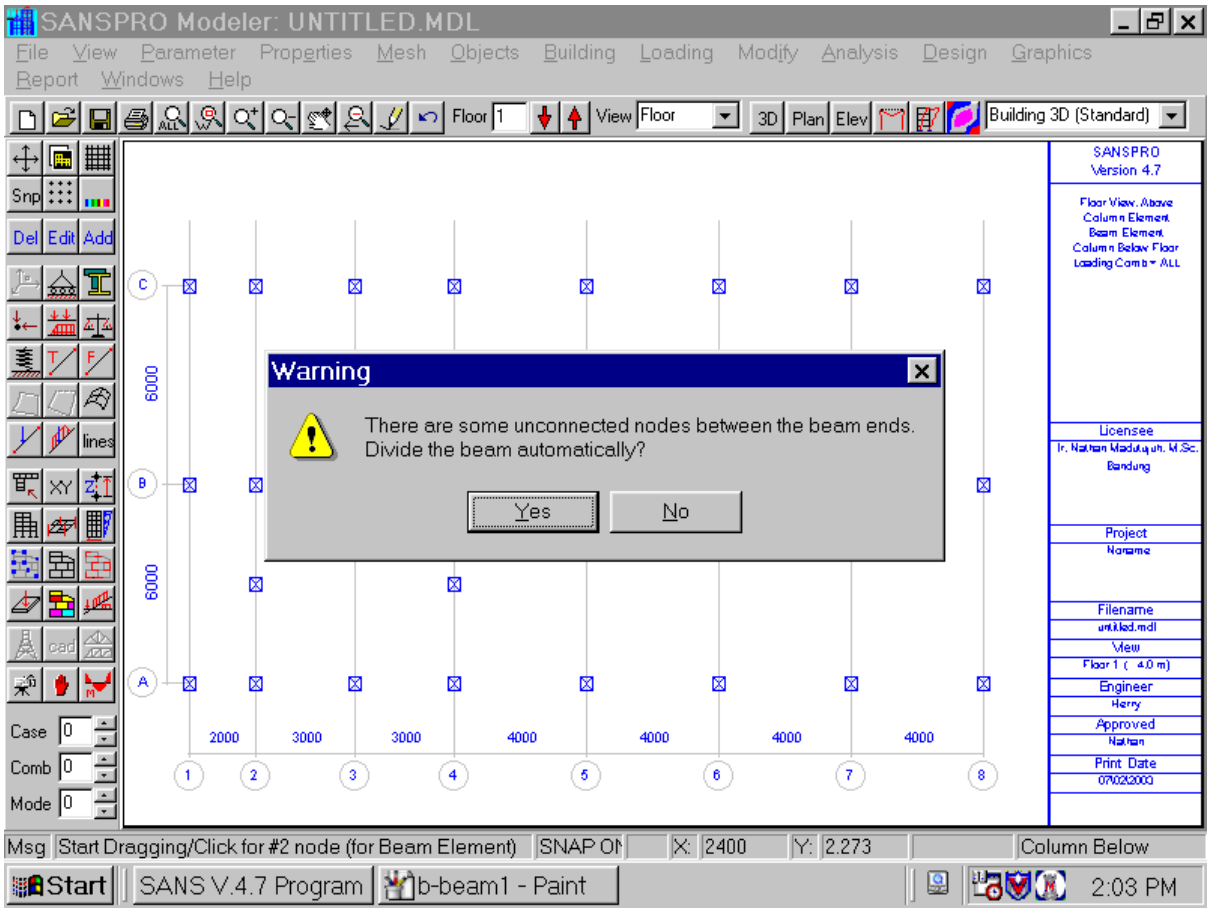
- Move to Floor No. 1
- Click Icon Beam Layout, Icon Add
- Click 2 nodes or Click a node and drag to another node and release the mouse button
- Select Beam Elset
- Select Beam Function:

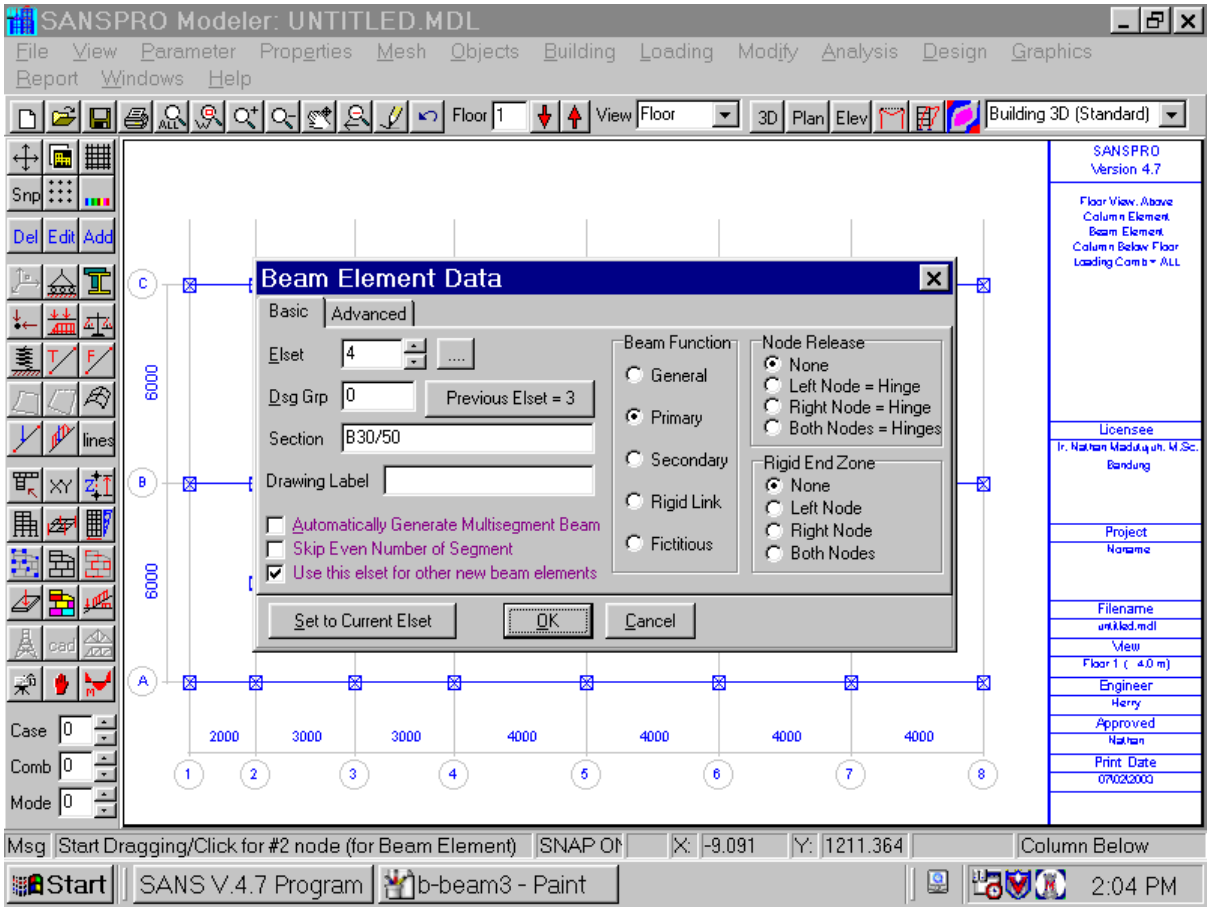
Primary	-> Beam will be included in capacity design
Secondary	-> Beam will be excluded in capacity design
Rigid Link	-> Beam will be weightless, exluded in any design
Dummy/Fictitious	-> Beam will be weightless, exluded in any design

- Rigid Link is used for load transfer beams
- Dummy or Fictitious is used for slab region dividing beams
- Add Rigid End and node release options.
- If a beam is crossing any midpoints, user will be asked if he wants to divide the beam automatically. Answer with Yes if so.
- You may auto divide/correct beam mesh later easily using SANSPRO automatic mesh correction.

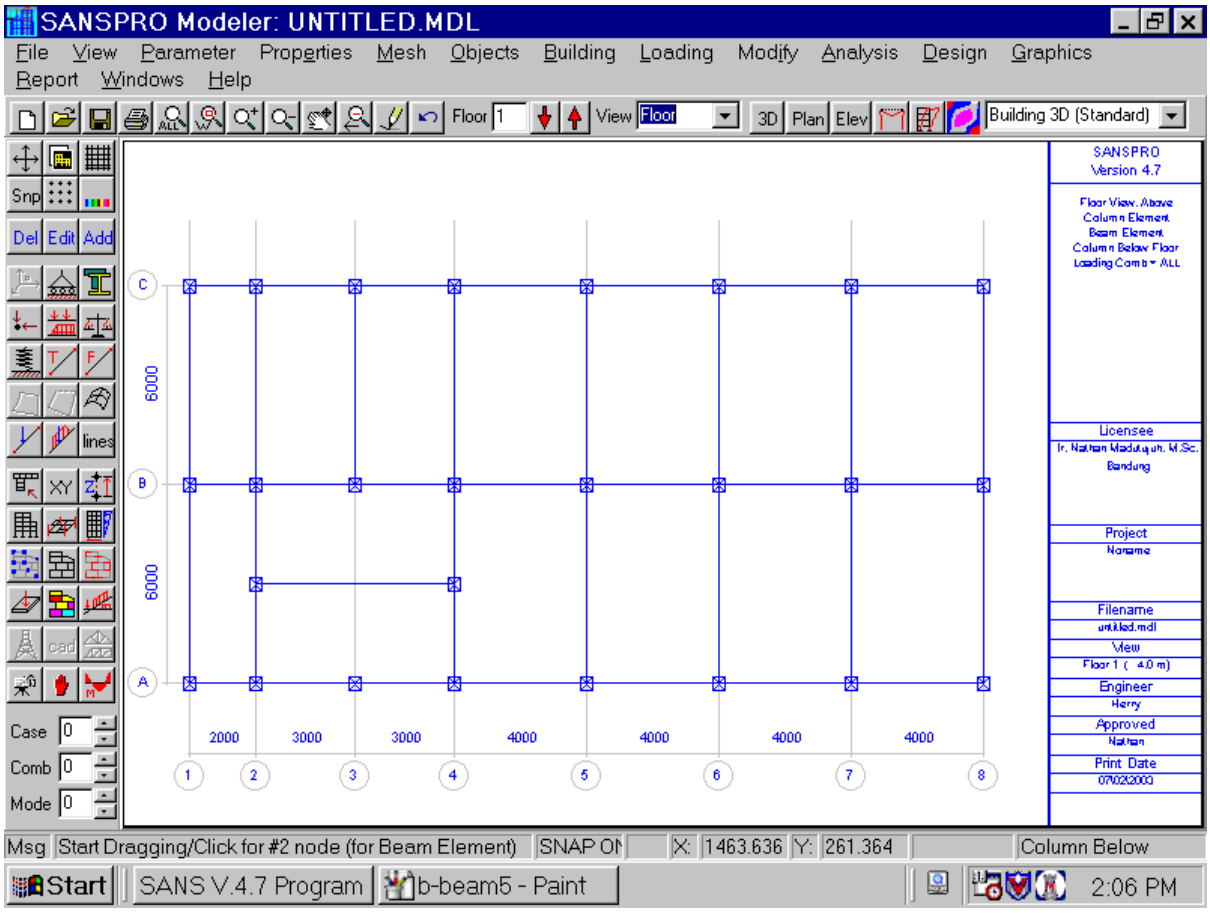
- Repeat for other floors with different beam layout
- You need to enter only beams at different layout, other floor with same beam layout will follow automatically.




SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Warning
There are some unconnected nodes between the beam ends.
Divide the beam automatically?
Yes
No
SANSPRO Version 4.7
Msg Start Dragging/Click for #2 node (for Beam Element)
Start
SANS V.4.7 Program
b-beam1 - Paint
2:03 PM


SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Beam Element Data
Basic
Advanced
Elset
Dsg Grp
Previous Elset = 3
Section
B30/50
Drawing Label
Automatically Generate Multisegment Beam
Skip Even Number of Segment
Use this elset for other new beam elements
Beam Function
General
Primary
Secondary
Rigid Link
Fictitious
Node Release
None
Left Node = Hinge
Right Node = Hinge
Both Nodes = Hinges
Rigid End Zone
None
Left Node
Right Node
Both Nodes
Set to Current Elset
OK
Cancel
SANSPRO Version 4.7
Msg Start Dragging/Click for #2 node (for Beam Element)
Start
SANS V.4.7 Program
b-beam3 - Paint
2:04 PM

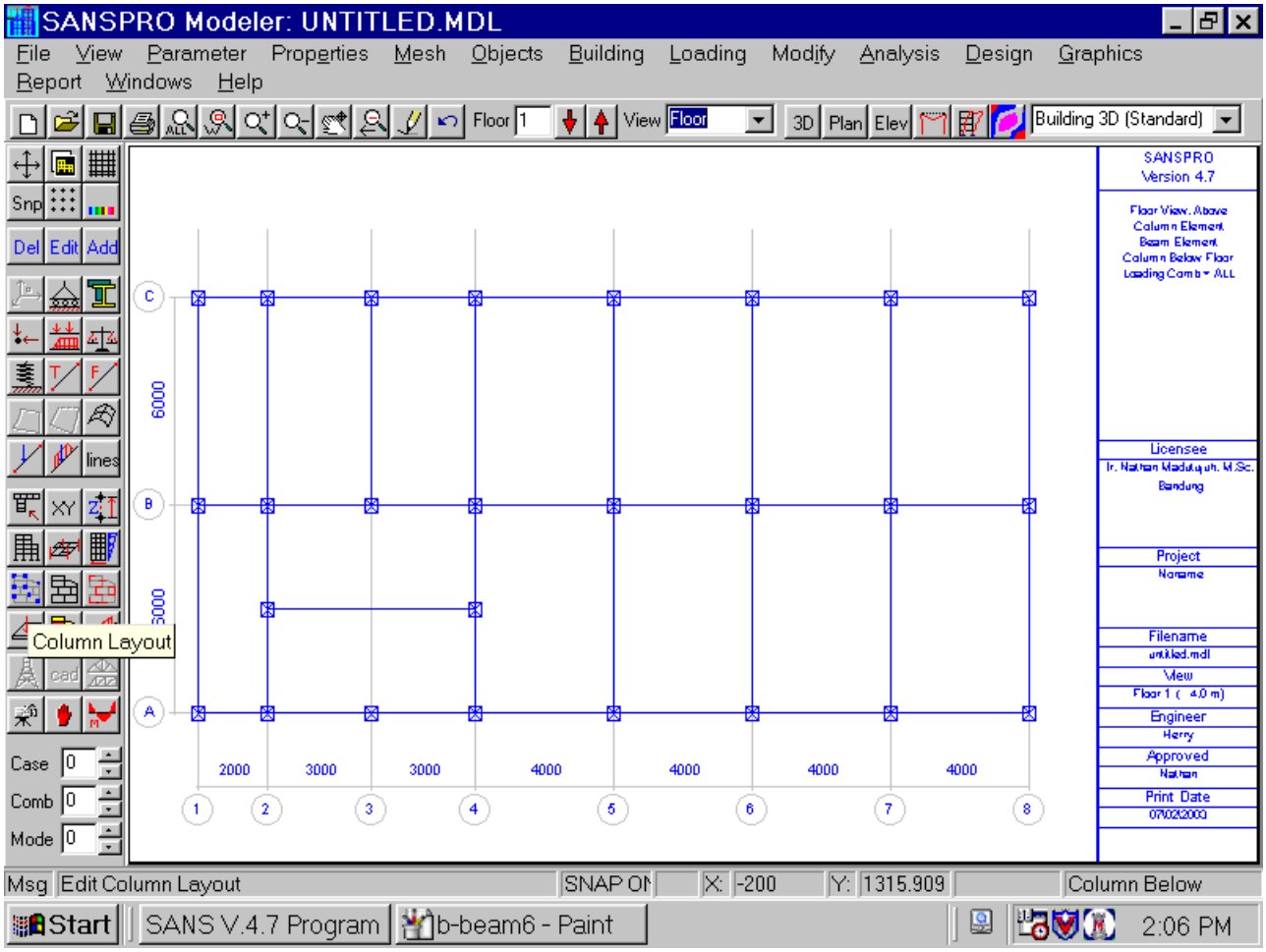
After adding all needed beams, beam layout for the Floor No.1 will look as follows:

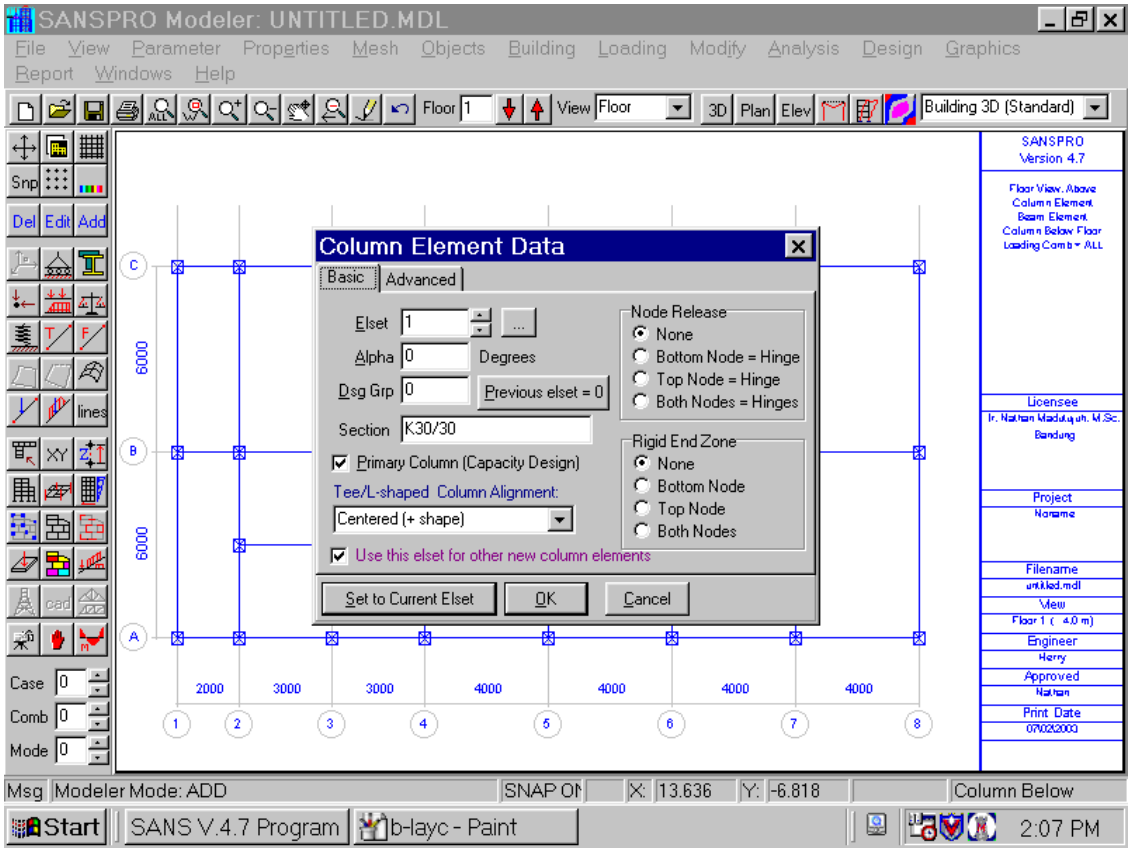


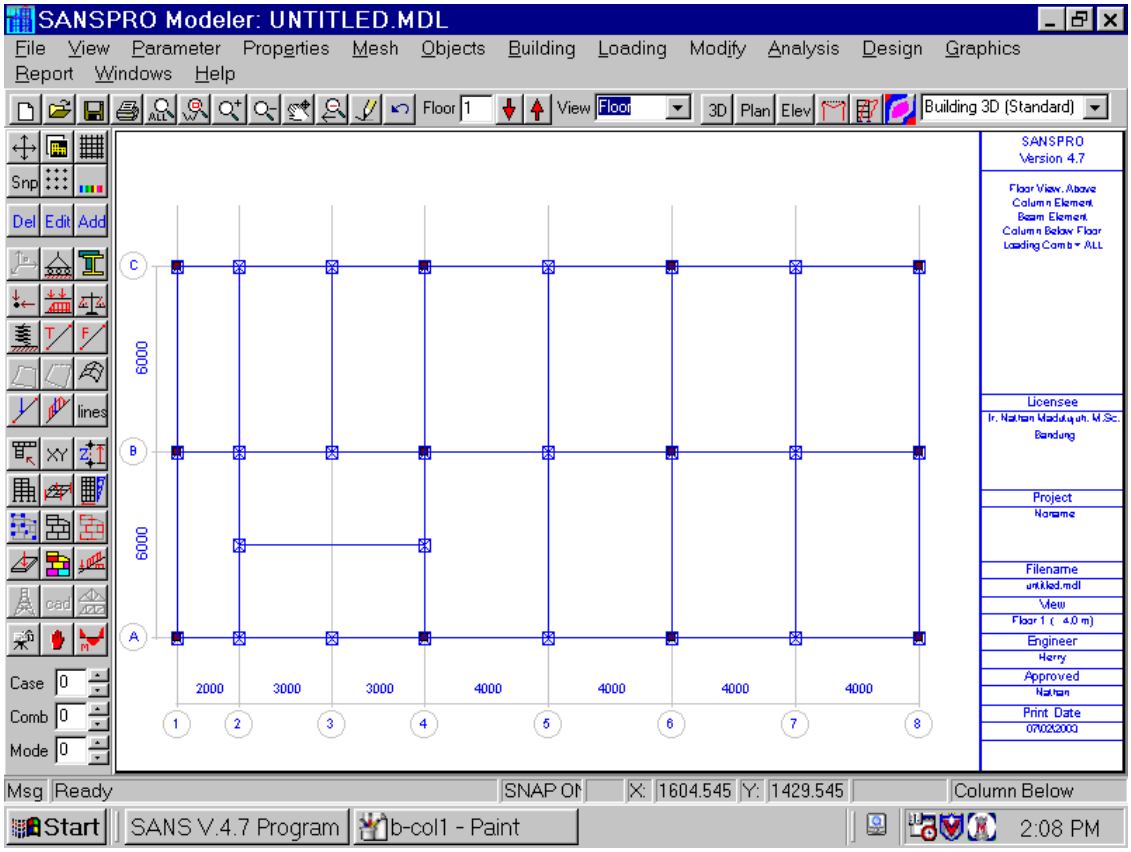
10. Adding Column Elements

- Move to Floor No. 1
- What you see is columns below Floor No. 1 (see status bar, at right)
- To see columns above Floor No. 1, Change the View Option
- Click Icon Column Layout, Icon Add
- Click 1 node
- Select Column Elset
- Select Column Function:
  - Primary -> Column will be included in capacity design
  - Secondary -> Column will be excluded in capacity design
- Add Rigid End and node release options.
- If a column is crossing any beam, the beam must be divided/splitted.
- You may auto divide/correct beam mesh later easily using SANSPRO automatic mesh correction.
- For Tee or L shaped columns, one can entered the alignment and direction of the flanges

- Repeat for other floors with different column layout
- You need to enter only columns at different layout, other floor with same beam layout will follow automatically.







- Press **F2** to save the model

11. Adding Shearwall Elements

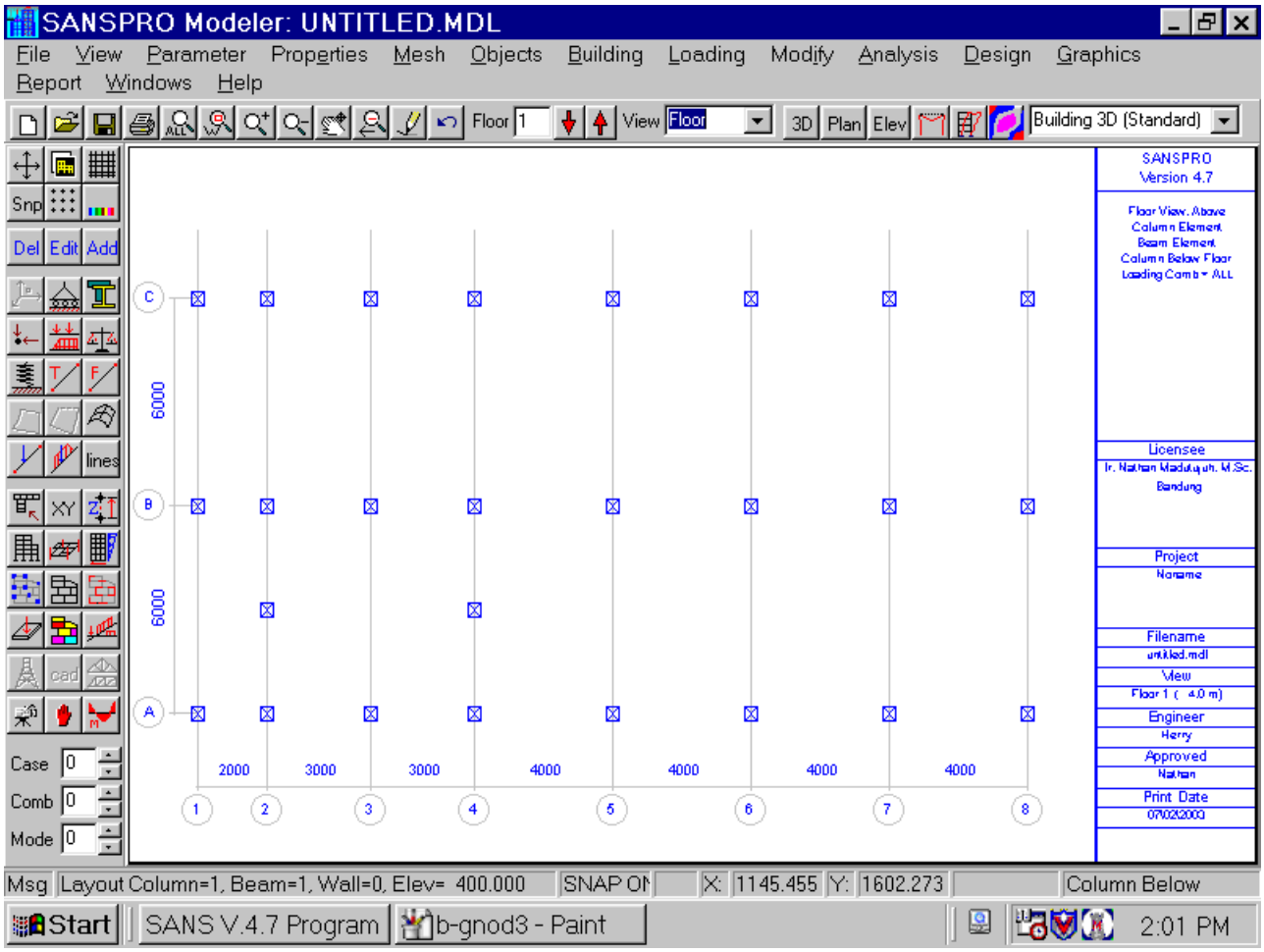
- In this example there is no shearwall but user can add shearwall as follows:
    - Goto Floor with Shearwall layout
    - Click Icon Shearwall
    - Click Icon Add
    - Click Two nodes
    - Select Shearwall properties
        - Shell option is used for wall resisting in-plane and out-of-plane forces
        - Plane stress/strain option is used for wall resisting in-plane forces
        - Plate bending option is used for wall resisting out-of-plane forces
        - Shear panel is for wall resisting in-plane shear only (eq. brick walls)
- The walls you see is walls under the current floor
- To see walls above Floor No. 1, Change the View Option

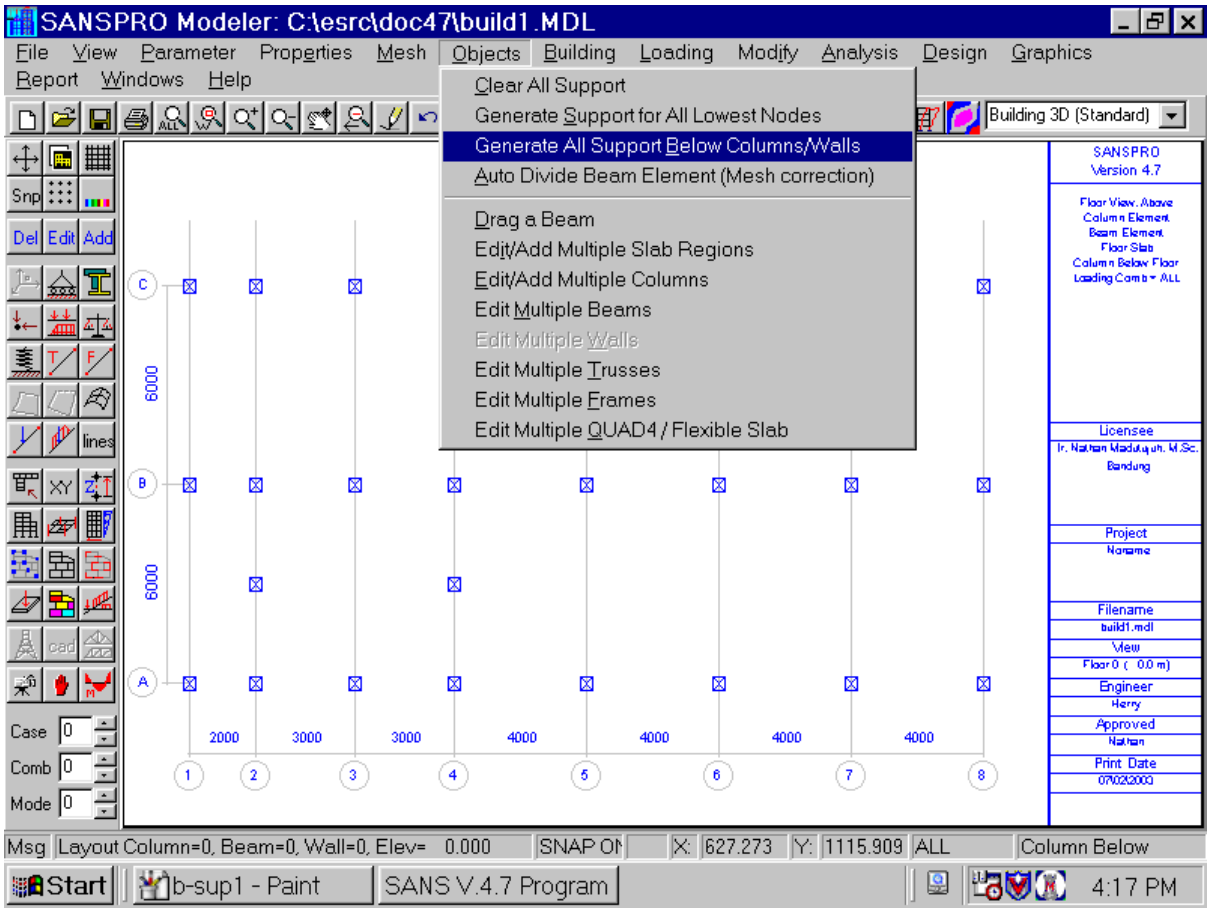
- Press **F2** to save the model

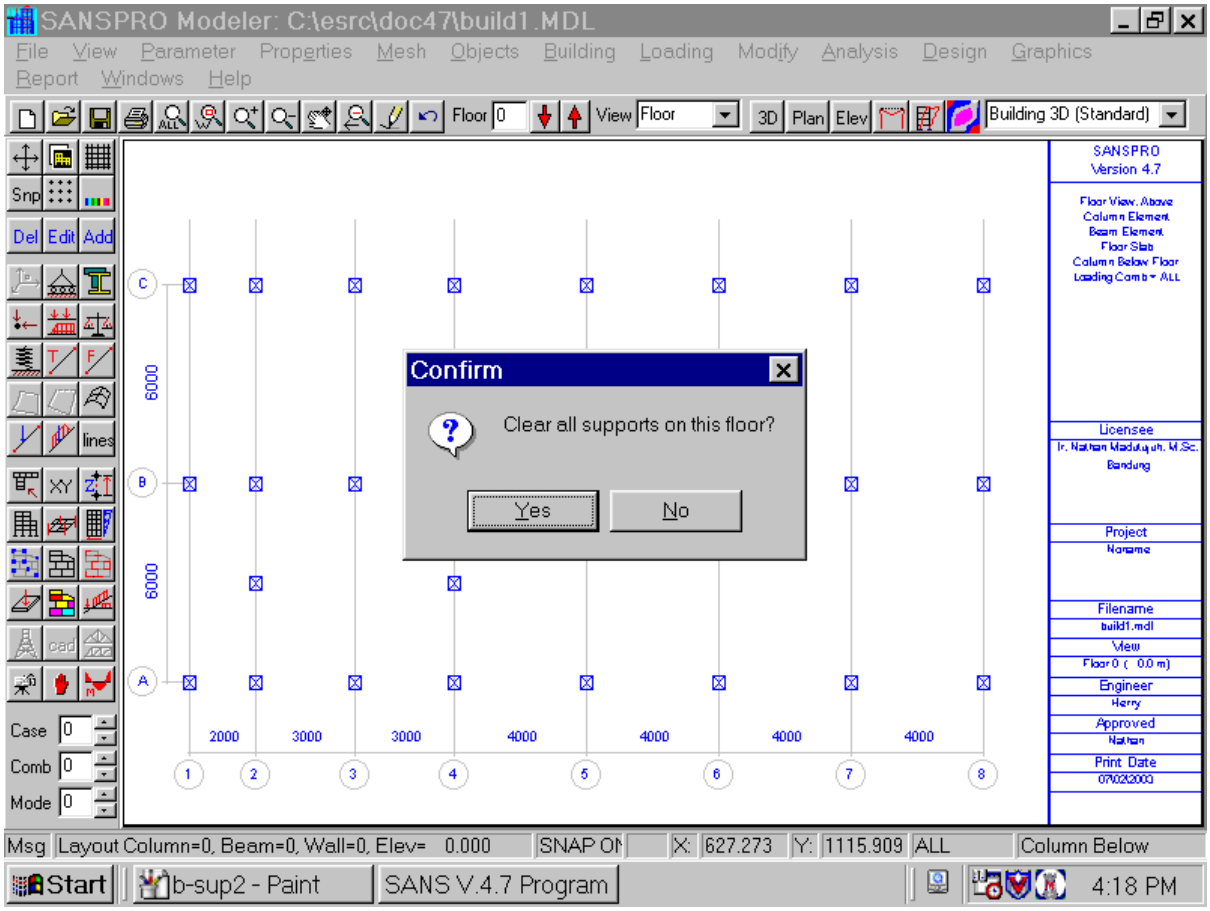
12. Adding Support

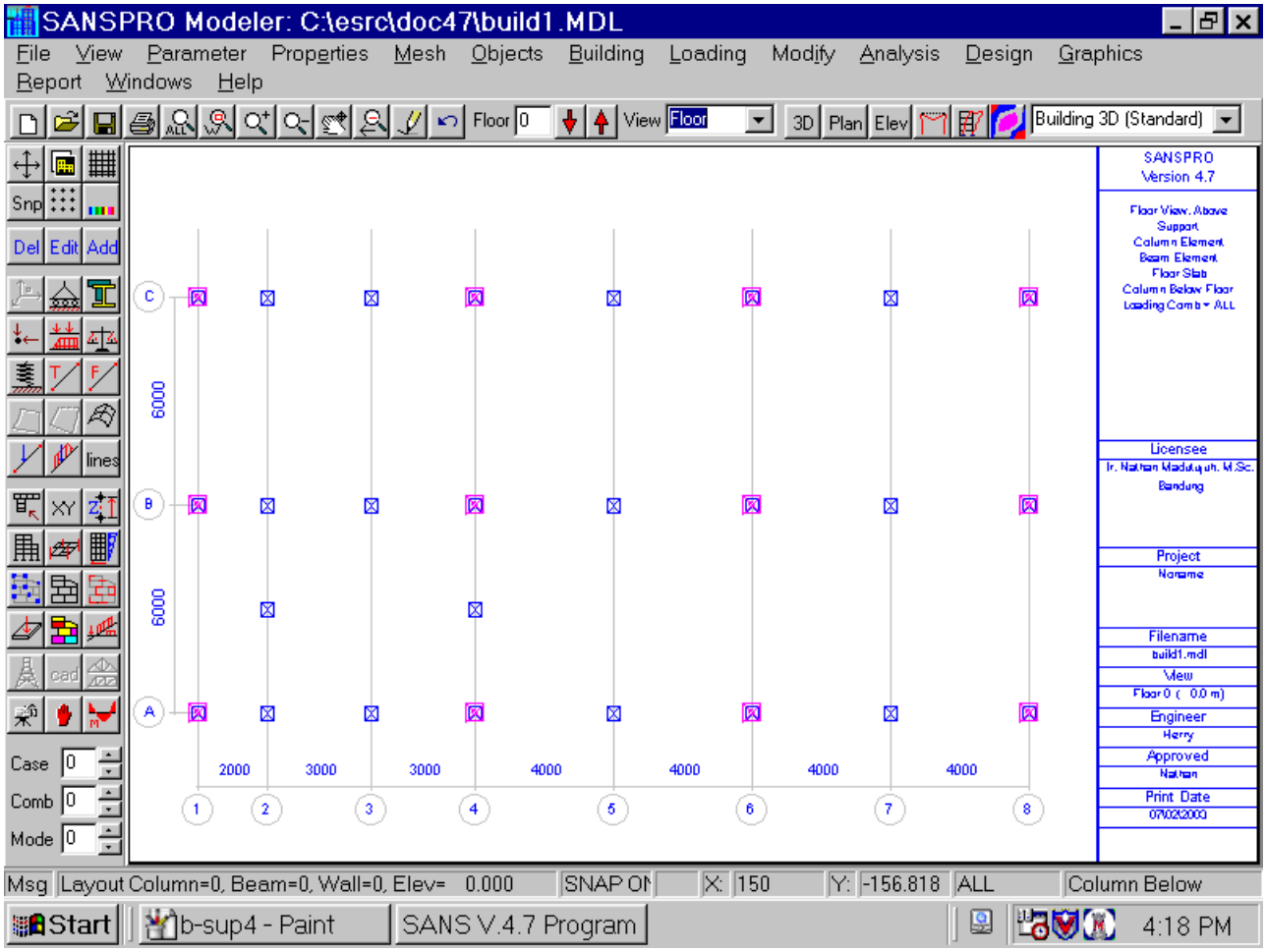
- Generate Nodal Supports manually as follows:
    - Move to Floor No. 0
    - Click Icon Support and Icon Add
    - Click any target node
    - Select Support Type
    - Click OK
    - Click Other nodes
    - Click Icon Redraw if finished

- Alternatively (recommended), one can generate all needed supports automatically using Support Object Generator as follows:
    - Move to Floor No. 0




SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Clear All Support
Generate Support for All Lowest Nodes
Generate All Support Below Columns/Walls
Auto Divide Beam Element (Mesh correction)
Drag a Beam
Edit/Add Multiple Slab Regions
Edit/Add Multiple Columns
Edit Multiple Beams
Edit Multiple Walls
Edit Multiple Trusses
Edit Multiple Frames
Edit Multiple QUAD4 / Flexible Slab
Building 3D (Standard)
SANSPRO Version 4.7
Licensee
Project
Filename
build1.mdl
Engineer
Approved
Print Date
Case
Comb
Mode
Msg Layout Column=0, Beam=0, Wall=0, Elev= 0.000
X: 627.273 Y: 1115.909 ALL Column Below
Start
b-sup1 - Paint
SANS V.4.7 Program
4:17 PM

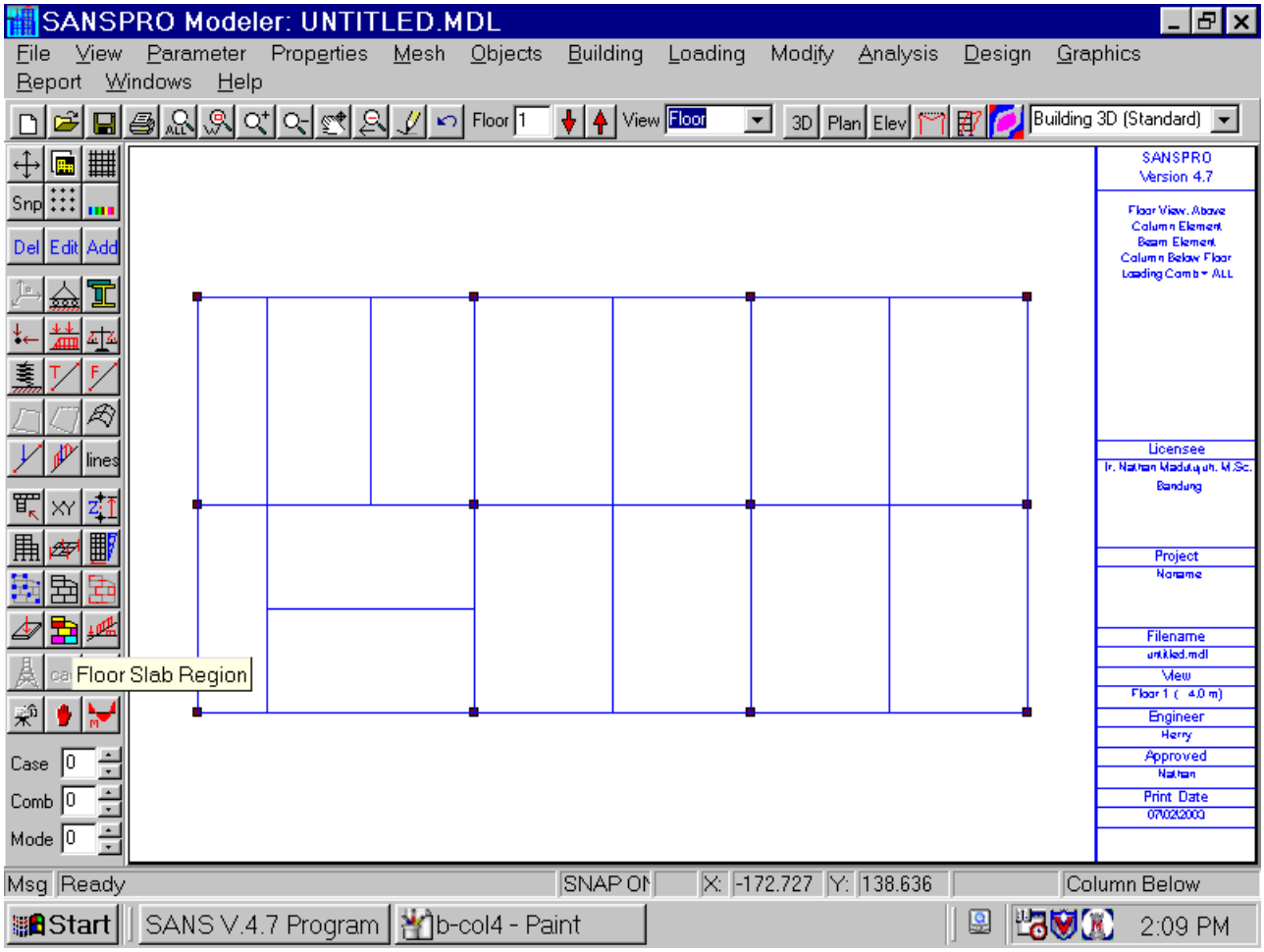

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
Confirm
Clear all supports on this floor?
Yes
No
Building 3D (Standard)
Msg Layout Column=0, Beam=0, Wall=0, Elev= 0.000
X: 627.273 Y: 1115.909 ALL Column Below
Start
b-sup2 - Paint
SANS V.4.7 Program
4:18 PM



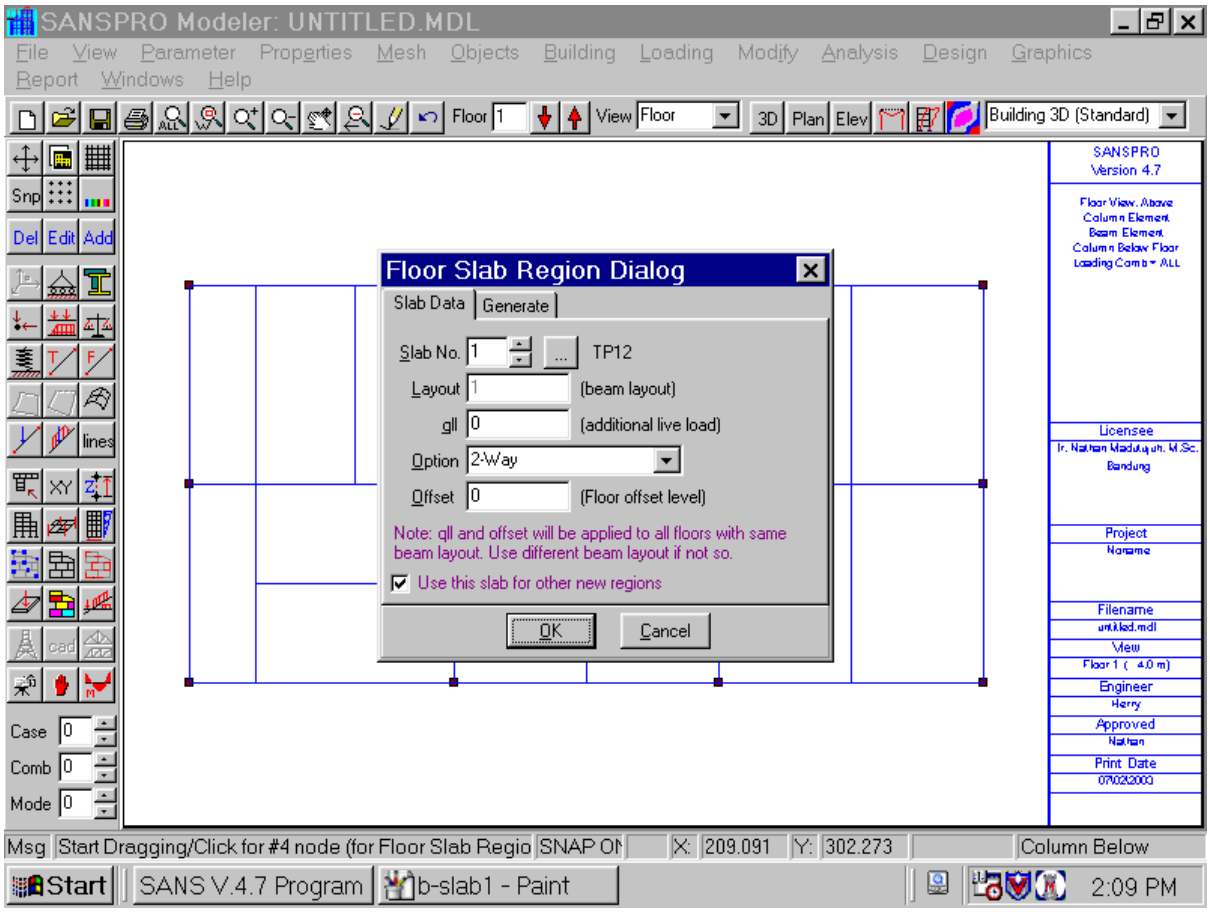
- Press **F2** to save the model

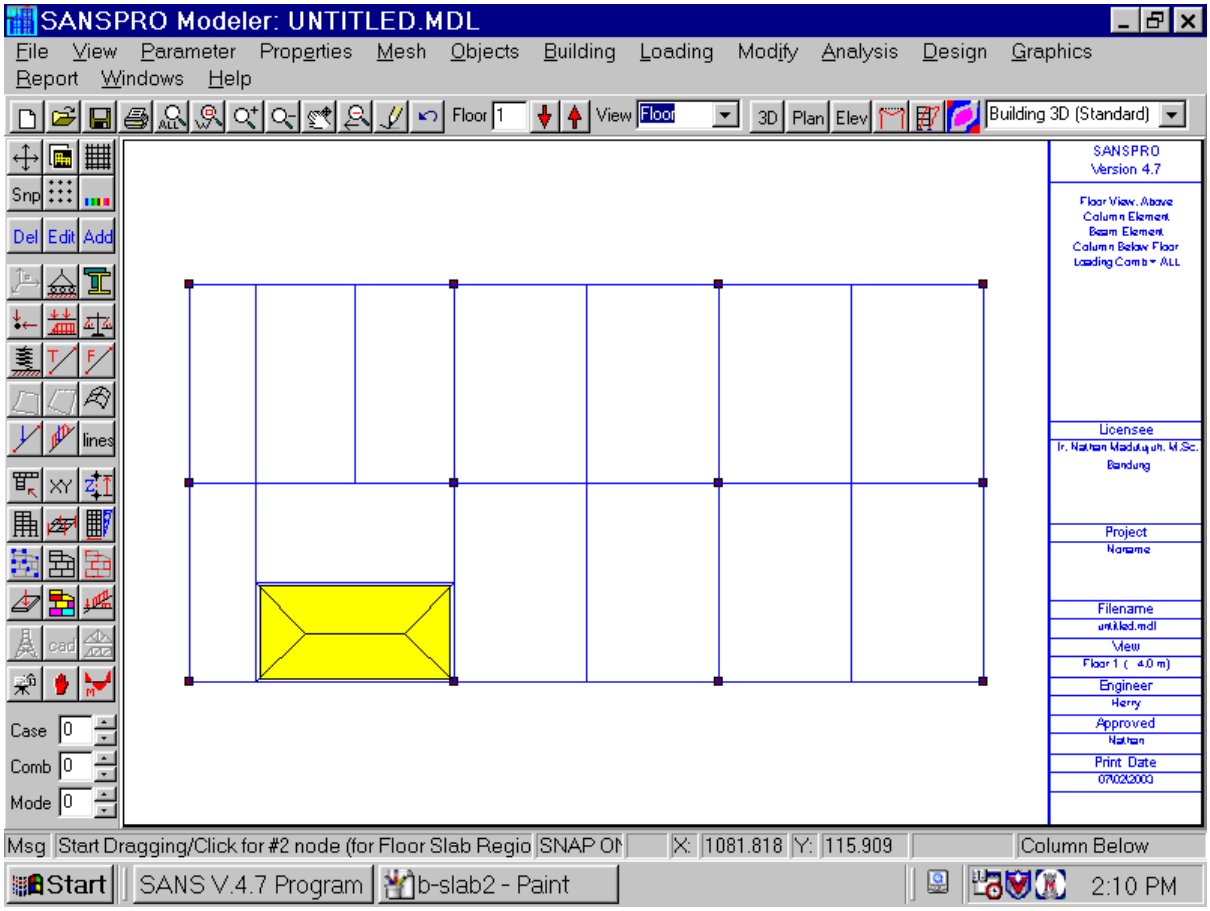
13. Adding Floor Regions

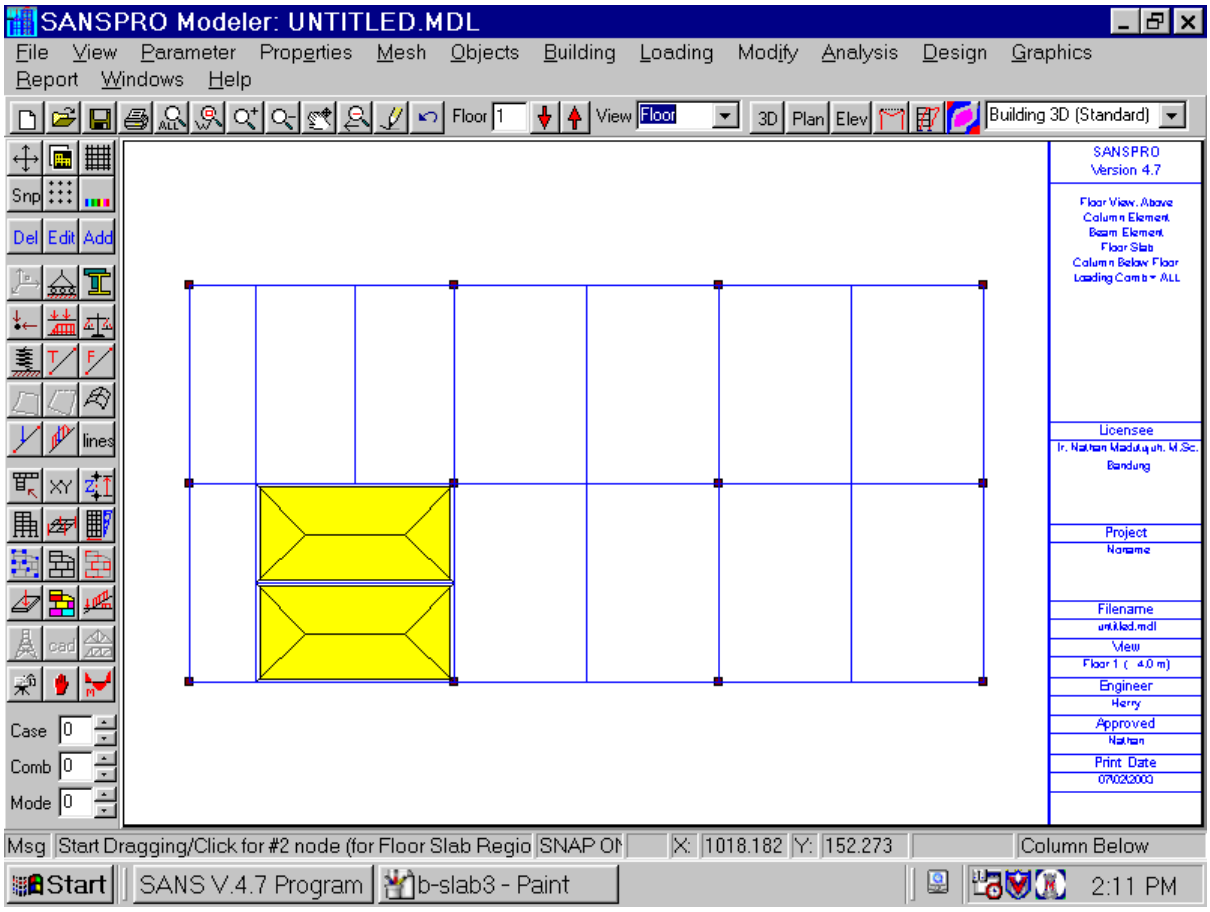
- SANSPRO models floor loading (self/dead load, live load) by dividing floor slab into 3 or 4 straight sides regions.
- Each region side can contain more than one beam.
- Default loading for floor slab is defined at Floor Slab Table previously entered.
- Additional Live Load for a certain region can be added to accommodate for different room functions.
- Click Icon Floor Regions, Icon Add, and Click 4 nodes as 4 corners of the regions.
- Click start from bottom-left, in counter-clockwise direction
- In case of 3 sides regions, Click the third node twice.
- Select Floor Slab Type
- Select Beam Load Pattern (One way or two way slab)
- A Load pattern will appear for the region
- If the load pattern is not nice, delete and add again with different starting node

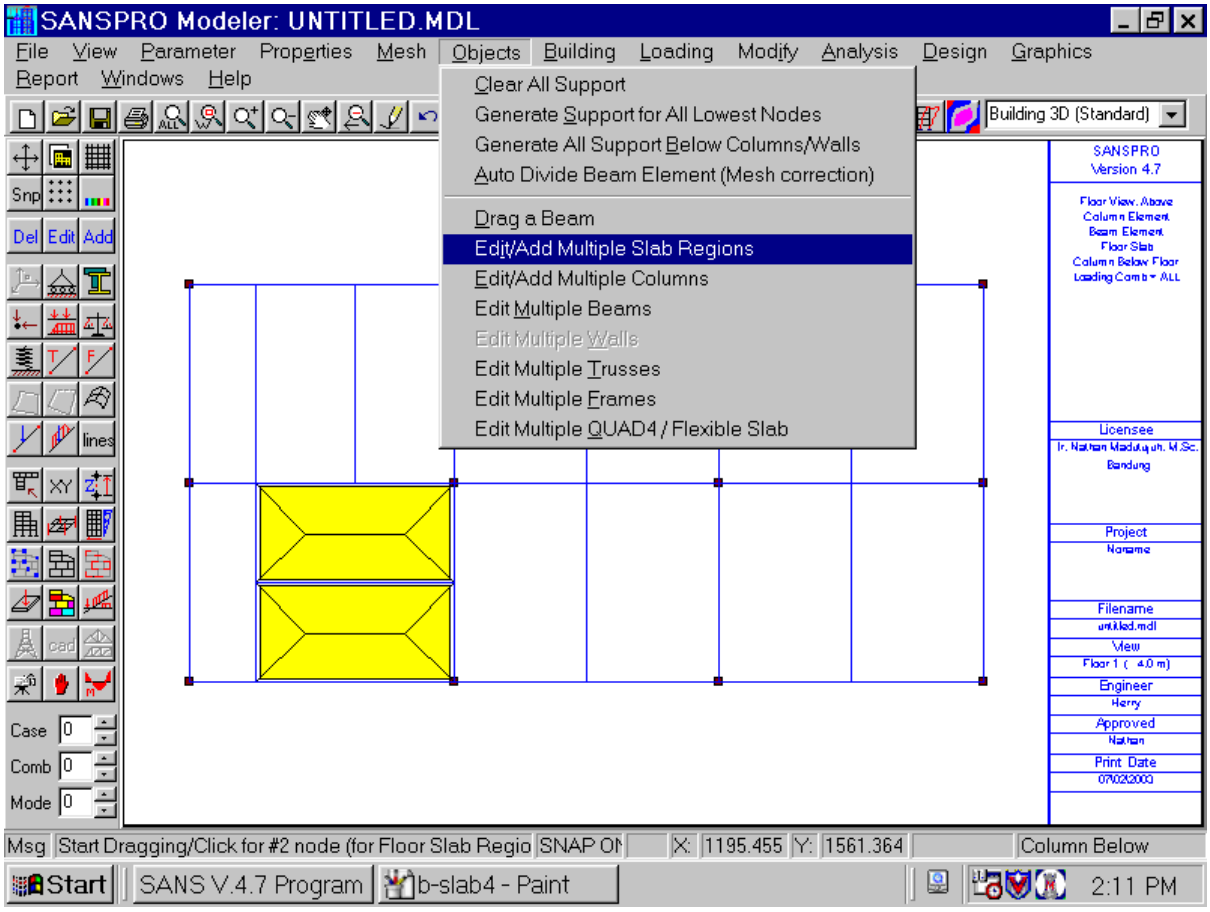


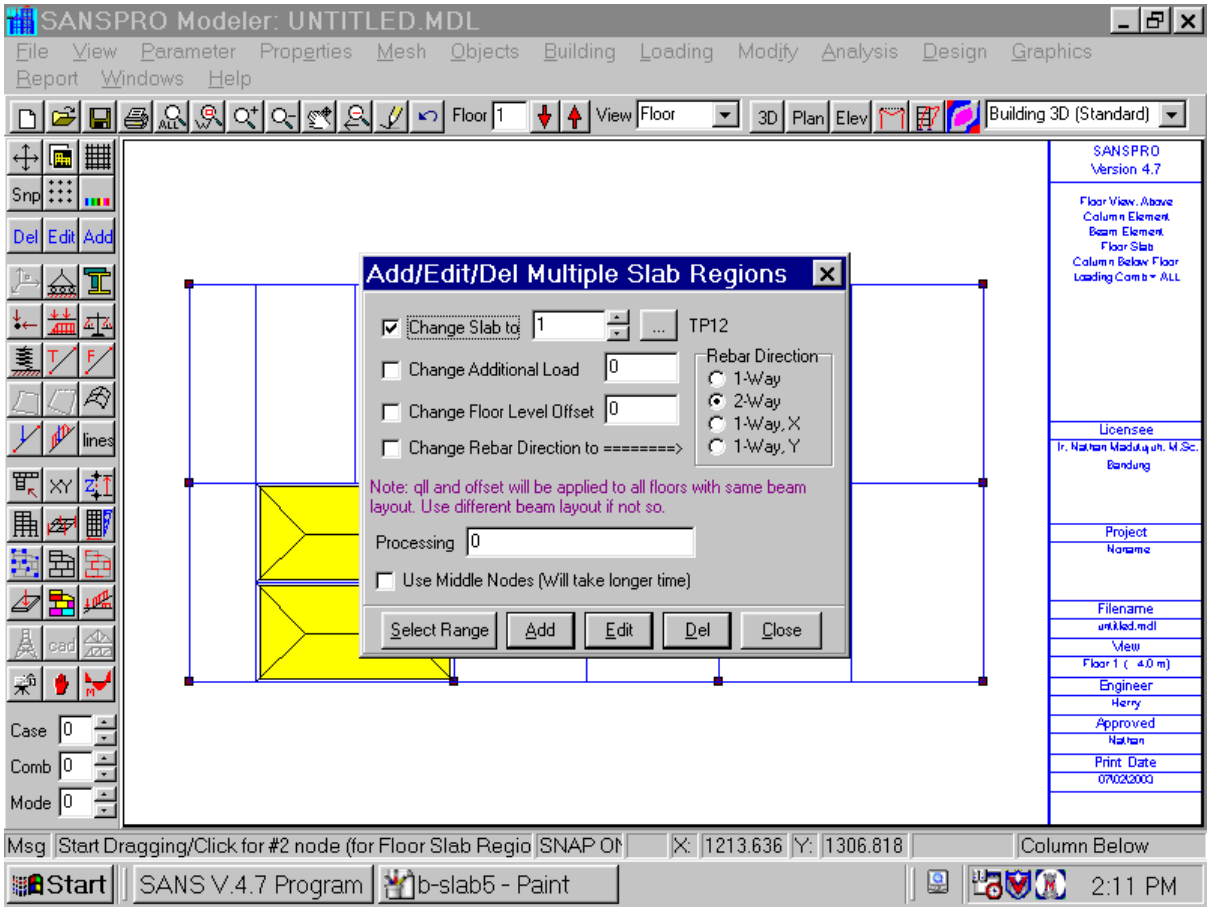
- Alternatively **(recommended)**, one can add all the regions of a floor at once using automatic floor regions generator.

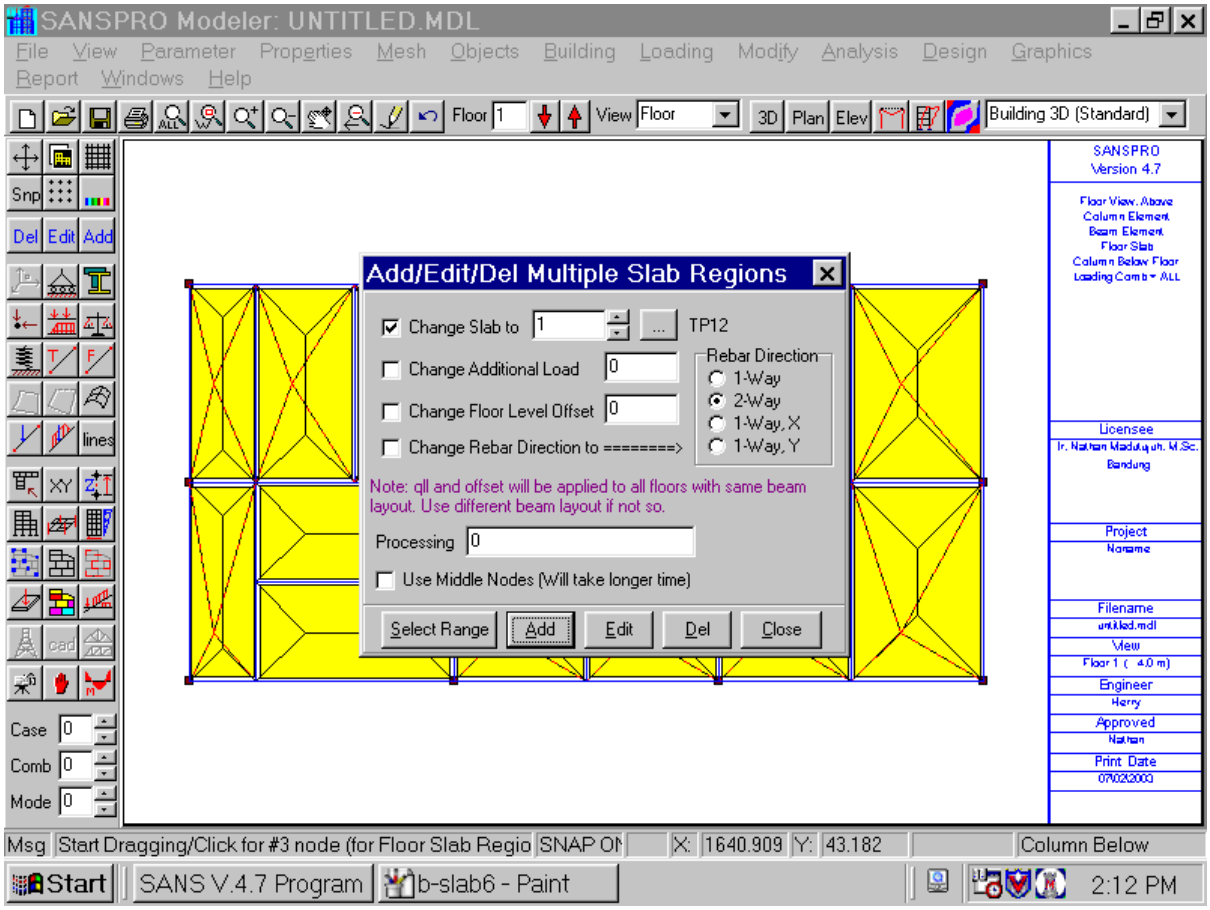

SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
Floor Slab Region Dialog
Slab Data Generate
Slab No. 1 TP12
Layout 1 (beam layout)
qll 0 (additional live load)
Option 2-Way
Offset 0 (Floor offset level)
Note: qll and offset will be applied to all floors with same beam layout. Use different beam layout if not so.
Use this slab for other new regions
OK Cancel
SANSPRO Version 4.7
Licensee
Project
Filename
untitled.mdl
Engineer
Approved
Print Date
Case 0
Comb 0
Mode 0
Msg Start Dragging/Click for #4 node (for Floor Slab Regio SNAP Of X: 209.091 Y: 302.273 Column Below
Start SANS V.4.7 Program b-slab1 - Paint 2:09 PM


SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
SANSPRO Version 4.7
Licensee
Project
Filename
untitled.mdl
Engineer
Approved
Print Date
Case 0
Comb 0
Mode 0
Msg Start Dragging/Click for #2 node (for Floor Slab Regio SNAP Of X: 1081.818 Y: 115.909 Column Below
Start SANS V.4.7 Program b-slab2 - Paint 2:10 PM

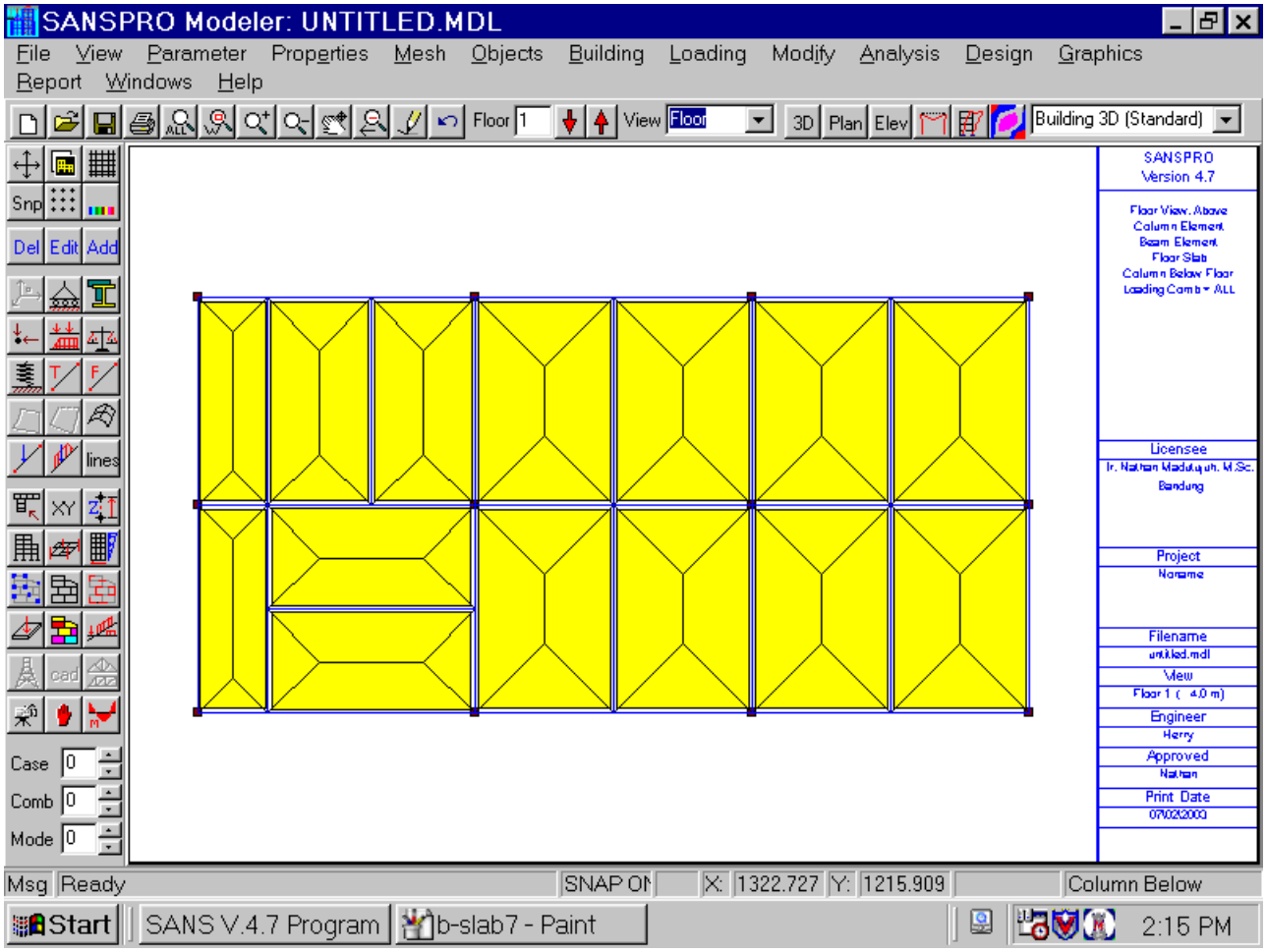

SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
Snp Del Edit Add lines XY
Case 0 Comb 0 Mode 0
SANSPRO Version 4.7
Floor View, Above
Column Element
Beam Element
Floor Slab
Column Below Floor
Loading Comb = ALL
Licensee
Ir. Nathan Maduluh, M.Sc.
Bandung
Project
Noname
Filename
untitled.mdl
View
Floor 1 (4.0 m)
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Msg Start Dragging/Click for #2 node (for Floor Slab Regio SNAP ON X: 1018.182 Y: 152.273 Column Below
Start SANS V.4.7 Program b-slab3 - Paint 2:11 PM


SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Clear All Support
Generate Support for All Lowest Nodes
Generate All Support Below Columns/Walls
Auto Divide Beam Element (Mesh correction)
Drag a Beam
Edit/Add Multiple Slab Regions
Edit/Add Multiple Columns
Edit Multiple Beams
Edit Multiple Walls
Edit Multiple Trusses
Edit Multiple Frames
Edit Multiple QUAD4 / Flexible Slab
Building 3D (Standard)
Snp Del Edit Add lines XY
Case 0 Comb 0 Mode 0
SANSPRO Version 4.7
Licensee
Ir. Nathan Maduluh, M.Sc.
Bandung
Project
Noname
Filename
untitled.mdl
View
Floor 1 (4.0 m)
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Msg Start Dragging/Click for #2 node (for Floor Slab Regio SNAP ON X: 1195.455 Y: 1561.364 Column Below
Start SANS V.4.7 Program b-slab4 - Paint 2:11 PM


SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1
View Floor
3D Plan Elev
Building 3D (Standard)
Add/Edit/Del Multiple Slab Regions
Change Slab to 1 ... TP12
Change Additional Load 0
Change Floor Level Offset 0
Change Rebar Direction to ========>
Rebar Direction
1-Way
2-Way
1-Way, X
1-Way, Y
Note: qll and offset will be applied to all floors with same beam layout. Use different beam layout if not so.
Processing 0
Use Middle Nodes (Will take longer time)
Select Range
Add
Edit
Del
Close
SANSPRO Version 4.7
Licensee
Project
Filename
untitled.mdl
View
Engineer
Approved
Print Date
Case 0
Comb 0
Mode 0
Msg Start Dragging/Click for #2 node (for Floor Slab Regio
X: 1213.636 Y: 1306.818
Column Below
Start
SANS V.4.7 Program
b-slab5 - Paint
2:11 PM


SANSPRO Modeler: UNTITLED.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1
View Floor
3D Plan Elev
Building 3D (Standard)
Add/Edit/Del Multiple Slab Regions
Change Slab to 1 ... TP12
Change Additional Load 0
Change Floor Level Offset 0
Change Rebar Direction to ========>
Rebar Direction
1-Way
2-Way
1-Way, X
1-Way, Y
Note: qll and offset will be applied to all floors with same beam layout. Use different beam layout if not so.
Processing 0
Use Middle Nodes (Will take longer time)
Select Range
Add
Edit
Del
Close
SANSPRO Version 4.7
Licensee
Project
Filename
untitled.mdl
View
Engineer
Approved
Print Date
Case 0
Comb 0
Mode 0
Msg Start Dragging/Click for #3 node (for Floor Slab Regio
X: 1640.909 Y: 43.182
Column Below
Start
SANS V.4.7 Program
b-slab6 - Paint
2:12 PM

- The red lines is the iteration/search lines used in the generation process
- After generating all floor slab regions, the floor layout will look as follows:



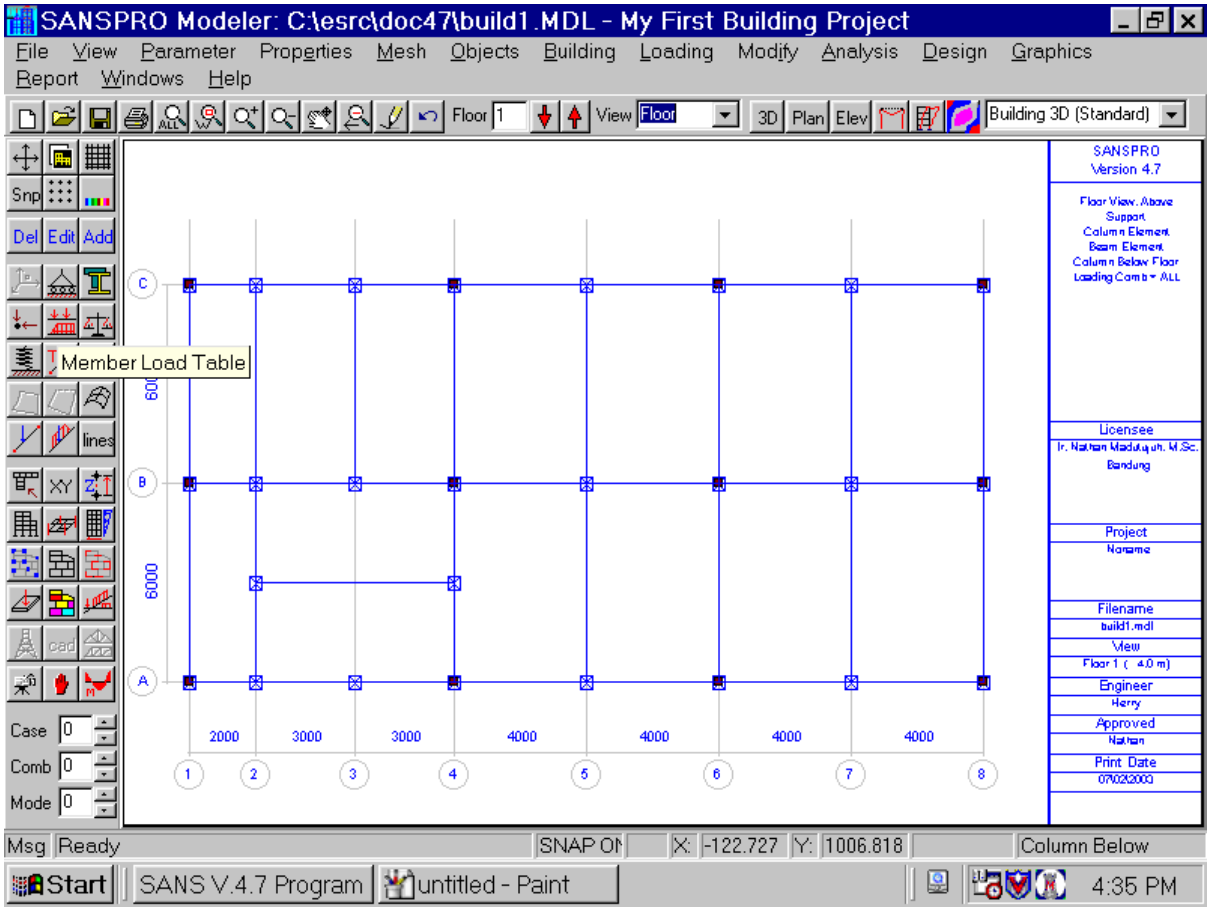
- Press **F2** to save the model

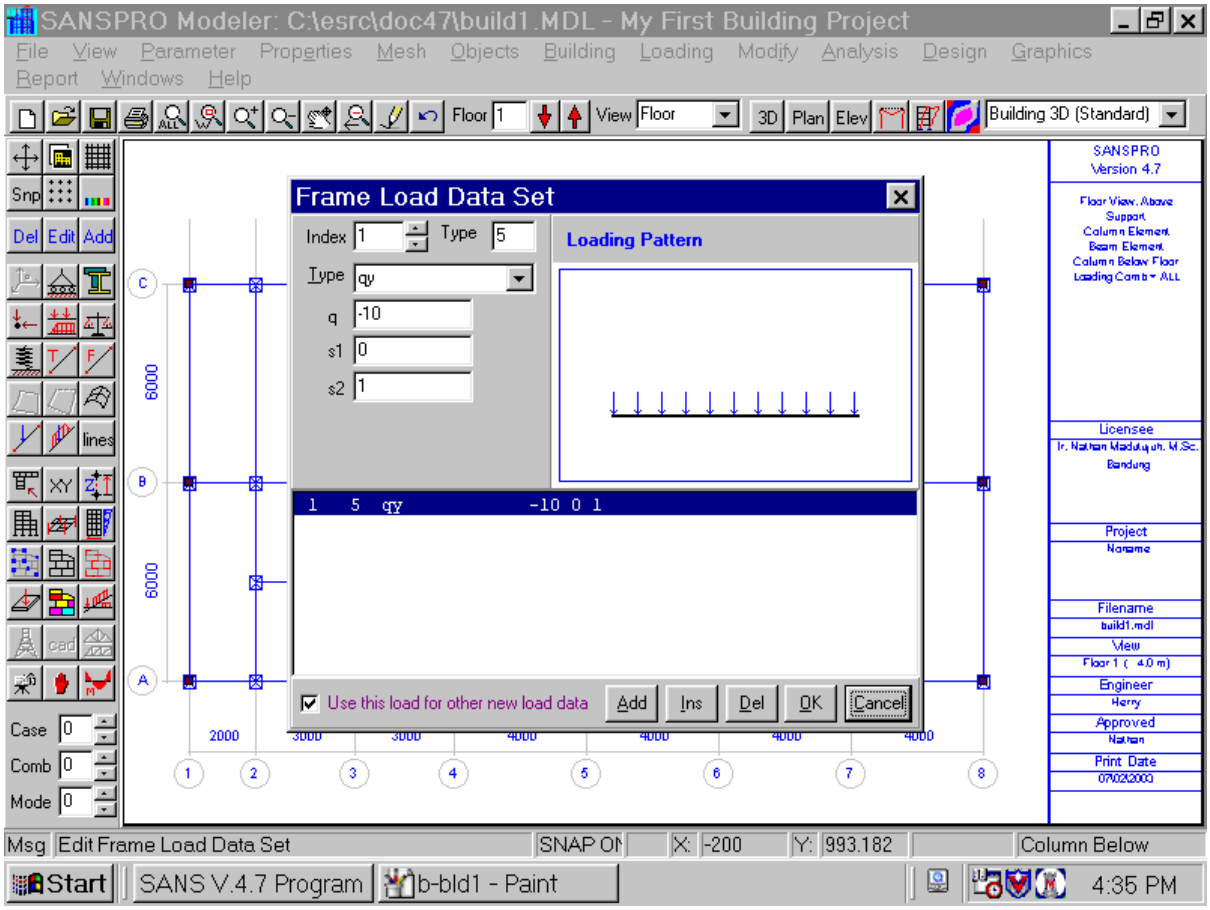
14. Adding Beam Load

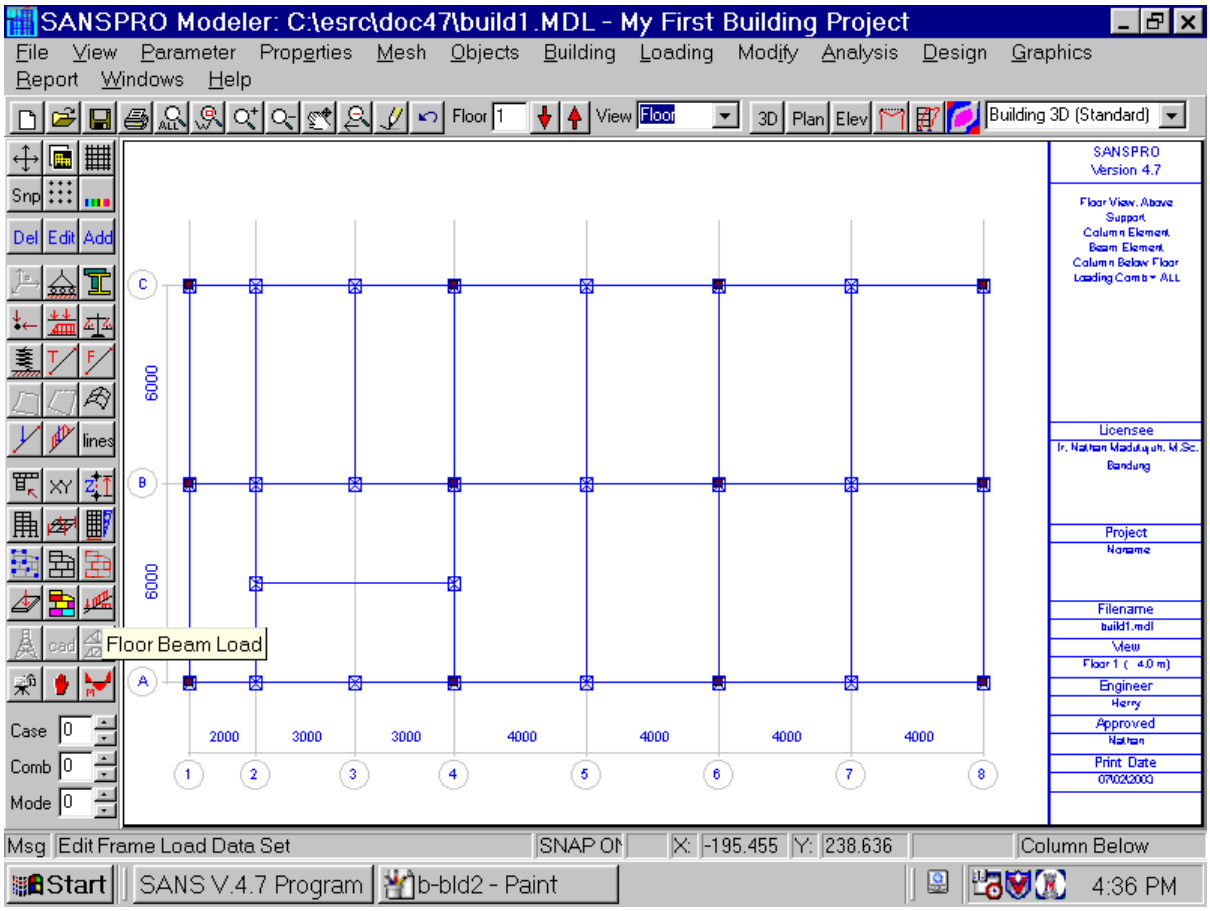
- To add some beam loads, we must create a beam load table first then assign the beam load to beams with beam load.
- SANSPRO provides 18 load types, including point load, distributed load, prestress load.
- Beam load table use absoulte distance (d1) or relative distance for load pattern definition, for example : s1 = d1/L
- Load units are kg, kg.cm, kg/cm

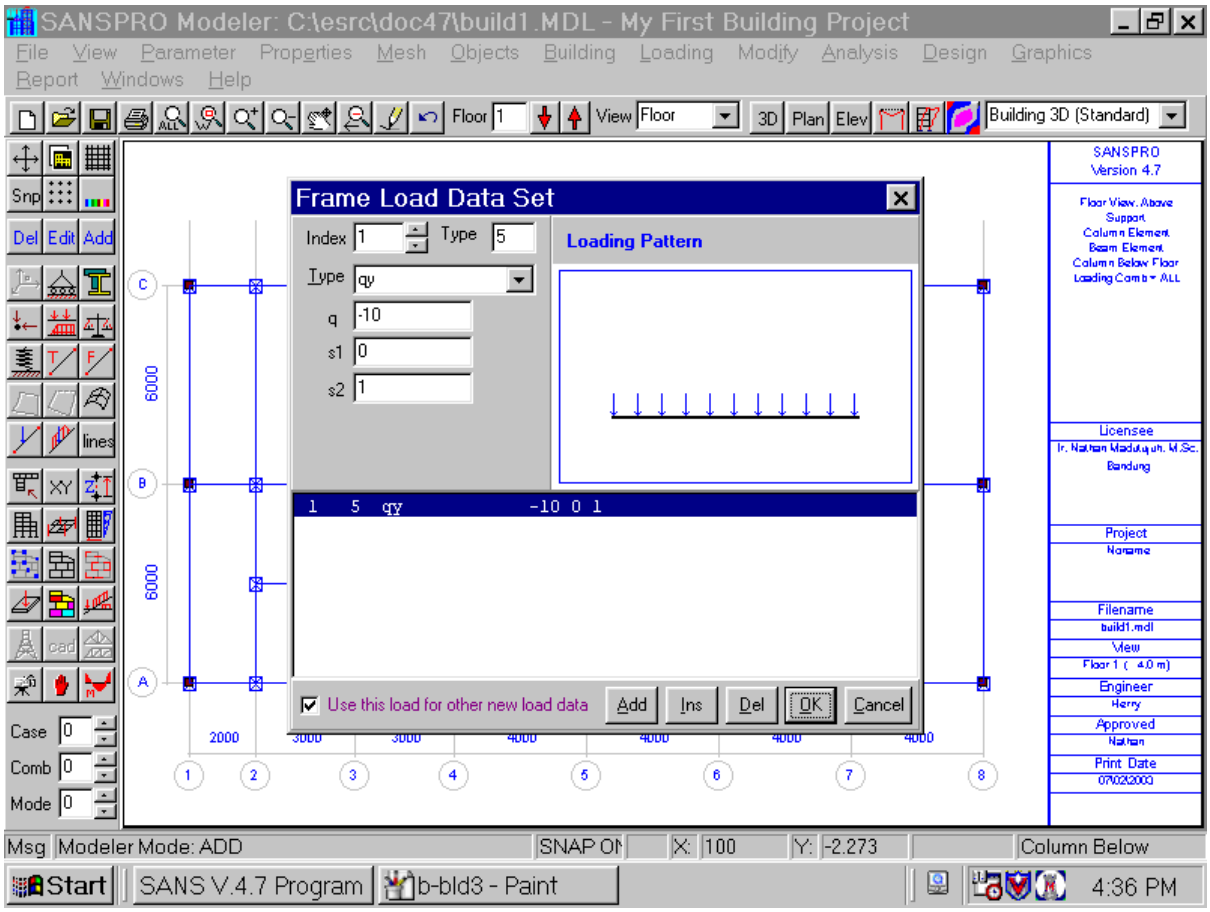
- Adding Member Load Table:
    - Click Member Load Table
    - Click Add
    - Select Load Type
    - Enter load magnitude and parameters

- Add beam load after creating the Member Load Table:
    - Click Beam Load
    - Click icon Add
    - Click target beam
    - Select beam load index
    - Select next target

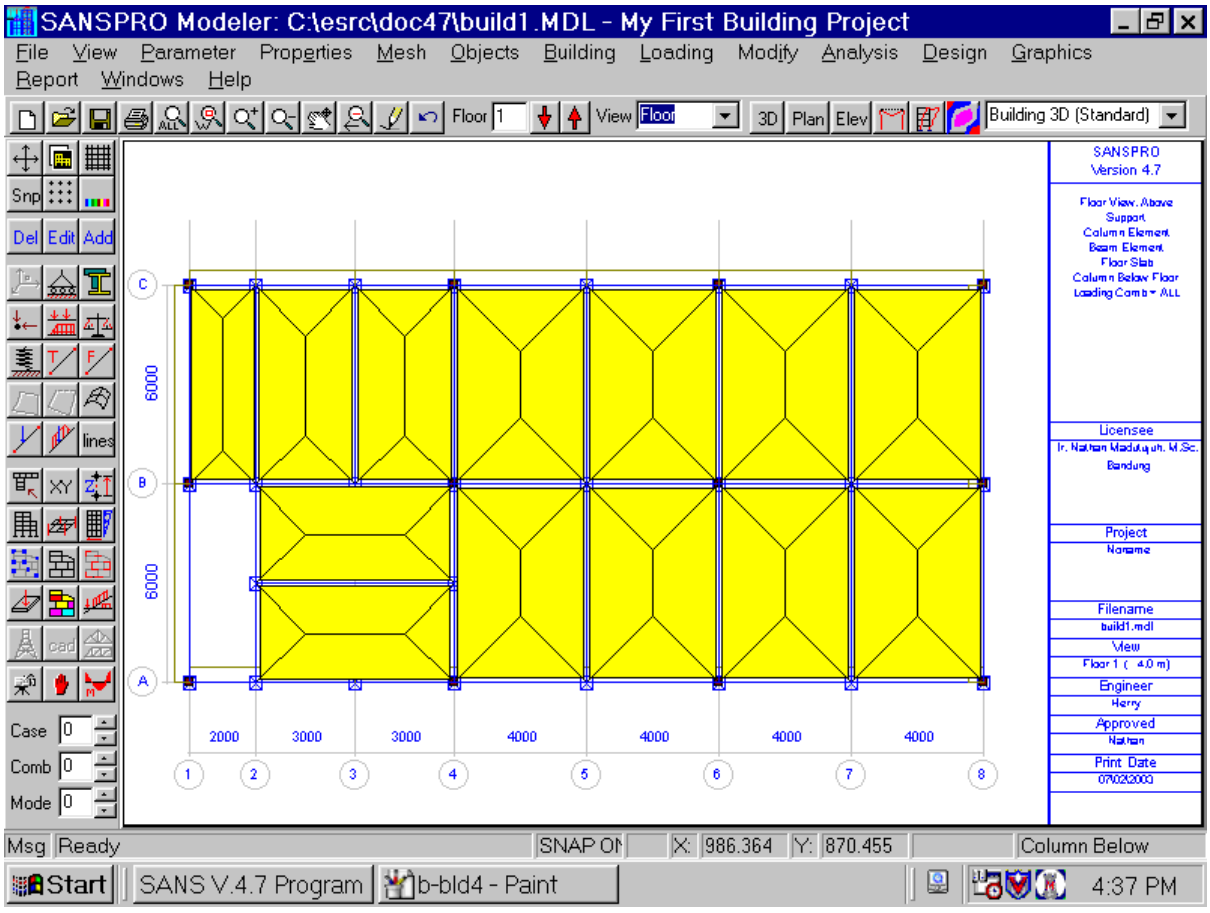



SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Frame Load Data Set
Index 1 Type 5
Type qy
q -10
s1 0
s2 1
Loading Pattern
1 5 qy -10 0 1
Use this load for other new load data
Add Ins Del OK Cancel
Msg Edit Frame Load Data Set
X: -200 Y: 993.182
Column Below
Start
SANS V.4.7 Program
b-bld1 - Paint
4:35 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor Beam Load
Msg Edit Frame Load Data Set
X: -195.455 Y: 238.636
Column Below
Start
SANS V.4.7 Program
b-bld2 - Paint
4:36 PM

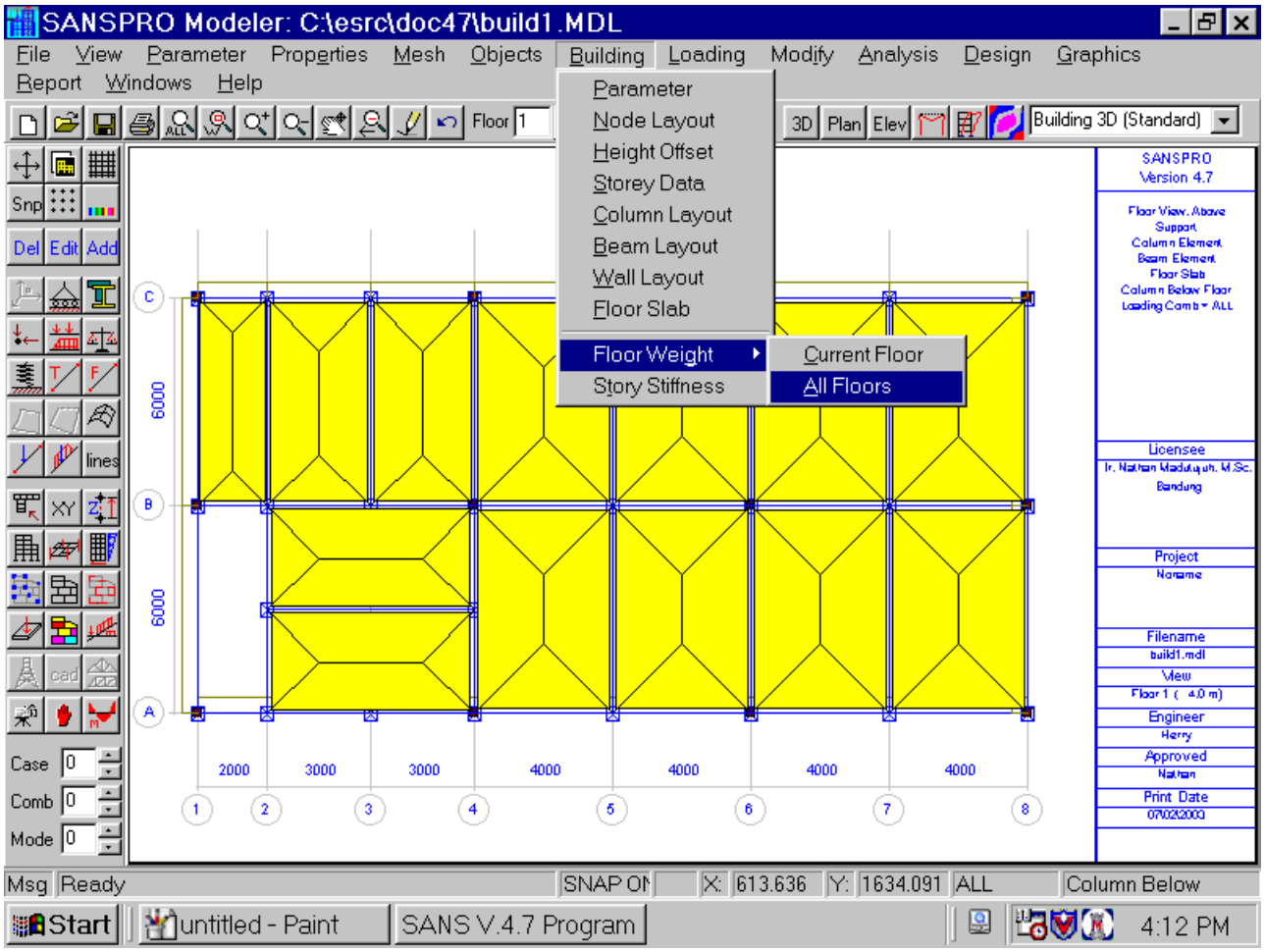


- Repeat for other beams:



15. Compute Floor Weight

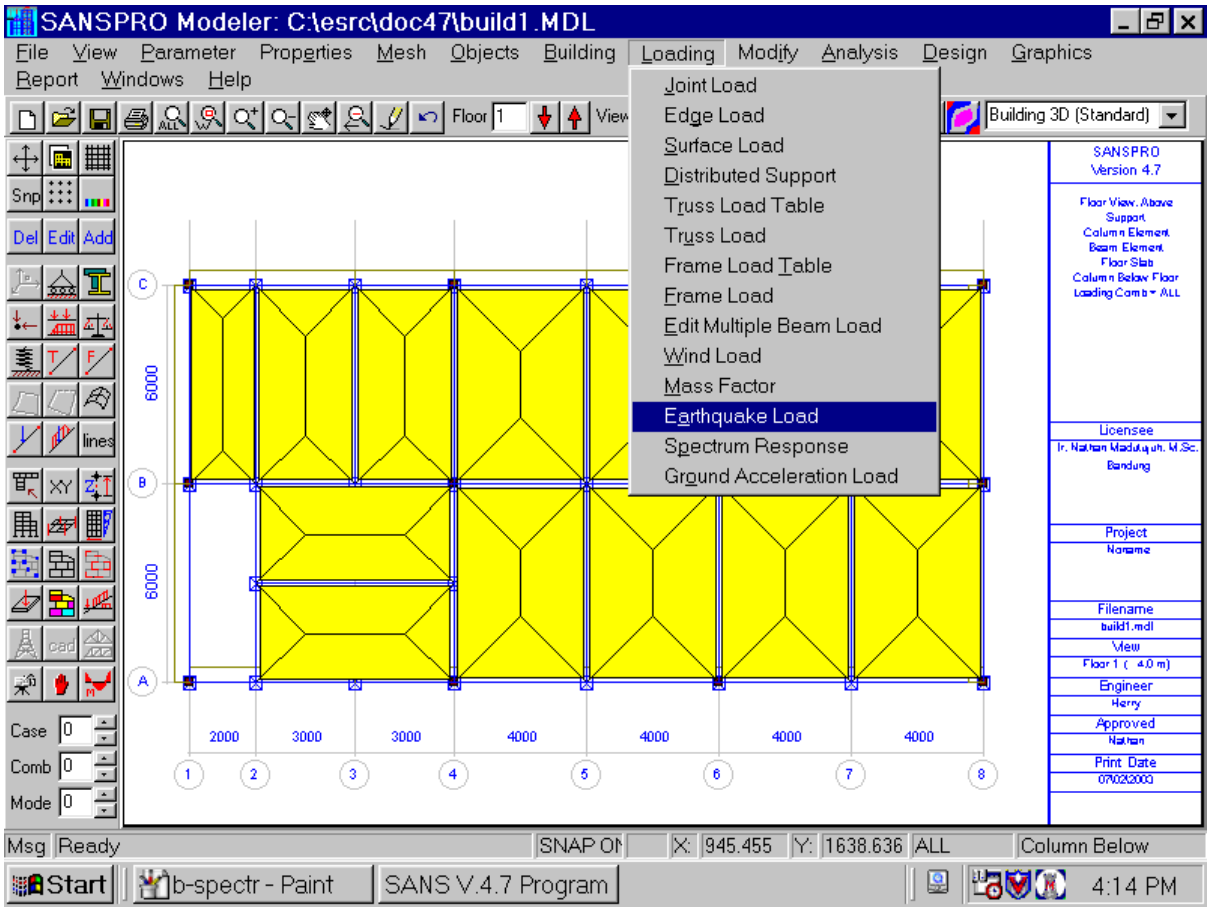
- For Static Equivalent Analysis, SANSPRO can calculate floor weight of all floors automatically using unit weight of each material use, section properties, and floor dead load and live load with live load reduction factors.
- User must input Cx, Cz, WidthX, WidthZ

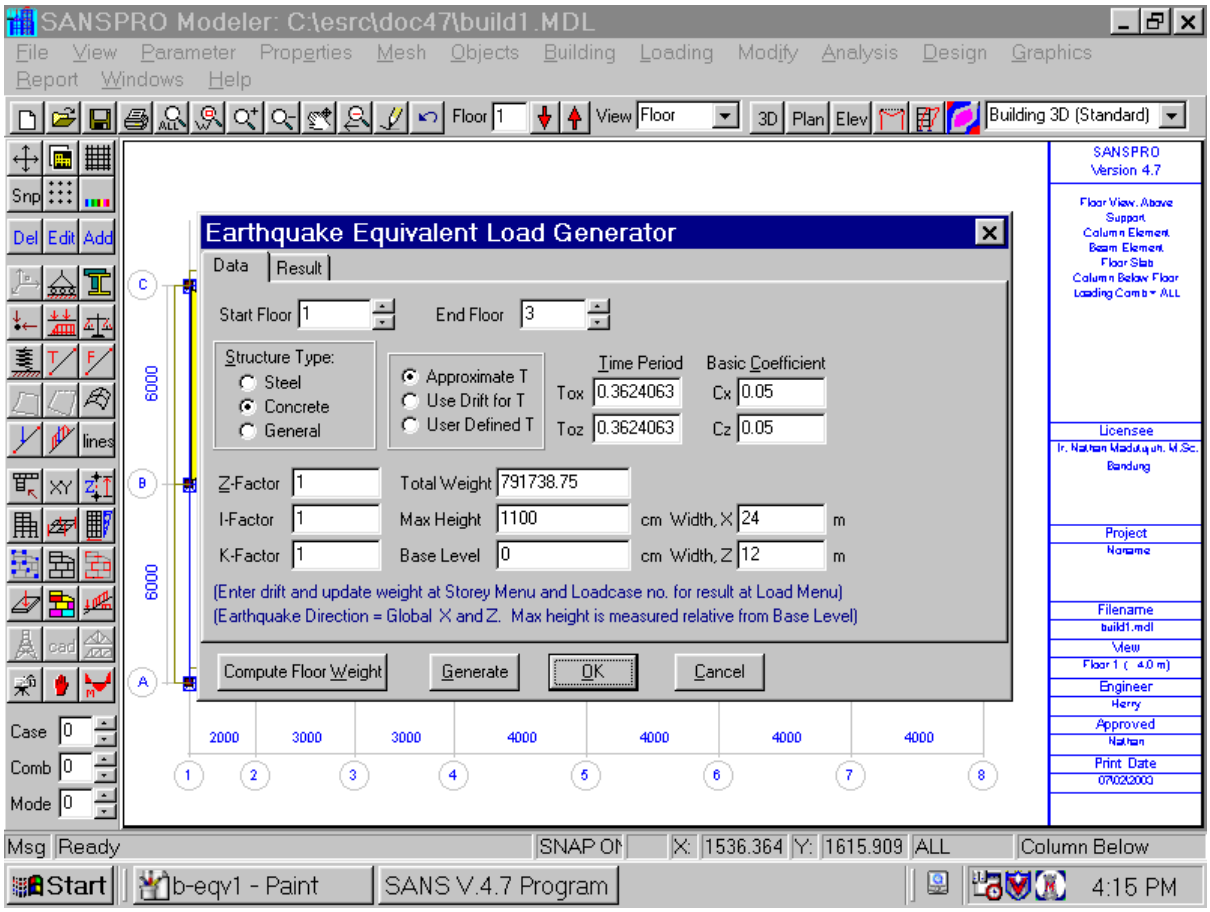


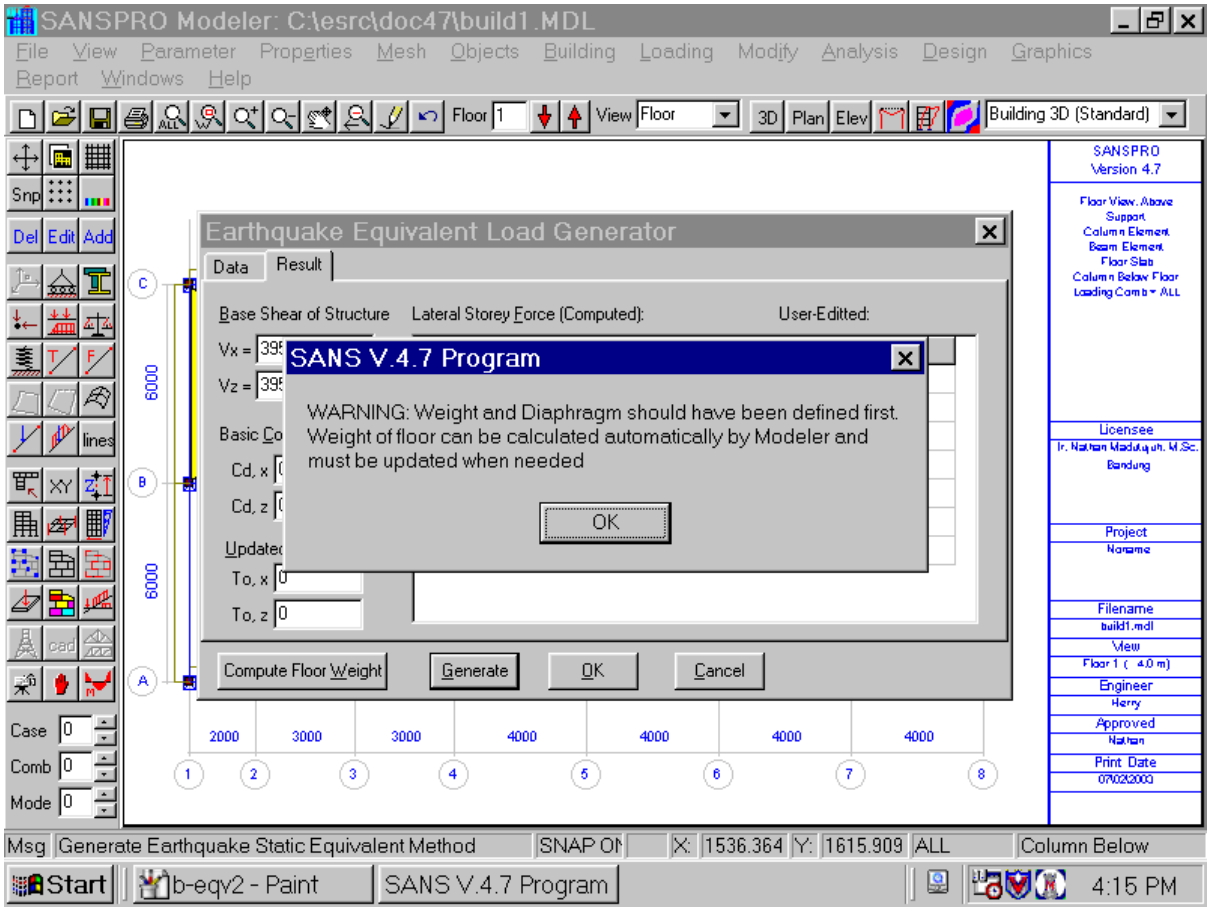
- Storey Data Table with updated floor weight and center of mass will appear
- Center of Mass can be shifted to enlarge eccentricity after first analysis to get center of rigidity.
- The design eccentricity can be computed also automatically by selecting the enlargement option and button *Compute Design Eccentricity*

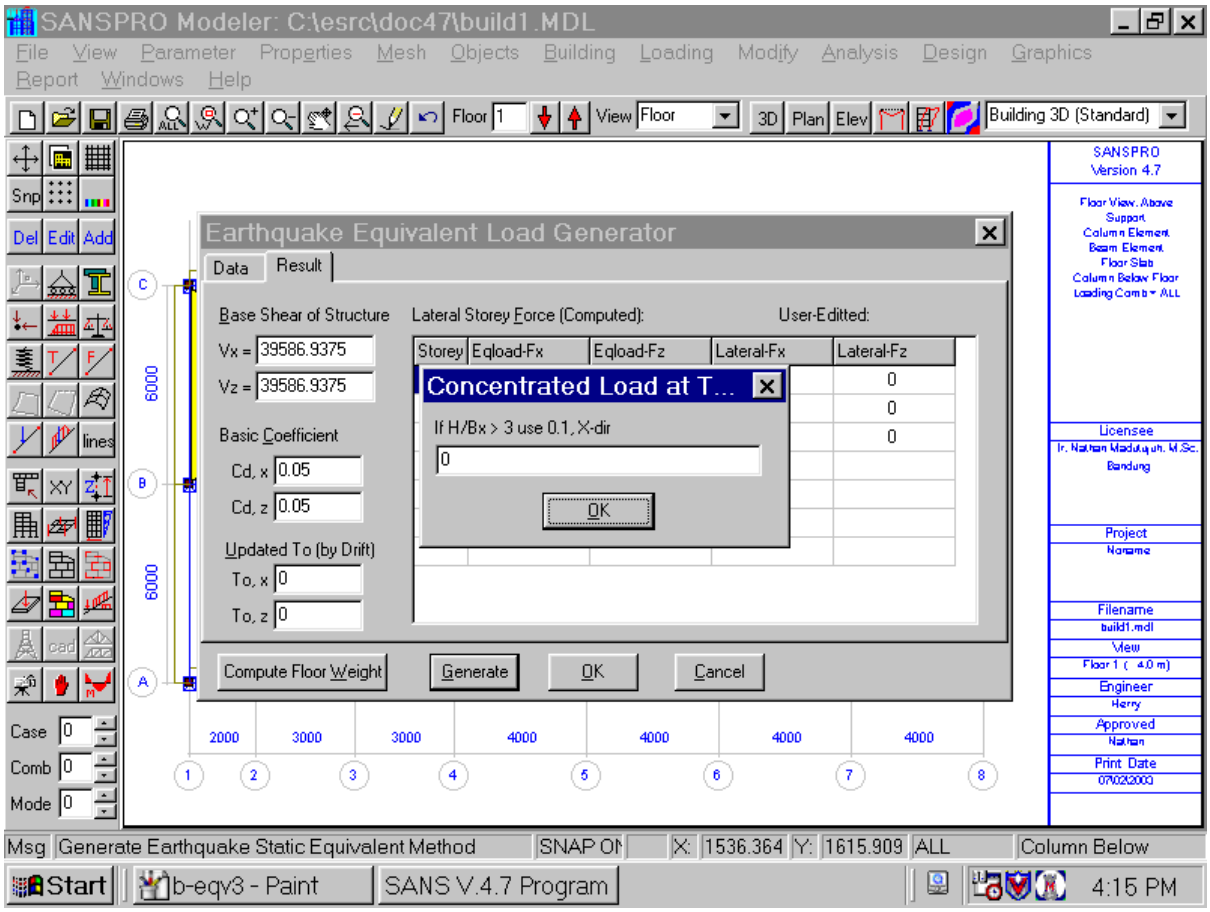
16. Generating Equivalent Load

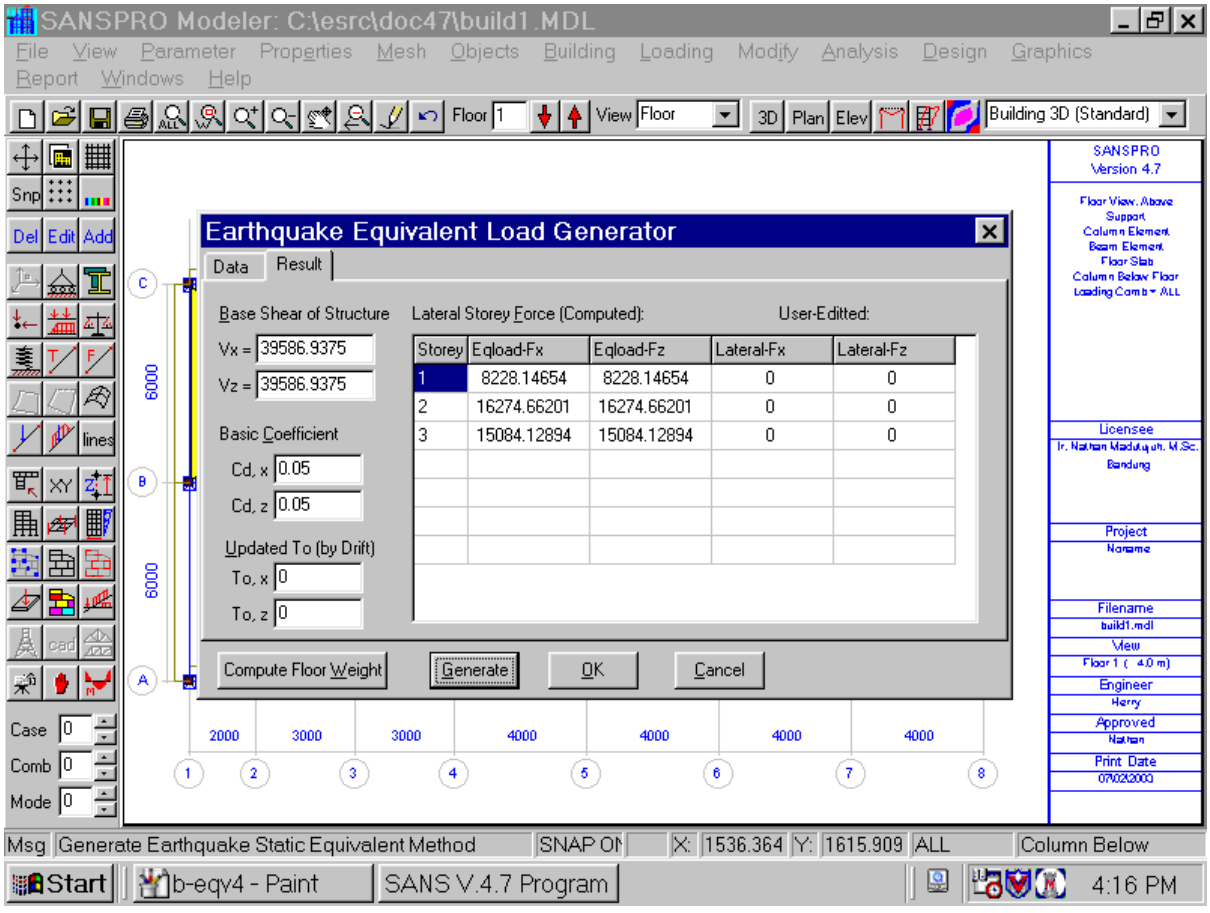
- SANSPRO generates equivalent load from floor weight, design earthquake coeficient, floor number, and building width and height.
- One can put additional load at top floor, usually 0.1 to 0.2 V.
- Natural period (To) will be calculated using approximate/empirical formula.
- For refined solution, one can enter drift for each floor to calculate more precisely the natural period (To) of the building




SANSPRO Modeler: C:\esrc\doc47\build1.MDL
Earthquake Equivalent Load Generator
Data
Result
Start Floor 1
End Floor 3
Structure Type:
Steel
Concrete
General
Approximate T
Use Drift for T
User Defined T
Time Period
Tox 0.3624063
Toz 0.3624063
Basic Coefficient
Cx 0.05
Cz 0.05
Z-Factor 1
I-Factor 1
K-Factor 1
Total Weight 791738.75
Max Height 1100
Base Level 0
Width, X 24 m
Width, Z 12 m
(Enter drift and update weight at Storey Menu and Loadcase no. for result at Load Menu)
(Earthquake Direction = Global X and Z. Max height is measured relative from Base Level)
Compute Floor Weight
Generate
OK
Cancel
Msg Ready
Column Below
Start
b-eqv1 - Paint
SANS V.4.7 Program
4:15 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL
Earthquake Equivalent Load Generator
Data
Result
Base Shear of Structure
Lateral Storey Force (Computed):
User-Editted:
SANS V.4.7 Program
WARNING: Weight and Diaphragm should have been defined first.
Weight of floor can be calculated automatically by Modeler and
must be updated when needed
OK
Compute Floor Weight
Generate
OK
Cancel
Msg Generate Earthquake Static Equivalent Method
Column Below
Start
b-eqv2 - Paint
SANS V.4.7 Program
4:15 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Earthquake Equivalent Load Generator
Data Result
Base Shear of Structure
Vx = 39586.9375
Vz = 39586.9375
Basic Coefficient
Cd, x 0.05
Cd, z 0.05
Updated To (by Drift)
To, x 0
To, z 0
Lateral Storey Force (Computed):
User-Editted:
Storey Eqload-Fx Eqload-Fz Lateral-Fx Lateral-Fz
Concentrated Load at T...
If H/Bx > 3 use 0.1, X-dir
OK
Compute Floor Weight
Generate
OK
Cancel
Msg Generate Earthquake Static Equivalent Method
Start
b-eqv3 - Paint
SANS V.4.7 Program
4:15 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Earthquake Equivalent Load Generator
Data Result
Base Shear of Structure
Vx = 39586.9375
Vz = 39586.9375
Basic Coefficient
Cd, x 0.05
Cd, z 0.05
Updated To (by Drift)
To, x 0
To, z 0
Lateral Storey Force (Computed):
User-Editted:
Storey Eqload-Fx Eqload-Fz Lateral-Fx Lateral-Fz
1 8228.14654 8228.14654 0 0
2 16274.66201 16274.66201 0 0
3 15084.12894 15084.12894 0 0
Compute Floor Weight
Generate
OK
Cancel
Msg Generate Earthquake Static Equivalent Method
Start
b-eqv4 - Paint
SANS V.4.7 Program
4:16 PM

17. Run Analysis

- The modelling process is now finished.
- Press **F2** to save the model
- Now we can export the data to Analysis module and run it.

- Why we need to export the data for analysis ? SANSPRO model file contains complicated data for modelling, analysis, design, drawing, cost calculation, while analysis module only needs geometry, loading, and constraint information.

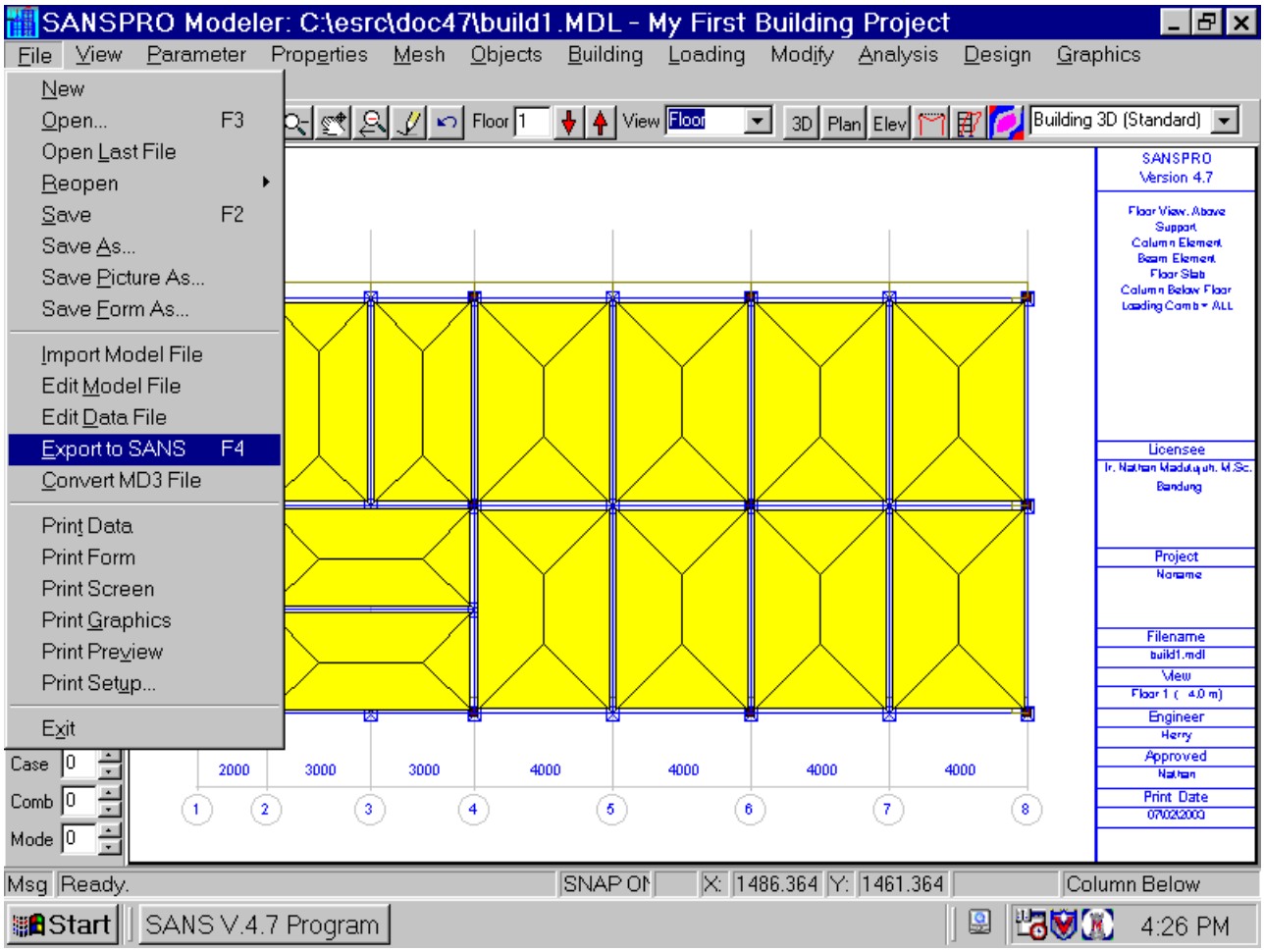
- Mesh Correction:

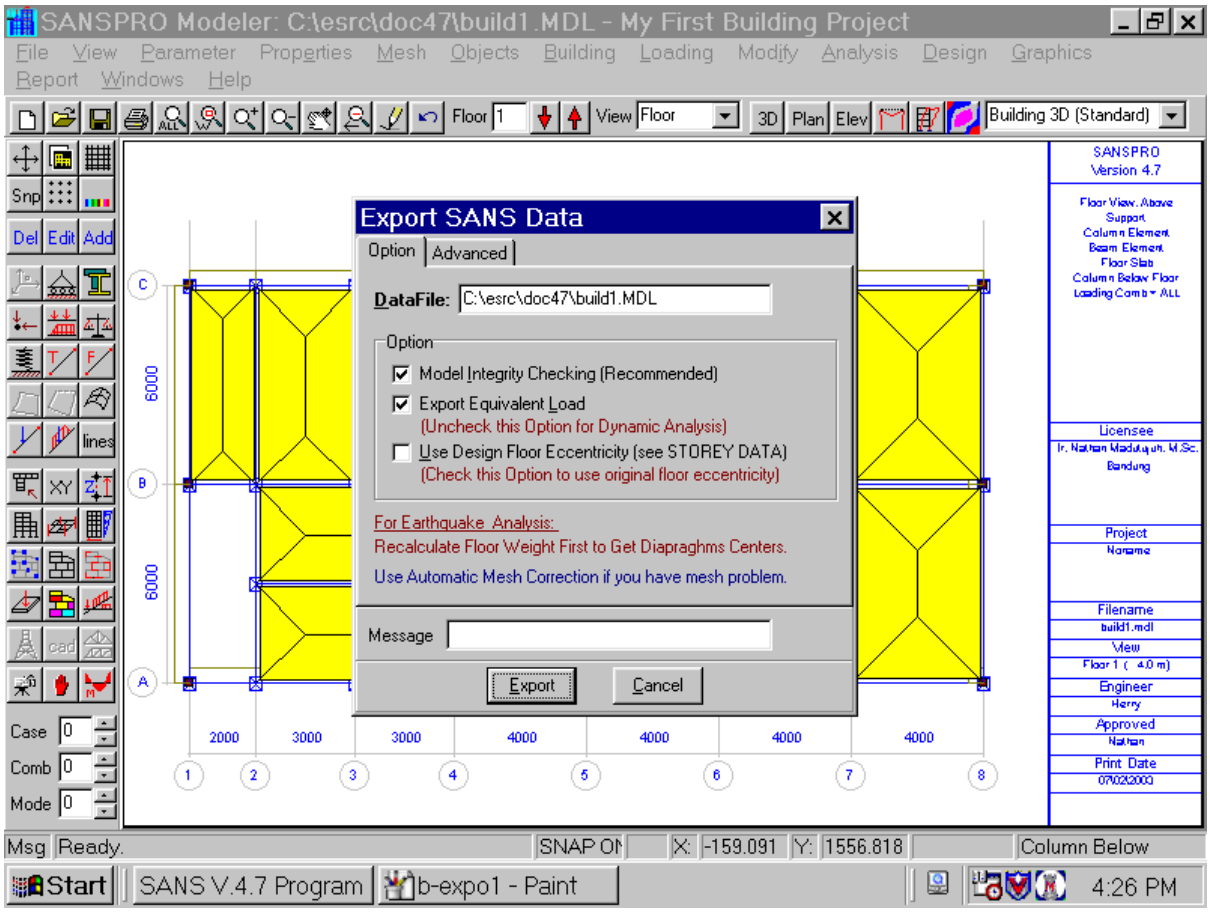
- Before run analysis, it is wise to check the mesh compatibility by calling Objects-Mesh Correction (Auto Fix). Every beam and column joints will be check for mesh compatibility and fixed automatically.

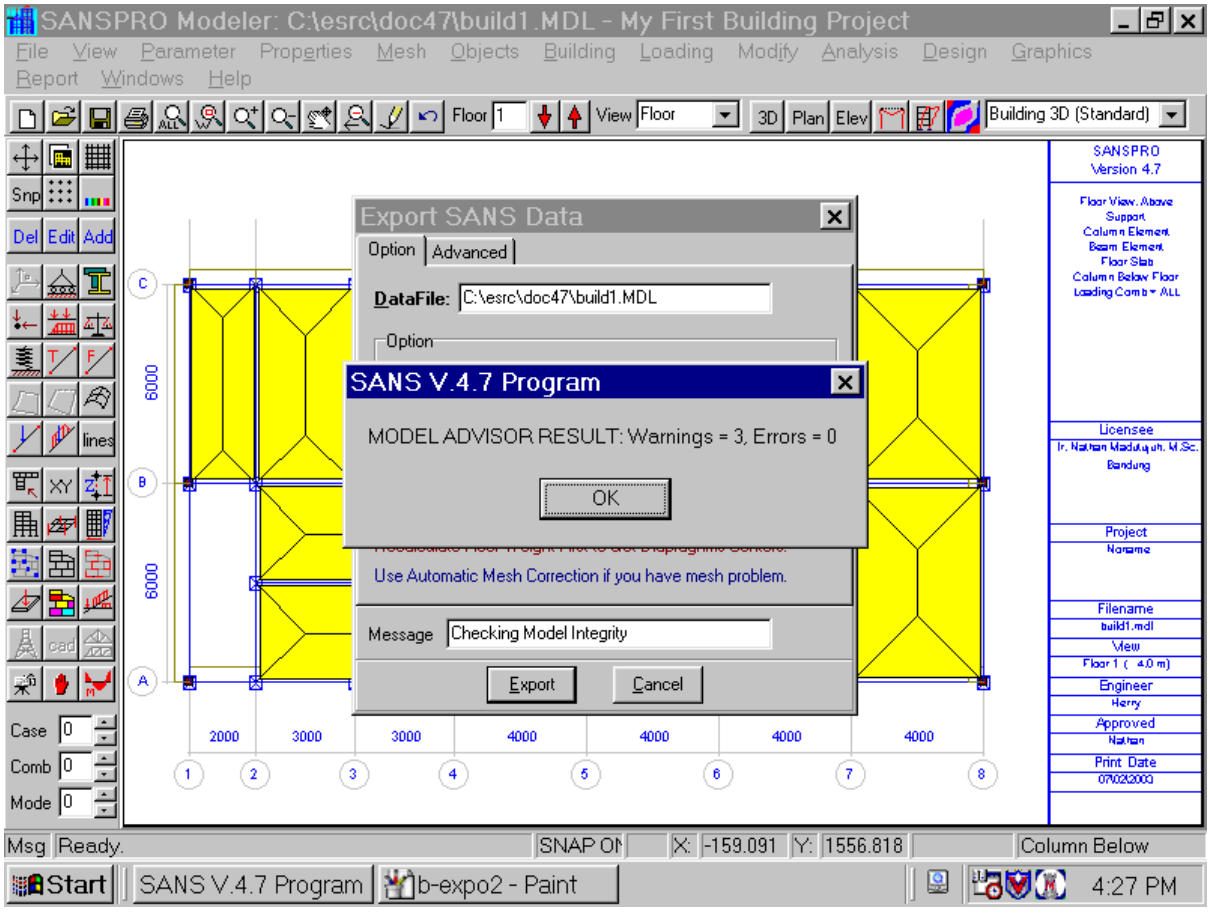
- Click File-Export or Press F4 to export the data
- SANSPRO will call for Model Advisor to intelligently check your model consistency.

- Check **Use Design Eccentricity** if you want to use the shifted center of mass
- Check **Export Equivalent Load** if you are using Static Equivalent Load Analysis

- Some of the warnings can be neglected if not applicable to the current model



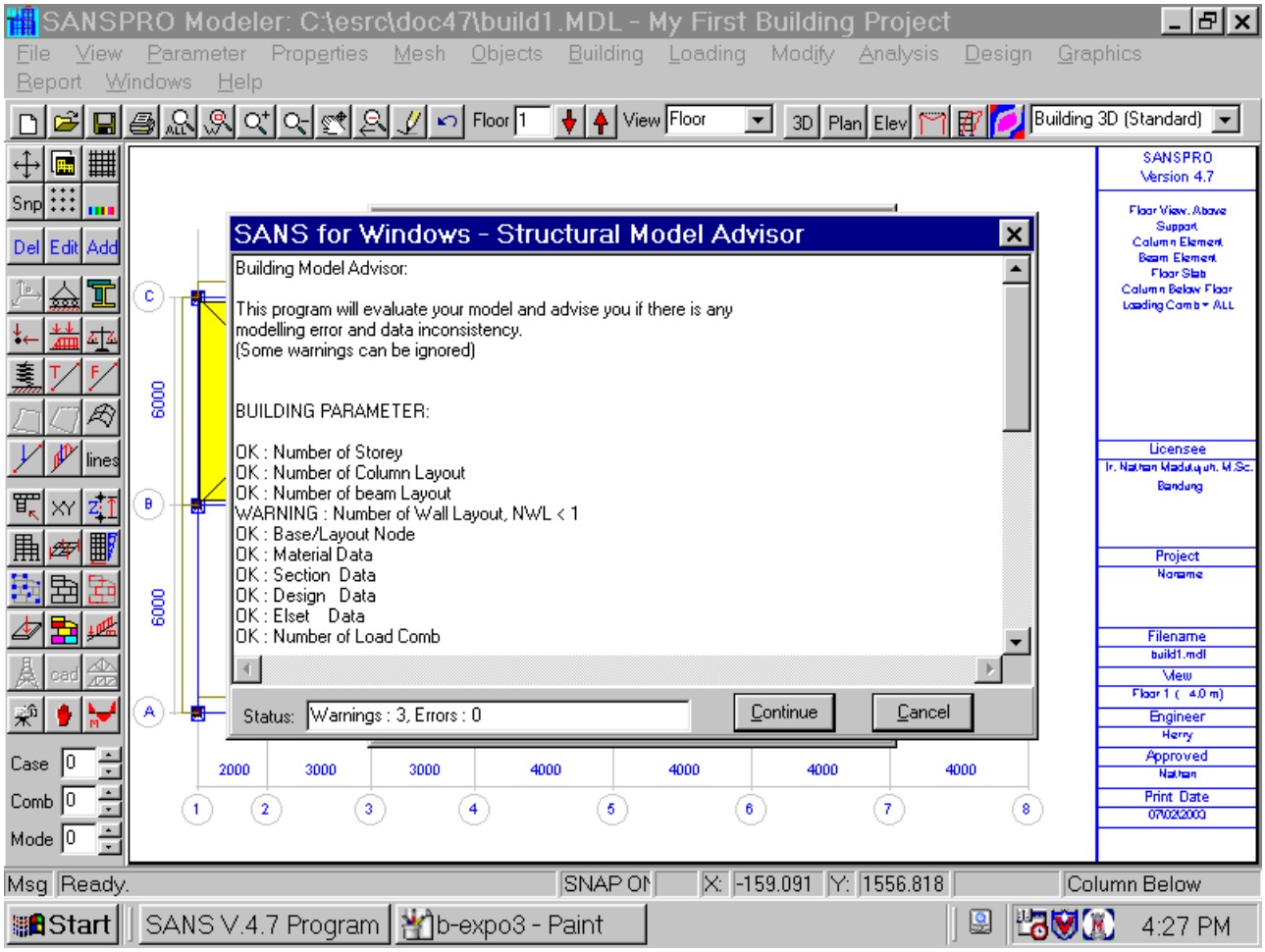

SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Export SANS Data
Option Advanced
DataFile: C:\esrc\doc47\build1.MDL
Option
Model Integrity Checking (Recommended)
Export Equivalent Load
(Uncheck this Option for Dynamic Analysis)
Use Design Floor Eccentricity (see STOREY DATA)
(Check this Option to use original floor eccentricity)
For Earthquake Analysis:
Recalculate Floor Weight First to Get Diapraghms Centers.
Use Automatic Mesh Correction if you have mesh problem.
Message
Export
Cancel
SANSPRO Version 4.7
Licensee
Project
Filename
build1.mdl
Engineer
Approved
Print Date
Msg Ready.
Column Below
Start
SANS V.4.7 Program
b-expo1 - Paint
4:26 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
Export SANS Data
SANS V.4.7 Program
MODEL ADVISOR RESULT: Warnings = 3, Errors = 0
OK
Message Checking Model Integrity
Export
Cancel
Msg Ready.
Column Below
Start
SANS V.4.7 Program
b-expo2 - Paint
4:27 PM

- What Model Advisor Check ?

- Invalid element properties
- Unconnected columns or beams
- Missing of support below columns or walls
- Missing Load data
- Zero Mass
- etc.

- Some of the warnings can be neglected if not applicable to the current model



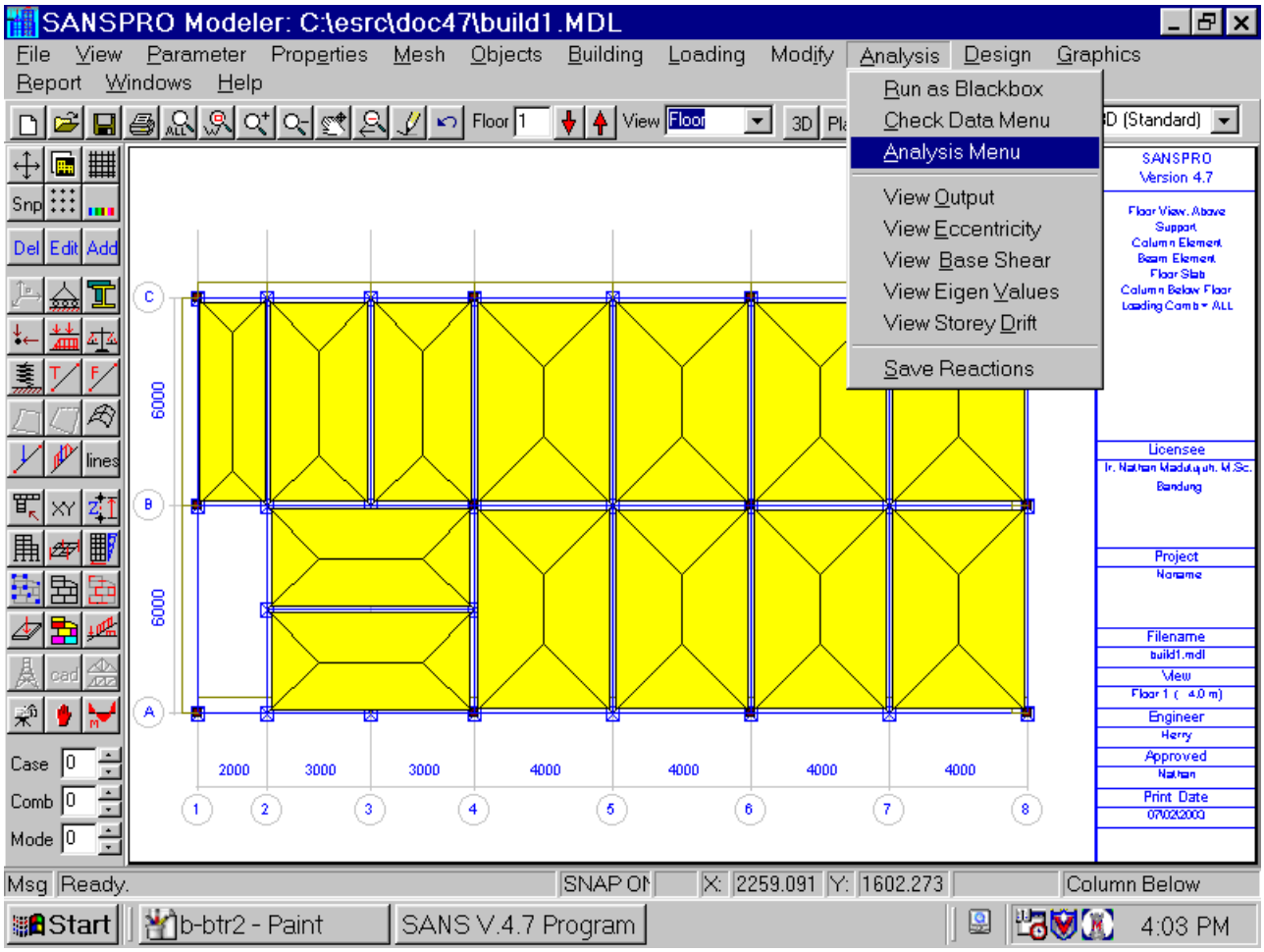
- After export, Click Run-Analysis Menu
- If you click Run as Blackbox, no warning messages will appear

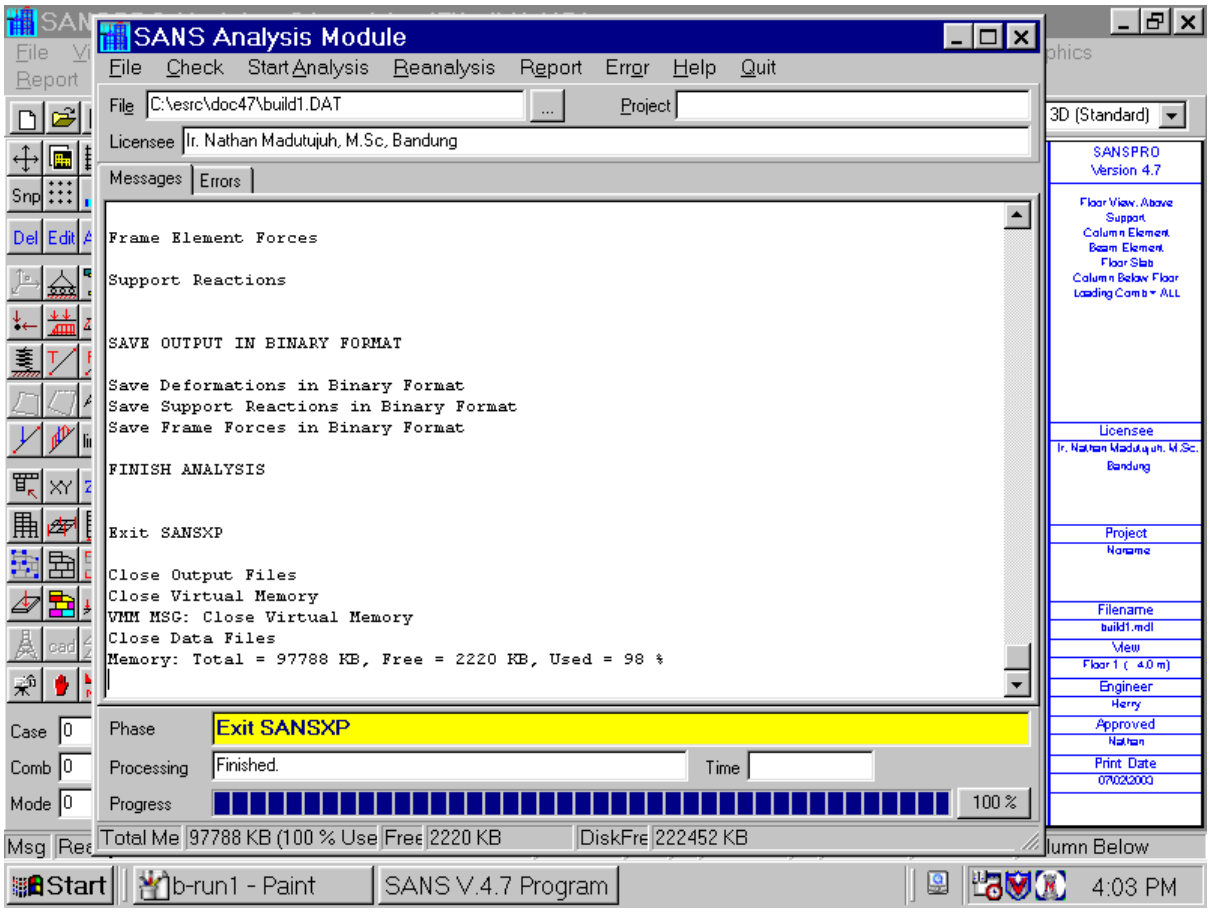
- If there are some errors, messages will be displayed.
- Please fix the error and do **F2-F4-Run** again

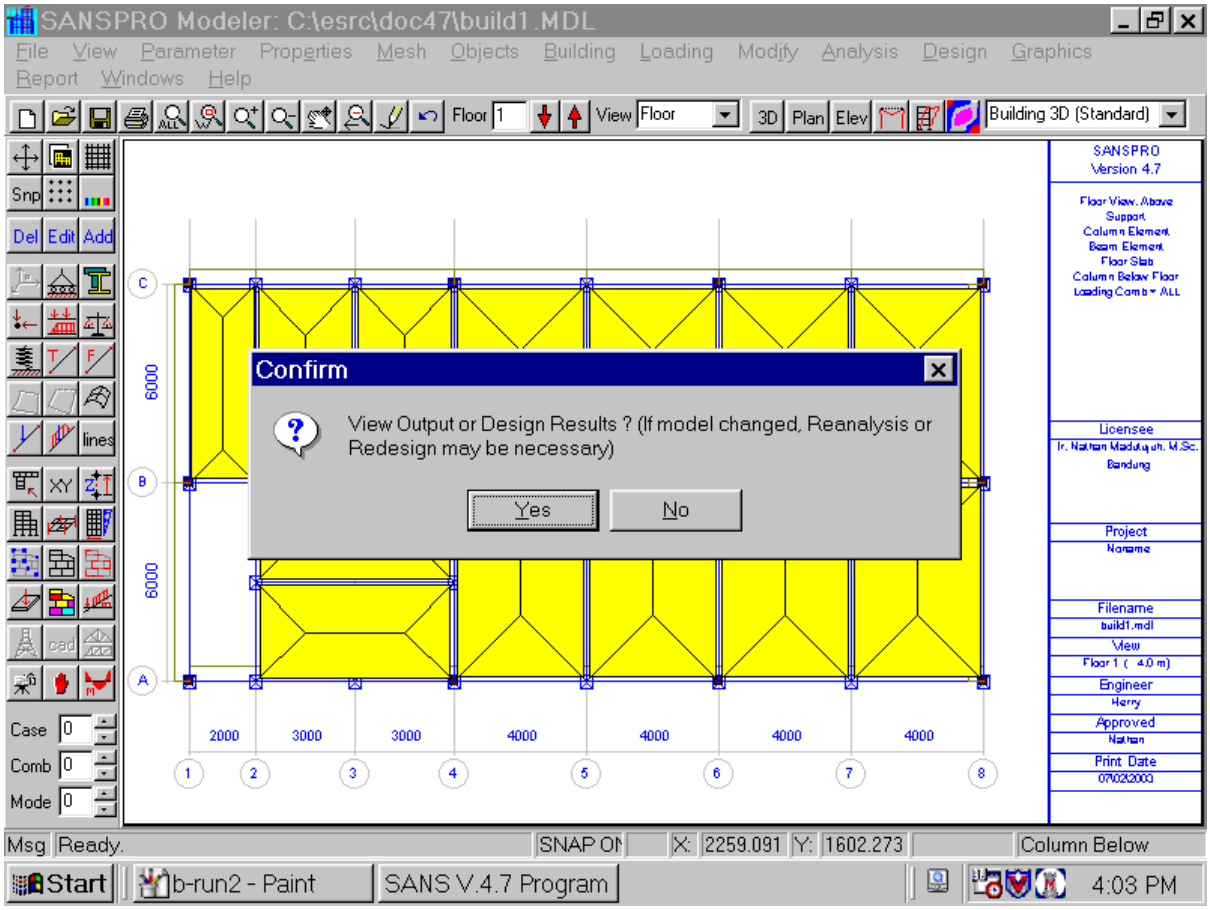
- After running, output file will be created and the model is ready for output view, moment diagram view.

BUILD1.DAT	- Data file from **Export Data**
BUILD1.REP	- Report File
BUILD1.GEN	- Expanded Data File
BUILD1.BIN	- Binary Format (Compacted) Output File
BUILD1.GRA	- Graphics File (SANSGRAPH)
BUILD1.OUT	- Text Format Output File (large size)

- SANSPRO will use BUILD1.BIN for visual reporting the output and moment diagram.
- BUILD1.OUT can be printted for complete report if asked
- After analysis, click **Quit** and Answer with **Yes** if you want to see the output view.



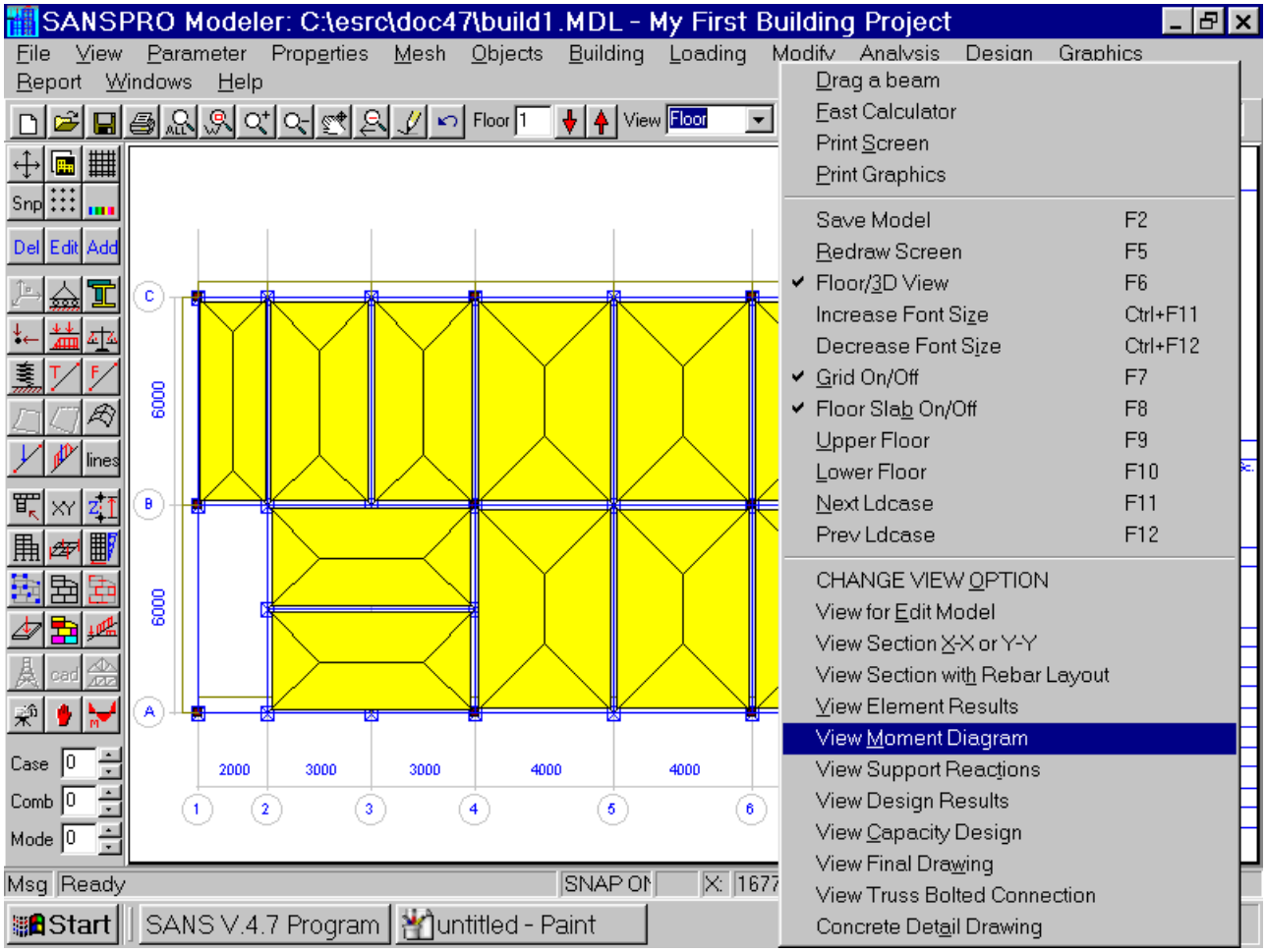

SANS Analysis Module
File Check Start Analysis Reanalysis Report Error Help Quit
File C:\esrc\doc47\build1.DAT
Project
Licensee Ir. Nathan Madutujuh, M.Sc, Bandung
Messages Errors
Frame Element Forces
Support Reactions
SAVE OUTPUT IN BINARY FORMAT
Save Deformations in Binary Format
Save Support Reactions in Binary Format
Save Frame Forces in Binary Format
FINISH ANALYSIS
Exit SANSXP
Close Output Files
Close Virtual Memory
VMM MSG: Close Virtual Memory
Close Data Files
Memory: Total = 97788 KB, Free = 2220 KB, Used = 98 %
Phase Exit SANSXP
Processing Finished.
Time
Progress
100 %
Total Me 97788 KB (100 % Use Free 2220 KB DiskFre 222452 KB
3D (Standard)
SANSPRO Version 4.7
Start
b-run1 - Paint
SANS V.4.7 Program
4:03 PM

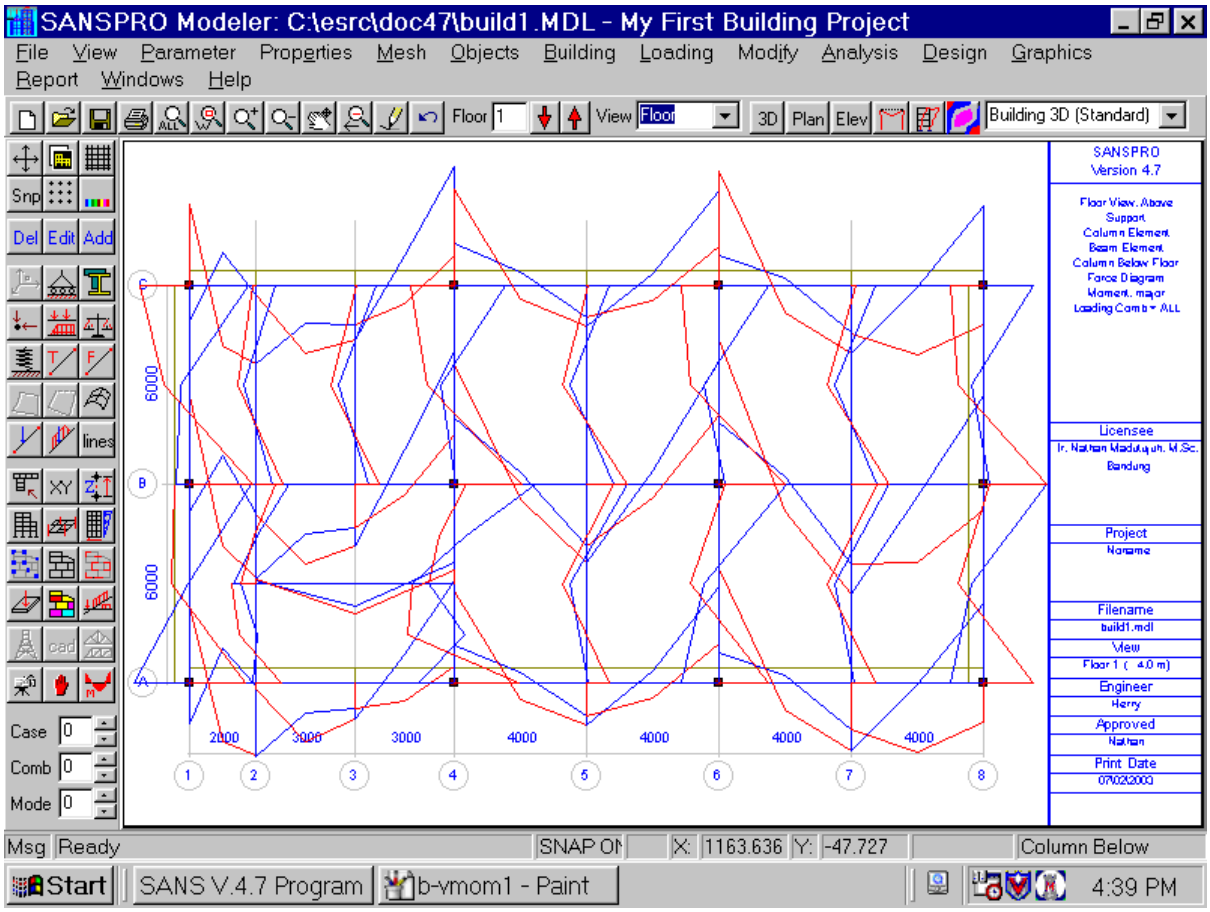

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1
View Floor
3D Plan Elev
Building 3D (Standard)
Confirm
View Output or Design Results ? (If model changed, Reanalysis or Redesign may be necessary)
Yes
No
6000
6000
2000 3000 3000 4000 4000 4000 4000
1 2 3 4 5 6 7 8
A B C
SANSPRO Version 4.7
Msg Ready.
SNAP Of
X: 2259.091 Y: 1602.273
Column Below
Start
b-run2 - Paint
SANS V.4.7 Program
4:03 PM

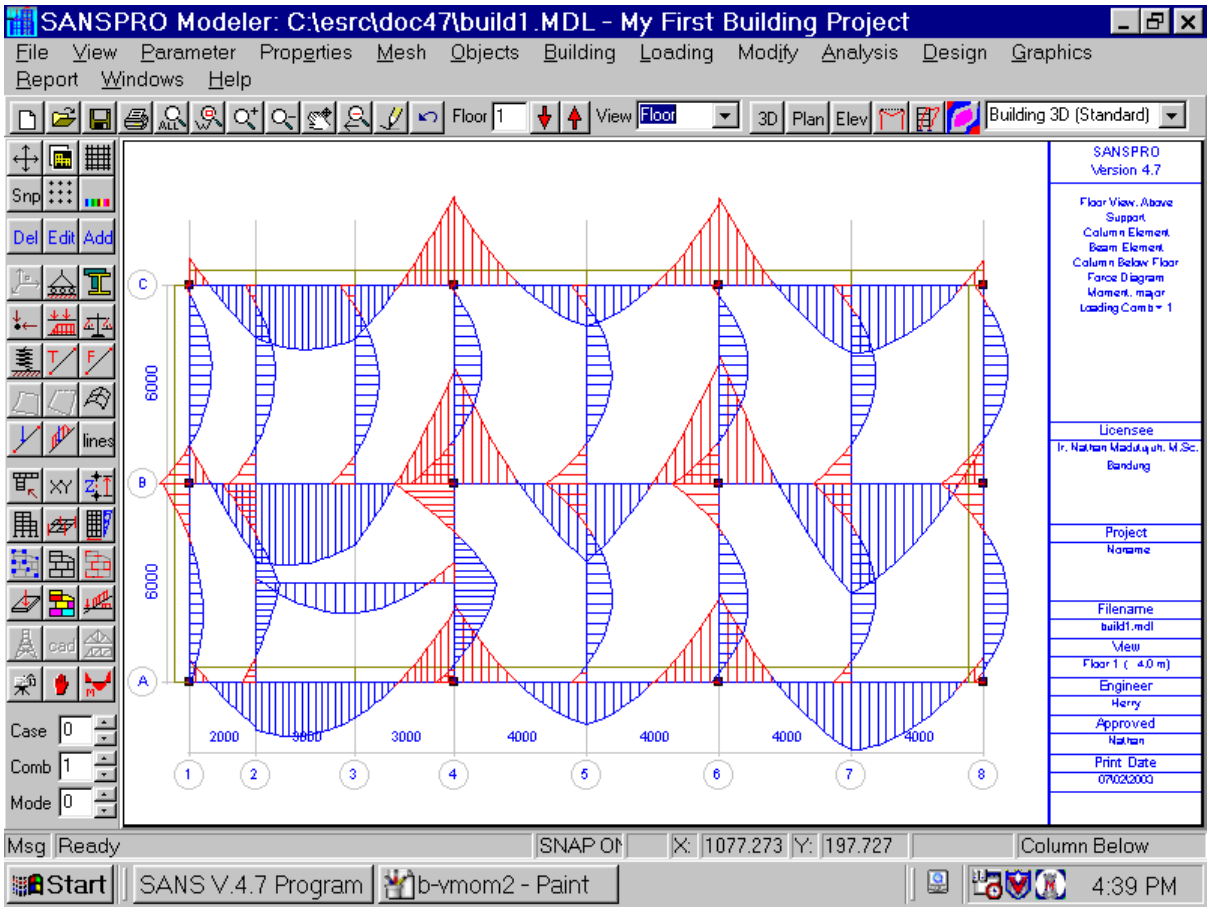
18. Output View

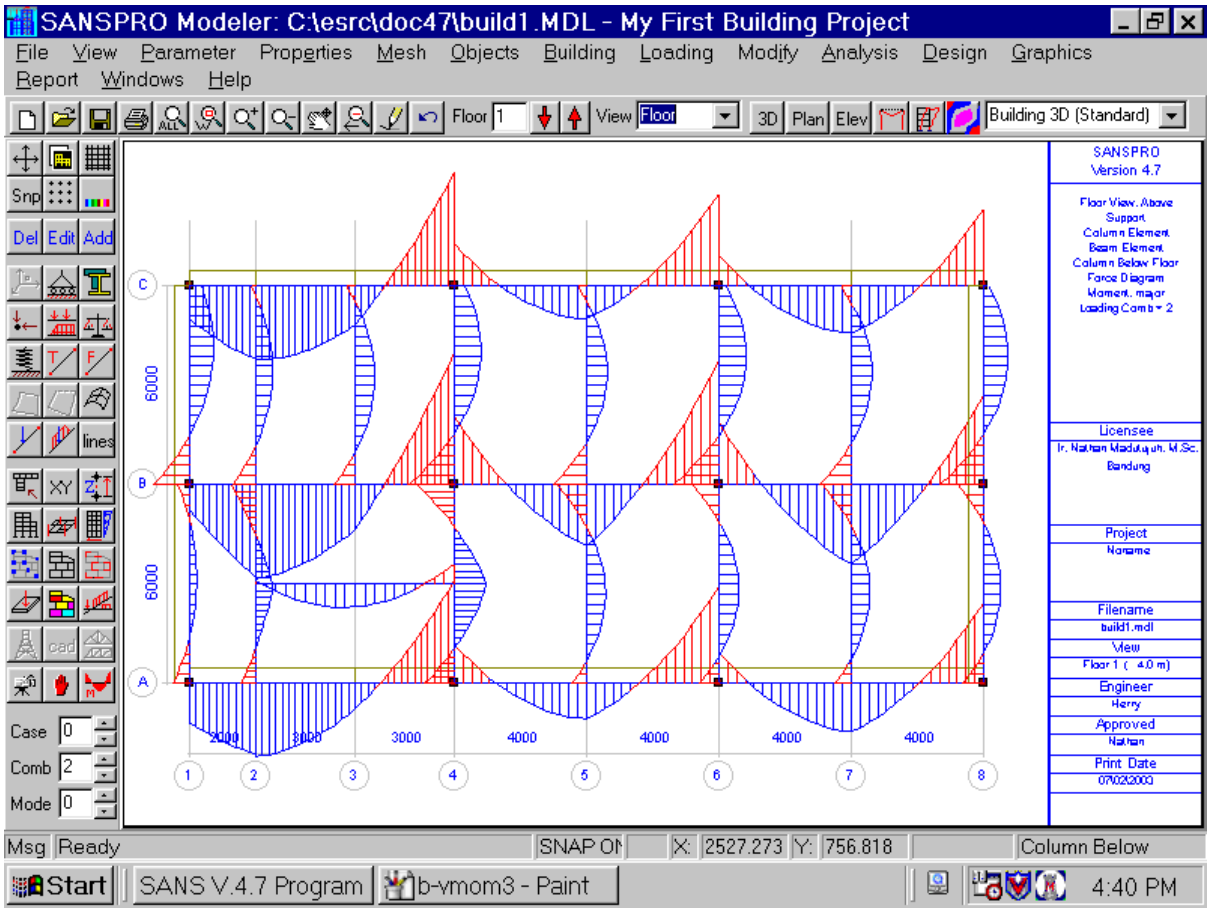
- After running analysis we can see deformations, element forces, support reactions, moment diagram, and animation of the deformation.
- By default, if no load combination selected, ldcomb = 0, then the envelope (maxima/minima) of the output will be displayed. In case of the moment diagram, two diagram will be drawn represent the range of the force values.
- If any load combination is selected, only the forces or diagram related to the diagram is shown.

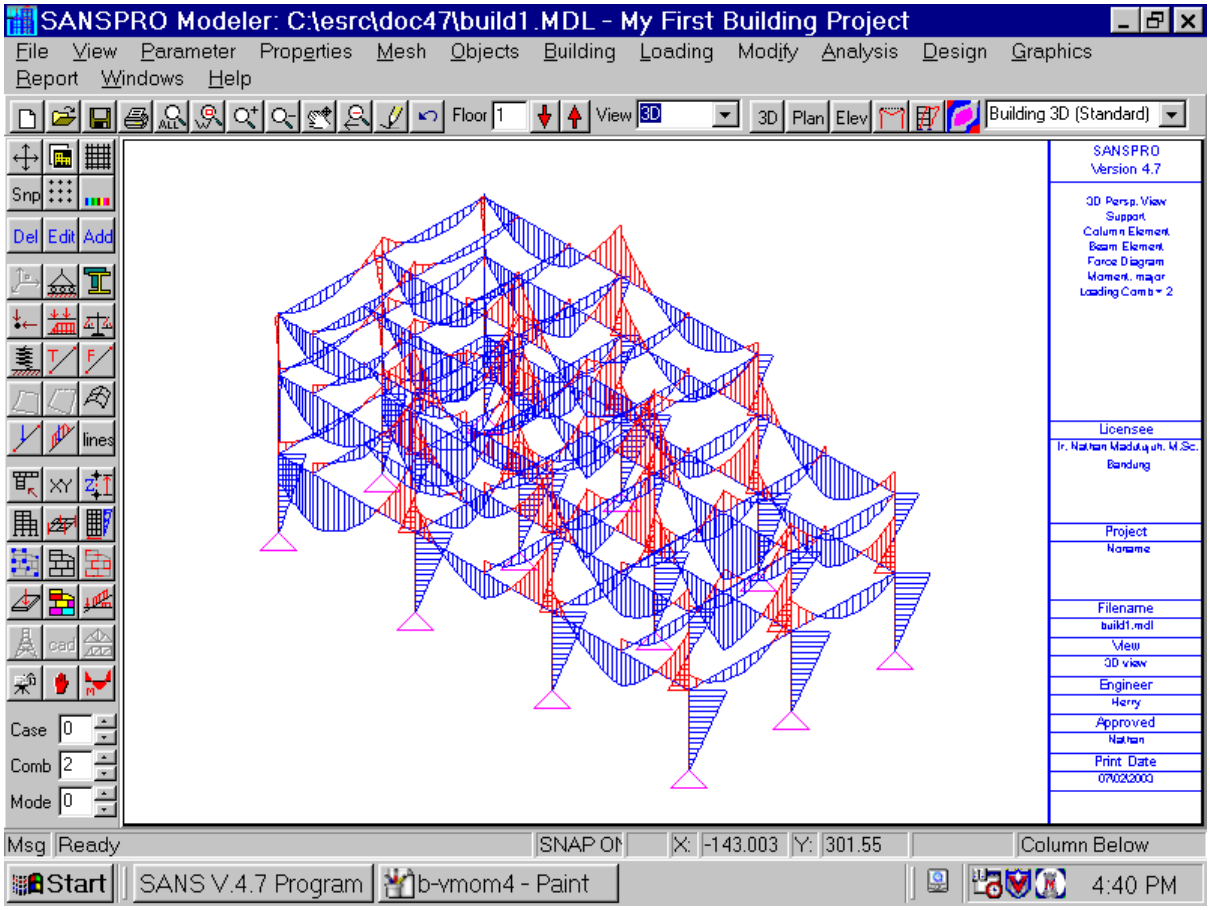
View Moment Diagram




SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
SANSPRO Version 4.7
Floor View, Above
Support
Column Element
Beam Element
Column Below Floor
Force Diagram
Moment, major
Loading Comb = ALL
Licensee
Ir. Nathan Maduluh, M.Sc.
Bandung
Project
Noname
Filename
build1.mdl
View
Floor 1 (4.0 m)
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Case 0
Comb 0
Mode 0
Msg Ready
SNAP Off
X: 1163.636 Y: -47.727
Column Below
Start
SANS V.4.7 Program
b-vmom1 - Paint
4:39 PM

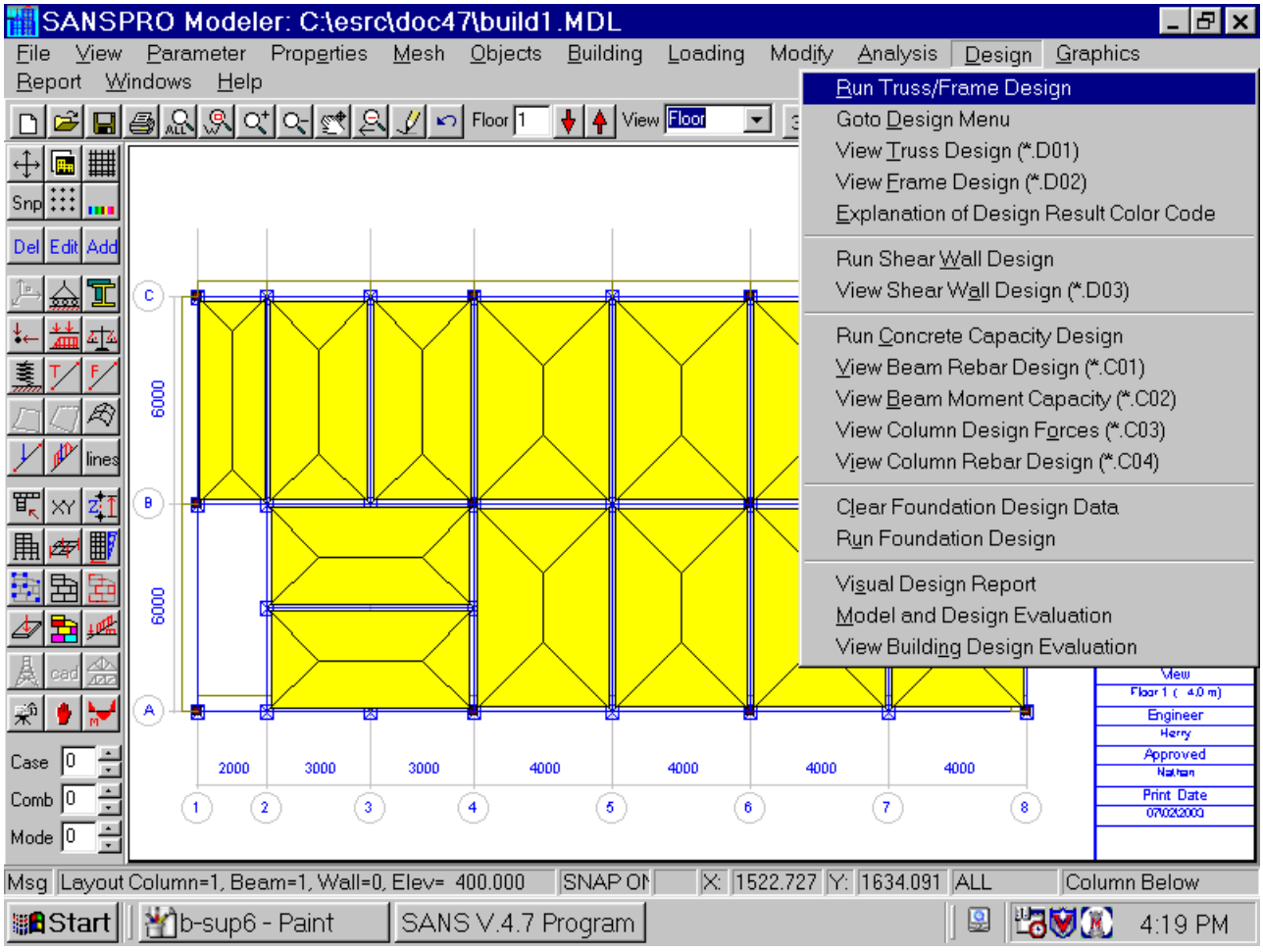

SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
SANSPRO Version 4.7
Floor View, Above
Support
Column Element
Beam Element
Column Below Floor
Force Diagram
Moment, major
Loading Comb = 1
Licensee
Ir. Nathan Maduluh, M.Sc.
Bandung
Project
Noname
Filename
build1.mdl
View
Floor 1 (4.0 m)
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Case 0
Comb 1
Mode 0
Msg Ready
SNAP Off
X: 1077.273 Y: 197.727
Column Below
Start
SANS V.4.7 Program
b-vmom2 - Paint
4:39 PM

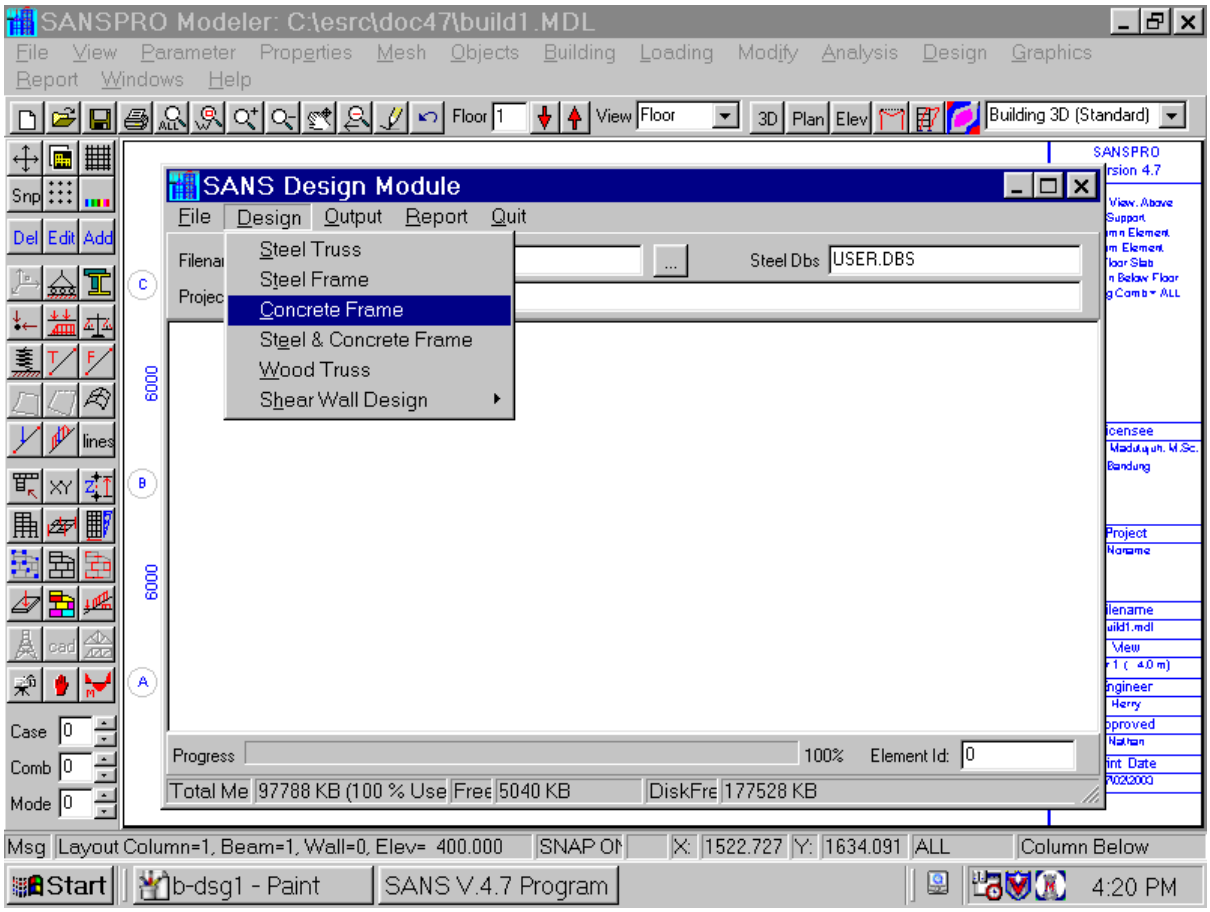

SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
SANSPRO Version 4.7
Floor View, Above
Support
Column Element
Beam Element
Column Below Floor
Force Diagram
Moment, major
Loading Comb = 2
Licensee
Ir. Nathan Madutujuh, M.Sc.
Bandung
Project
Noname
Filename
build1.mdl
View
Floor 1 (4.0 m)
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Case 0
Comb 2
Mode 0
6000
6000
3000
3000
4000
4000
4000
4000
Msg Ready
SNAP Off
X: 2527.273 Y: 756.818
Column Below
Start
SANS V.4.7 Program
b-vmom3 - Paint
4:40 PM

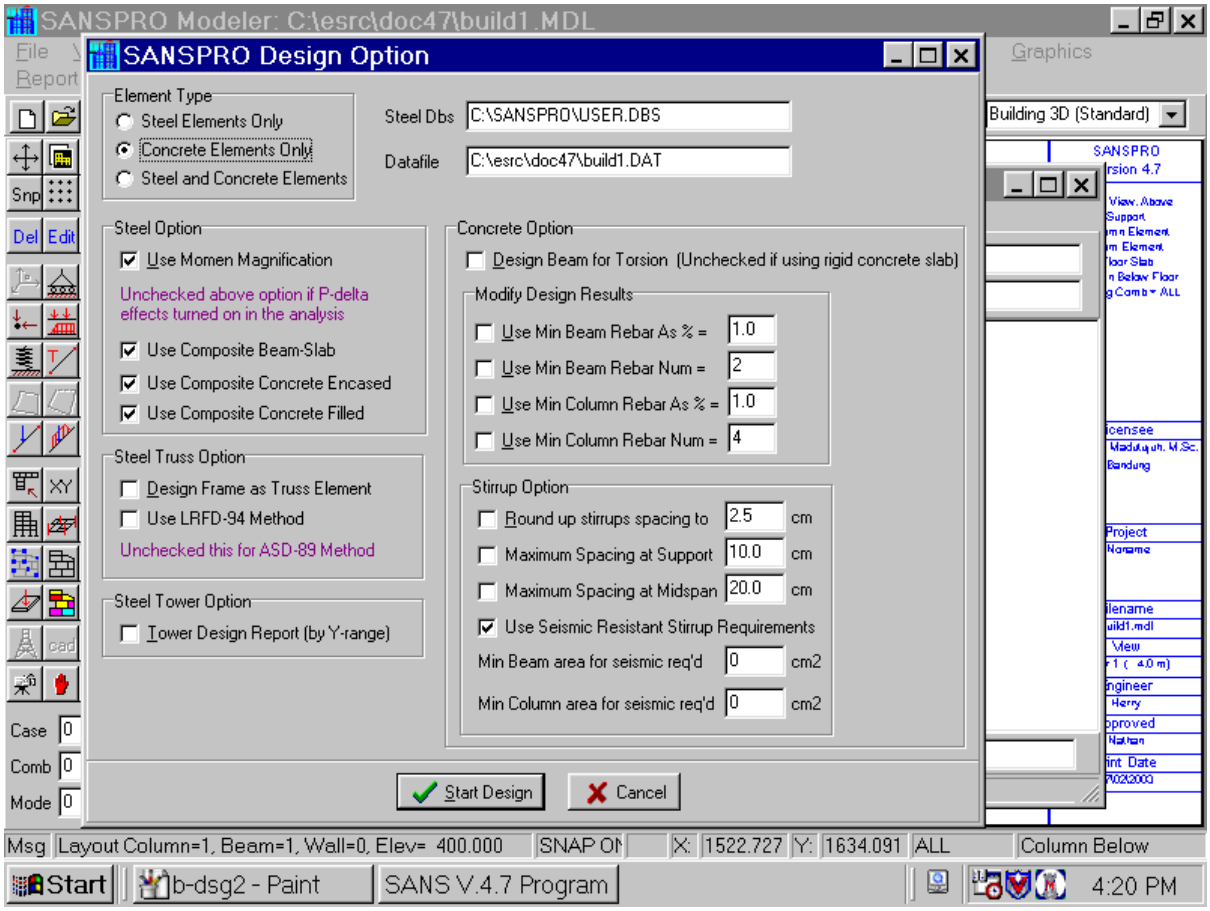

SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View 3D 3D Plan Elev Building 3D (Standard)
SANSPRO Version 4.7
3D Persp. View
Support
Column Element
Beam Element
Force Diagram
Moment, major
Loading Comb = 2
Licensee
Ir. Nathan Madutujuh, M.Sc.
Bandung
Project
Noname
Filename
build1.mdl
View
3D view
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Case 0
Comb 2
Mode 0
Msg Ready
SNAP Off
X: -143.003 Y: 301.55
Column Below
Start
SANS V.4.7 Program
b-vmom4 - Paint
4:40 PM

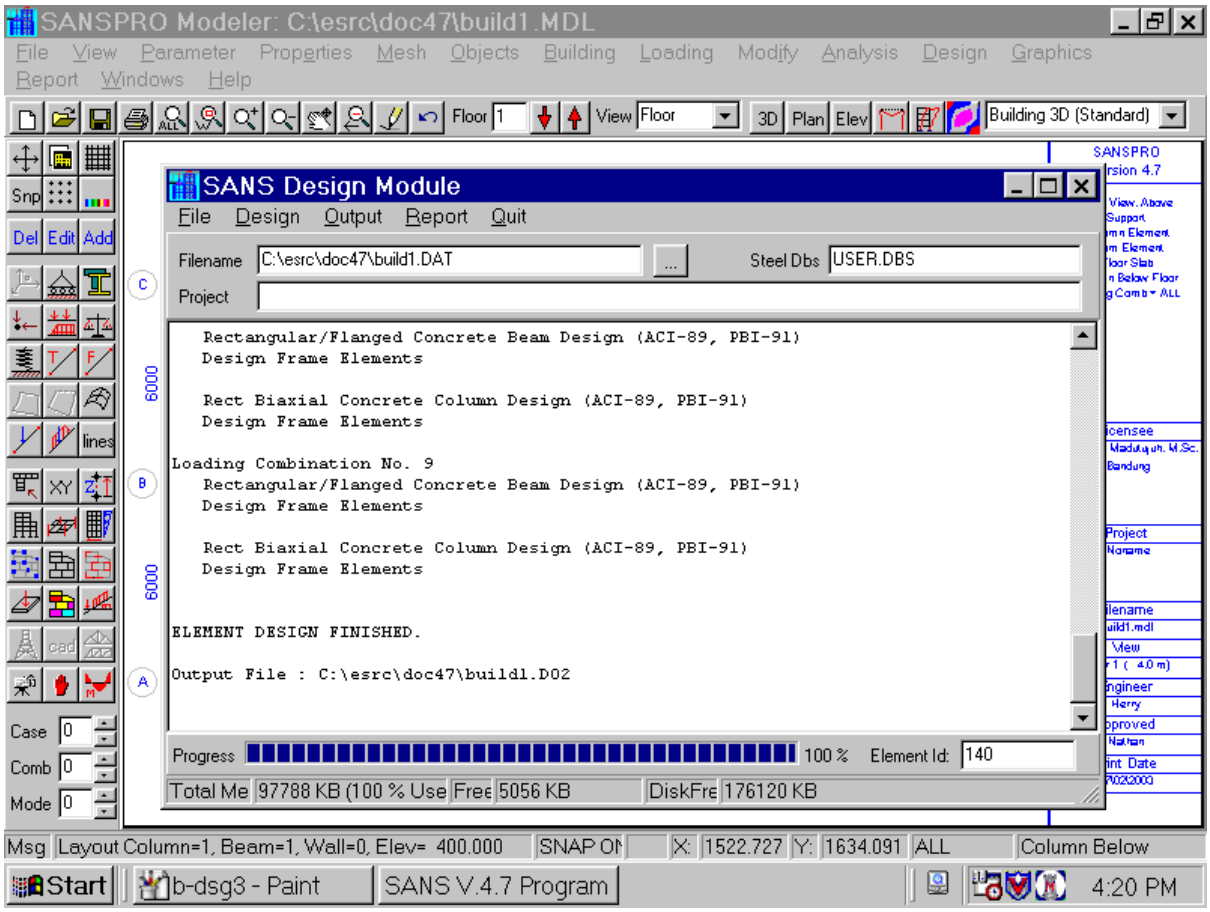
19. Element Design

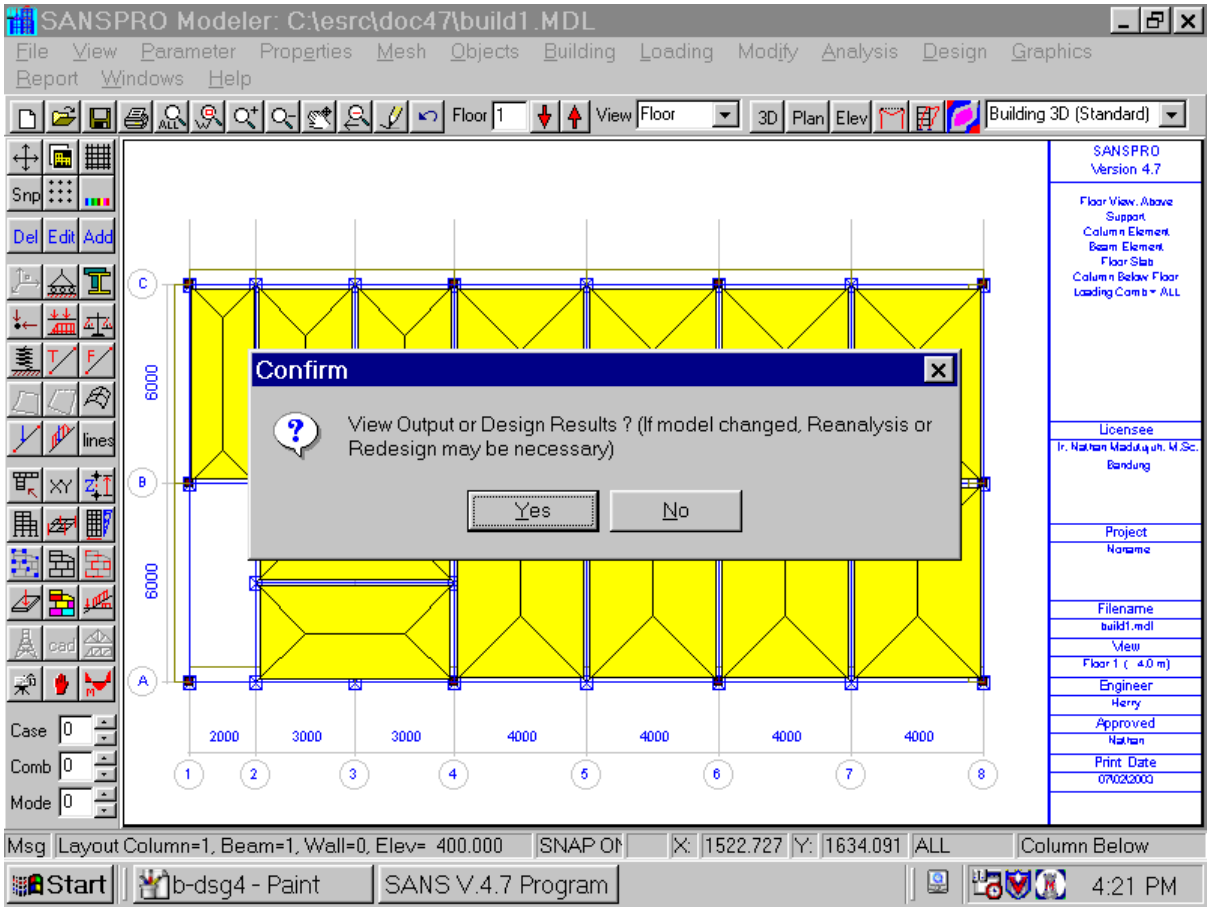
- After analysis, we can run element design.
- Click Design – Truss/Frame Design
- Click Start Design
- Check **Design Beam for Torsion** for precast slab system or beam at void area
- Check Use **Seismic Resistant Stirrup Requirements** for building in seismic area
- After final design, you can check the **Modify Design Results** and **Round Up** option to refine the design result. This option should be checked only after you get a good design result.
- SANSPRO will design all elements of the building for all load combinations at stored at file BUILD1.DSB (binary format) and BUILD1.D02 (text format)
- Answer **View Design Results?** with Yes to activate visual design result view mode.




SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
SANS Design Module
File Design Output Report Quit
Steel Truss
Steel Frame
Concrete Frame
Steel & Concrete Frame
Wood Truss
Shear Wall Design
Steel Dbs USER.DBS
Progress 100% Element Id: 0
Msg Layout Column=1, Beam=1, Wall=0, Elev= 400.000 SNAP On X: 1522.727 Y: 1634.091 ALL Column Below
Start b-dsg1 - Paint SANS V.4.7 Program 4:20 PM

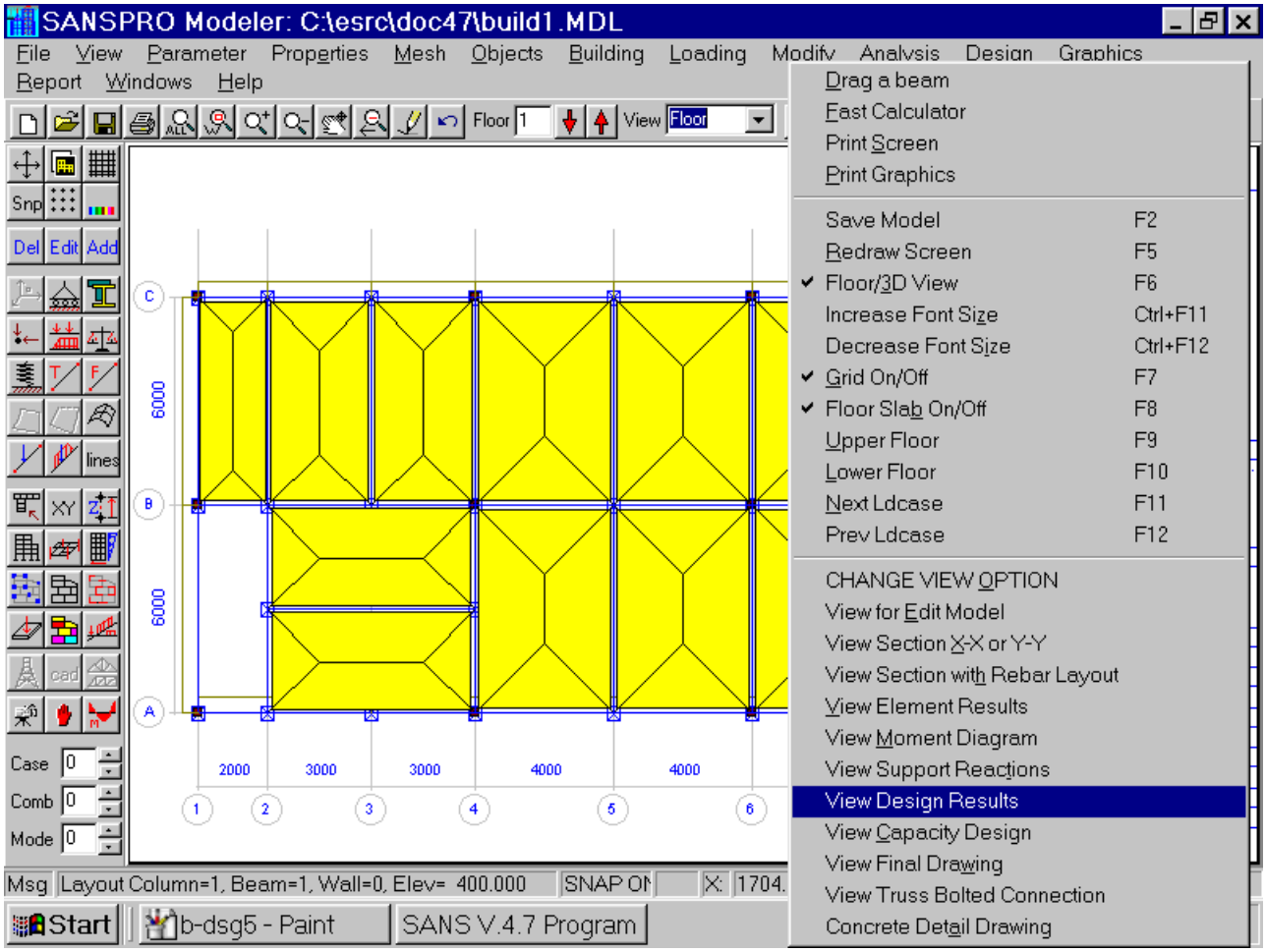

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
SANSPRO Design Option
Element Type
Steel Elements Only
Concrete Elements Only
Steel and Concrete Elements
Steel Dbs C:\SANSPRO\USER.DBS
Datafile C:\esrc\doc47\build1.DAT
Steel Option
Use Momen Magnification
Unchecked above option if P-delta effects turned on in the analysis
Use Composite Beam-Slab
Use Composite Concrete Encased
Use Composite Concrete Filled
Steel Truss Option
Design Frame as Truss Element
Use LRFD-94 Method
Unchecked this for ASD-89 Method
Steel Tower Option
Tower Design Report (by Y-range)
Concrete Option
Design Beam for Torsion (Unchecked if using rigid concrete slab)
Modify Design Results
Use Min Beam Rebar As % = 1.0
Use Min Beam Rebar Num = 2
Use Min Column Rebar As % = 1.0
Use Min Column Rebar Num = 4
Stirrup Option
Round up stirrups spacing to 2.5 cm
Maximum Spacing at Support 10.0 cm
Maximum Spacing at Midspan 20.0 cm
Use Seismic Resistant Stirrup Requirements
Min Beam area for seismic req'd 0 cm2
Min Column area for seismic req'd 0 cm2
Start Design
Cancel
Msg Layout Column=1, Beam=1, Wall=0, Elev= 400.000 SNAP On X: 1522.727 Y: 1634.091 ALL Column Below
Start b-dsg2 - Paint SANS V.4.7 Program 4:20 PM

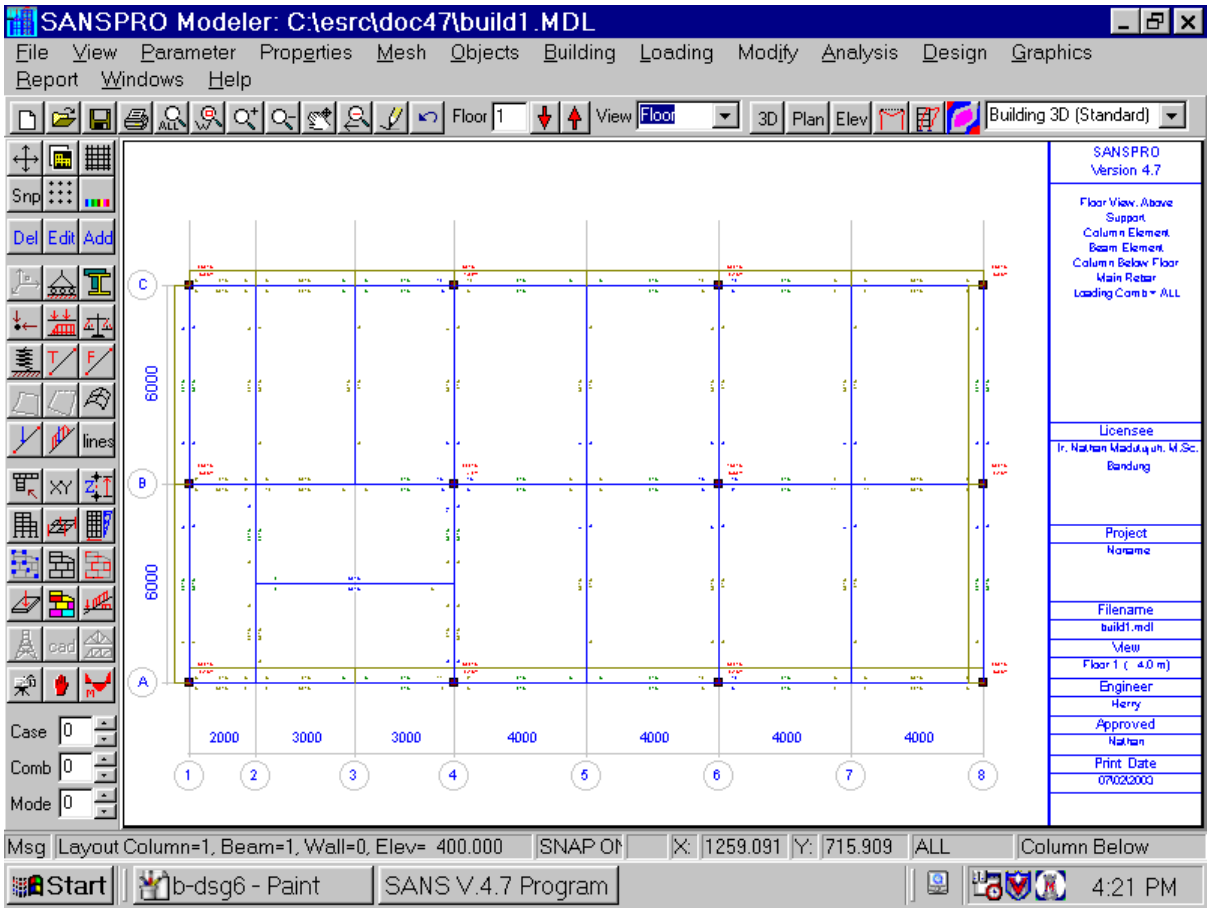

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
SANS Design Module
File Design Output Report Quit
Filename C:\esrc\doc47\build1.DAT
Steel Dbs USER.DBS
Project
Rectangular/Flanged Concrete Beam Design (ACI-89, PBI-91)
Design Frame Elements
Rect Biaxial Concrete Column Design (ACI-89, PBI-91)
Design Frame Elements
Loading Combination No. 9
Rectangular/Flanged Concrete Beam Design (ACI-89, PBI-91)
Design Frame Elements
Rect Biaxial Concrete Column Design (ACI-89, PBI-91)
Design Frame Elements
ELEMENT DESIGN FINISHED.
Output File : C:\esrc\doc47\build1.D02
Progress 100 % Element Id: 140
Total Me 97788 KB (100 % Use Free 5056 KB DiskFre 176120 KB
Msg Layout Column=1, Beam=1, Wall=0, Elev= 400.000 SNAP Of X: 1522.727 Y: 1634.091 ALL Column Below
Start b-dsg3 - Paint SANS V.4.7 Program 4:20 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Confirm
View Output or Design Results ? (If model changed, Reanalysis or Redesign may be necessary)
Yes
No
2000 3000 3000 4000 4000 4000 4000
1 2 3 4 5 6 7 8
Msg Layout Column=1, Beam=1, Wall=0, Elev= 400.000 SNAP Of X: 1522.727 Y: 1634.091 ALL Column Below
Start b-dsg4 - Paint SANS V.4.7 Program 4:21 PM

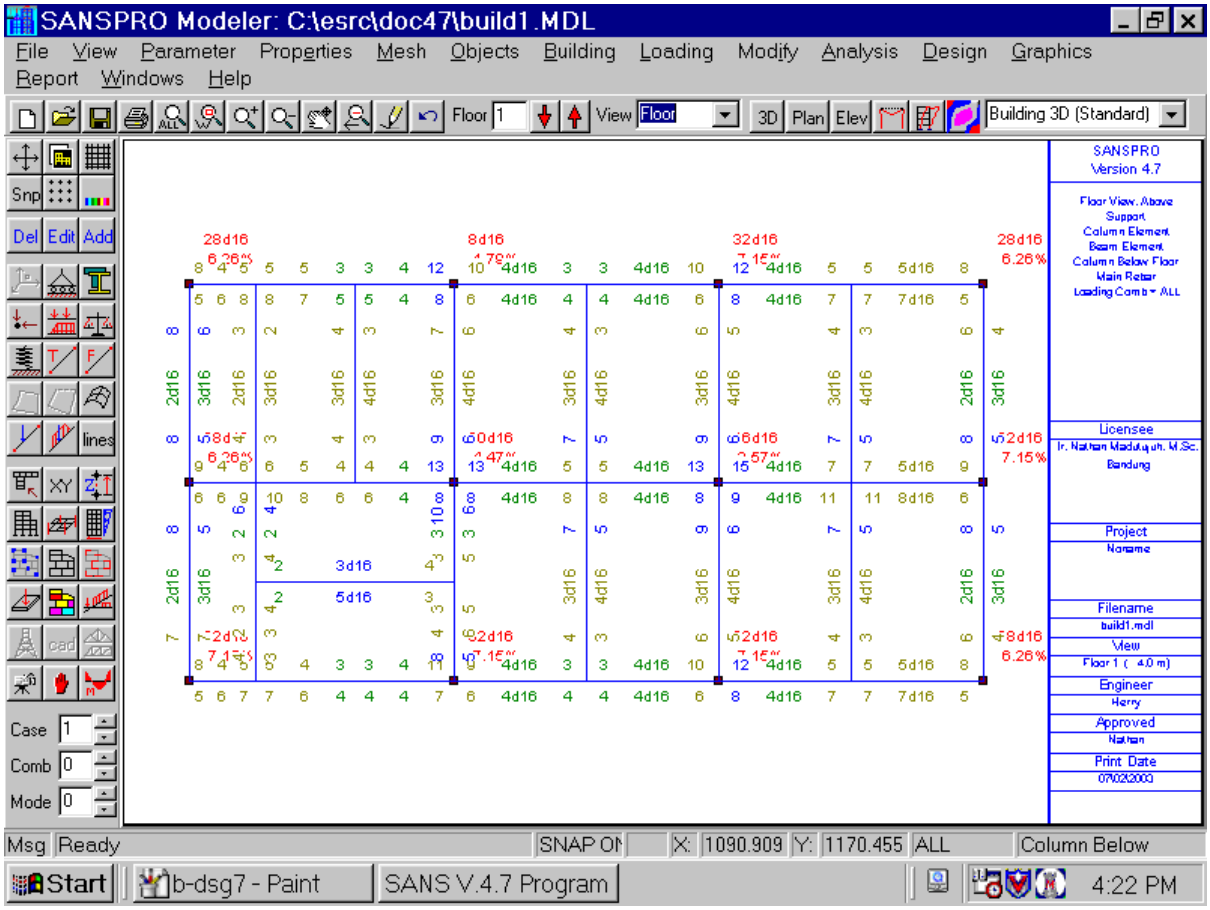
20. Design View

- After design, we can view design results interactively.
- **Right-Click** and **Select View Design Results**
- Beam rebar and column rebar will be displayed in color code
- SANSPRO uses color code to speed up visual design checking process

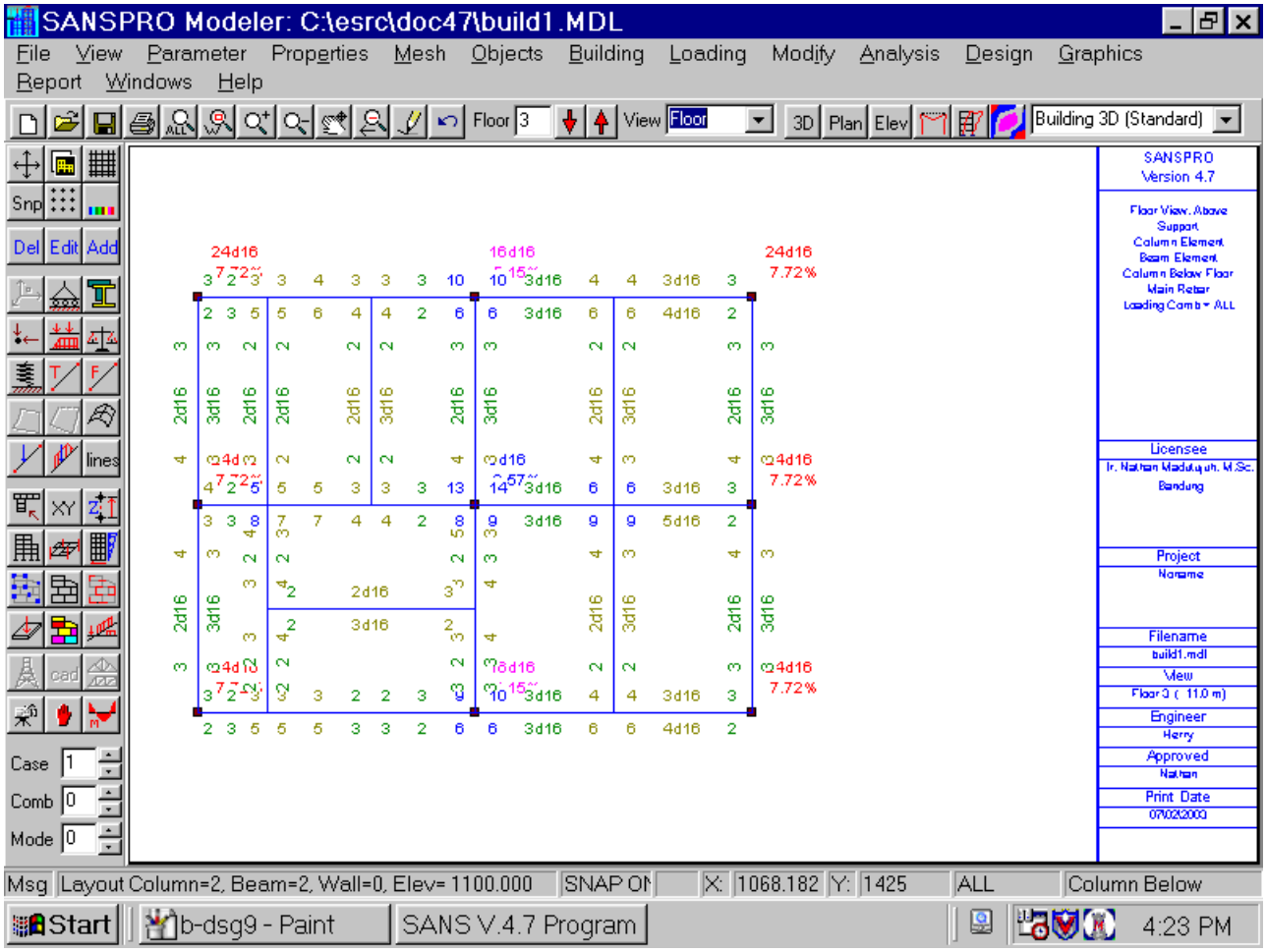




- Use Ctrl-F11 to increase font size and Ctrl-F12 to decrease font size



- To see rebar layout at next floor, click **up-down** button



21. Element Design View

- To see element-by-element output and design results, do:
- Right-Click, Select View Element Results
- Select any element (beam, column) type icon
- Select any element (beam, column)
- A dialog box will be displayed containing element results in three pages:
    - Element forces for selected load combination, maximum, minimum
    - Design results for selected load combination, maximum, minimum
    - Moment Diagram for selected load combination, maximum, minimum

22. Design Color Code

To make design result checking easily, SANSPRO provides a scheme called Design Color Code. The color of the design output is related to its design quality. Currently the color table is fixed, but newer version of SANSPRO will allow user to change the color code to customize the design result for local use.

Your target color depends on your material cost and your beam depth requirement. For guidance, good steel structures will give **Blue** color code, while good concrete structure will give **Green, Olive, Blue** color code.

Steel Strees Ratio Color Code:

Stress Ratio	Color Code	Description
<= 0.6	Green	OK, Section too big
0.6 >= fr < 1.0	Blue	OK, Good
1.0 >= fr < 1.3	Magenta	Not OK, but OK for temporer load
>= 1.3	Red	Not OK

Concrete Rebar Color Code:

Beam Rebar	Color Code	Description
0.0-0.5%	Blue	OK, Section maybe too big
0.5-1.0%	Olive	OK, Section maybe too big
1.0-2.0%	Blue	OK, Rebar maybe too densed
2.0-3.0%	Purple	OK, Rebar too dense
>= 3.0%	Red	Not OK, Rebar too dense

Column Rebar	Color Code	Description
0.0-1.5%	Green	OK, Section maybe too big
1.5-3.0%	Blue	OK
3.0-4.0%	Maroon	OK, Rebar maybe too densed
4.0-6.0%	Purple	OK, Rebar too dense
>= 6.0%	Red	Not OK, Rebar too dense

You can also limit beam rebar percentage range by changing the View Option.
This option is useful to find beams with certain rebar percentage among a large number of beams.

23. Invalidate Design

If you are in Design View mode, and you press a menu or icon that may affect the model, SANSPRO will automatically ask you if you want to invalidate the design results.

If you answer with Yes then you will not be able to view the design results anymore, you must run the design process again.

If you answer with No then you will be asked if you want to reset the flag so you will not be asked again until another model changes.

24. Revise Design (Design Optimization)

If analysis failed or some element design failed, you can change the element by :

- Changing the Section table or Design table
  (This change will affect the whole elements using the same elset)
- Changing element elset index
  (This change will only affect the target element)

After changing the model, then Save (F2), Export (F4), Run and Design again.
Then21 visually check the design results again.
Repeat the whole process until we get a good and economical design results.

25. Printing Report

Visual Report can be printed by selecting **File-Print Graphics** or **Right-Click** and select **Print Graphics**.

To print output file (text format), click **Analysis-View Output File-Print**, etc.
To print design output file (text format), click **Design-View Design Output-Print**

SANSPRO can print drawings to A4 to A1 size printers.
Drawing can also be first exported to Autocad for further processing before printing using Autocad.

26. Foundation Design

SANSPRO provides easy and fast way to do foundation design. Just press a button and in seconds you will get a detailed report of foundation design of all the supports.

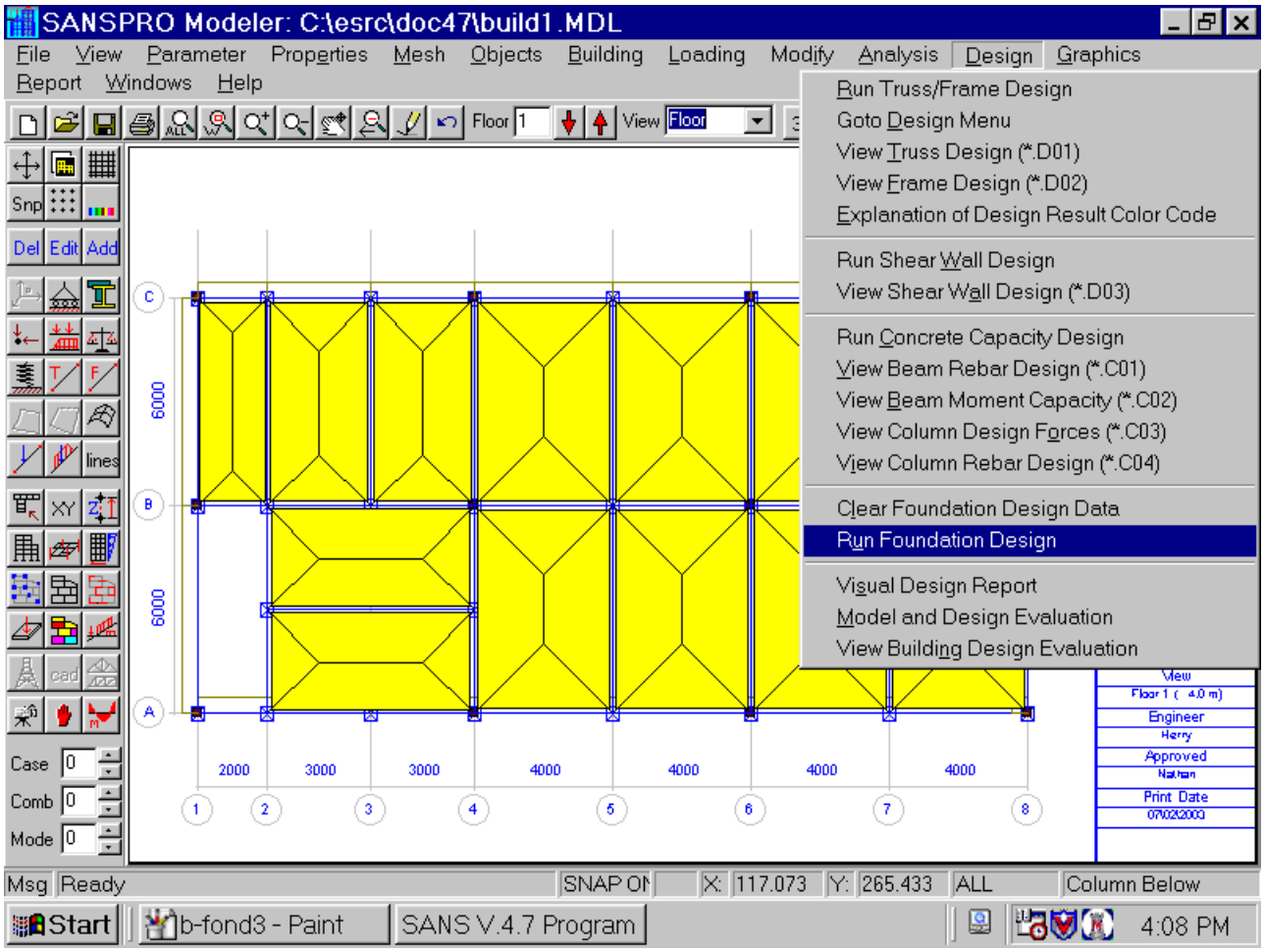
SANSPRO foundation design can deal with several types of foundation or pile. It will search for pile type with minimum cost.

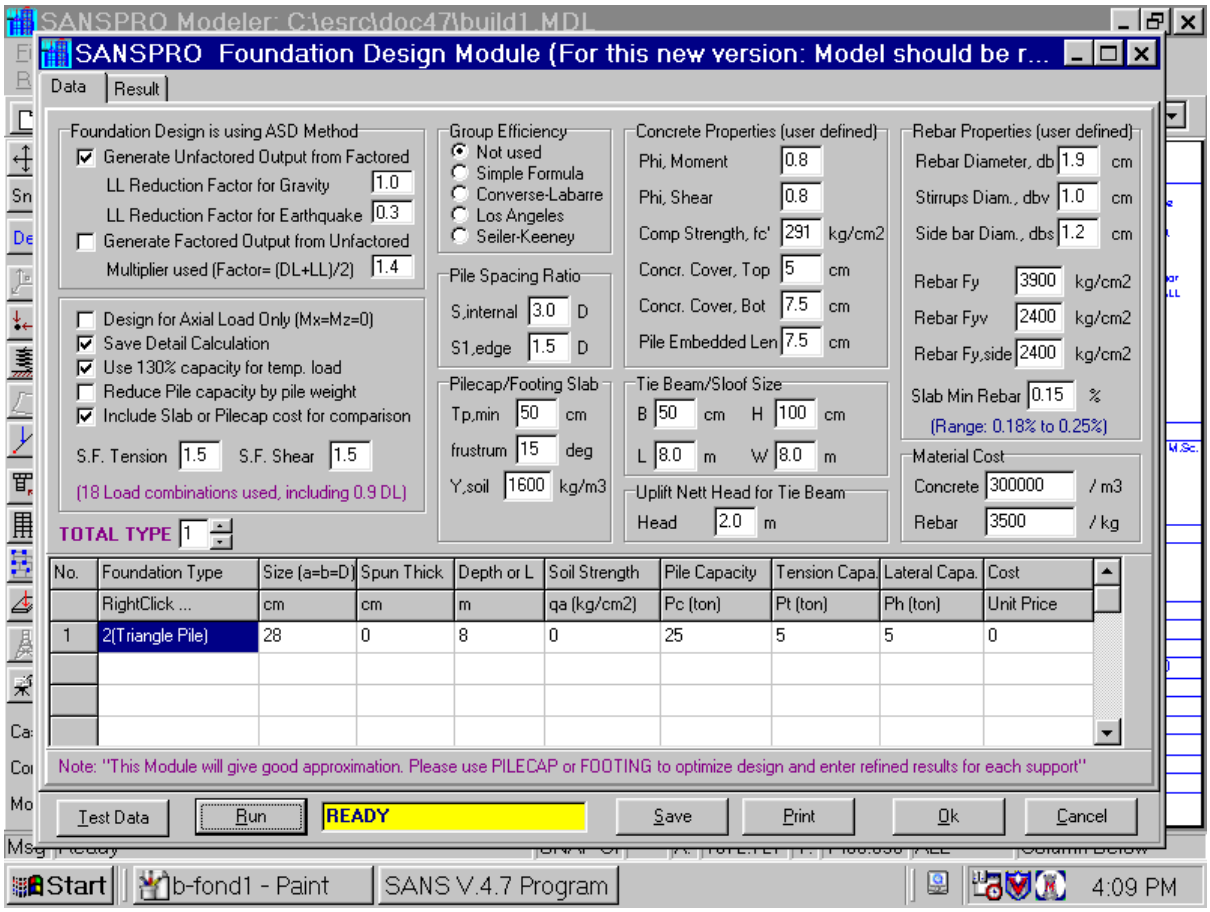
Because foundation design needs factored load for concrete design and unfactored load for pile design, SANSPRO is designed to be able to retrieve the factored or unfactored reaction forces automatically from your model with factored load.

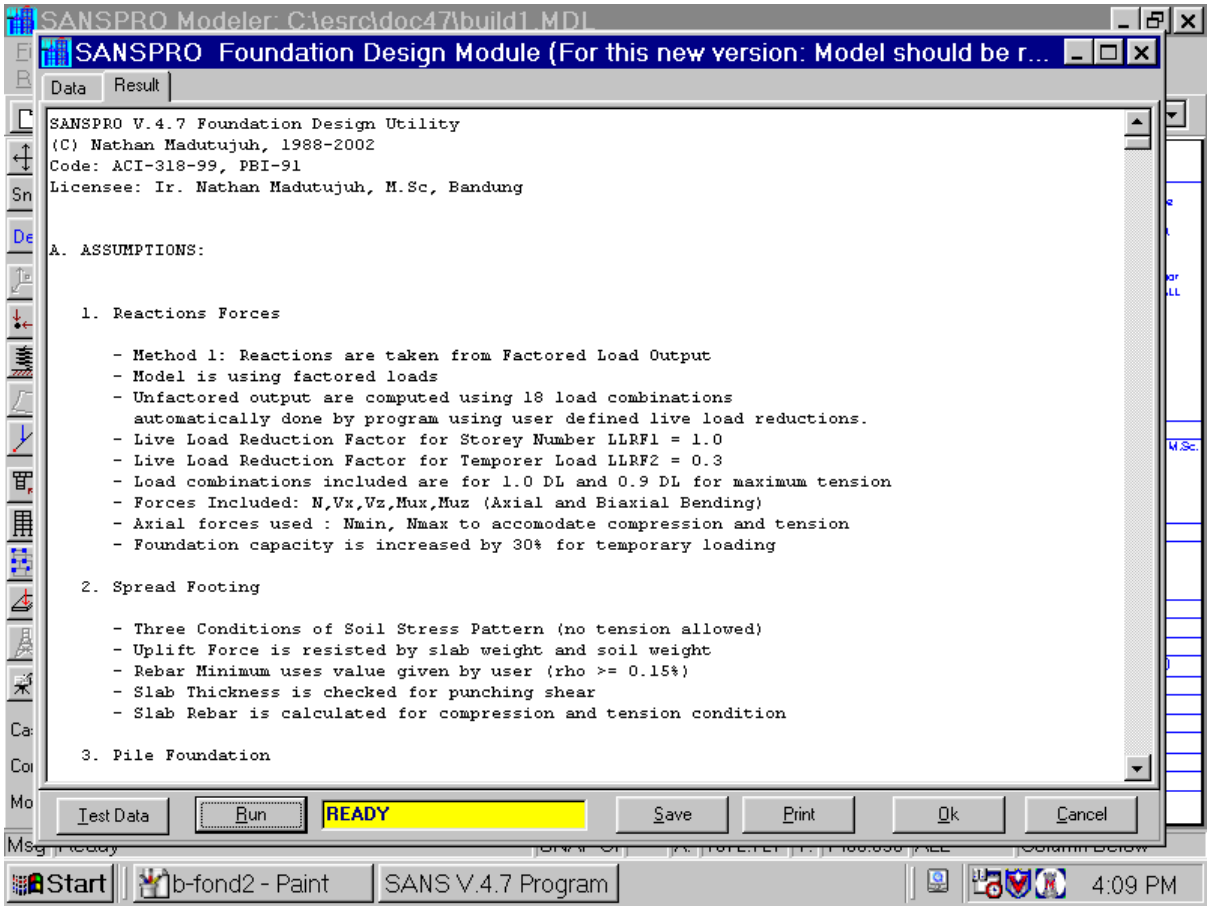
What SANSPRO do for foundation design are:

- Pile and footing design for axial, shear and biaxial bending moment
- Pilecap design for punching shear and bending moment
- Simplified tie beam design
- Volume and cost of pilecap and pile
- Select pile type with minimum cost

Detail design report can be saved to a file or print directly.
All assumptions used are attached to the front of the report.




SANSPRO Modeler: C:\esrc\doc47\build1.MDL
SANSPRO Foundation Design Module (For this new version: Model should be r...
Data
Result
Foundation Design is using ASD Method
Generate Unfactored Output from Factored
LL Reduction Factor for Gravity 1.0
LL Reduction Factor for Earthquake 0.3
Generate Factored Output from Unfactored
Multiplier used (Factor= (DL+LL)/2) 1.4
Design for Axial Load Only (Mx=Mz=0)
Save Detail Calculation
Use 130% capacity for temp. load
Reduce Pile capacity by pile weight
Include Slab or Pilecap cost for comparison
S.F. Tension 1.5 S.F. Shear 1.5
(18 Load combinations used, including 0.9 DL)
TOTAL TYPE 1
Group Efficiency
Not used
Simple Formula
Converse-Labarre
Los Angeles
Seiler-Keeney
Pile Spacing Ratio
S,internal 3.0 D
S1,edge 1.5 D
Pilecap/Footing Slab
Tp,min 50 cm
frustrum 15 deg
Y,soil 1600 kg/m3
Concrete Properties (user defined)
Phi, Moment 0.8
Phi, Shear 0.8
Comp Strength, fc' 291 kg/cm2
Concr. Cover, Top 5 cm
Concr. Cover, Bot 7.5 cm
Pile Embedded Len 7.5 cm
Tie Beam/Sloof Size
B 50 cm H 100 cm
L 8.0 m W 8.0 m
Uplift Nett Head for Tie Beam
Head 2.0 m
Rebar Properties (user defined)
Rebar Diameter, db 1.9 cm
Stirrups Diam., dbv 1.0 cm
Side bar Diam., dbs 1.2 cm
Rebar Fy 3900 kg/cm2
Rebar Fyv 2400 kg/cm2
Rebar Fy,side 2400 kg/cm2
Slab Min Rebar 0.15 %
(Range: 0.18% to 0.25%)
Material Cost
Concrete 300000 / m3
Rebar 3500 / kg
No. Foundation Type Size (a=b=D) Spun Thick Depth or L Soil Strength Pile Capacity Tension Capa. Lateral Capa. Cost
RightClick ... cm cm m qa (kg/cm2) Pc (ton) Pt (ton) Ph (ton) Unit Price
1 2(Triangle Pile) 28 0 8 0 25 5 5 0
Note: "This Module will give good approximation. Please use PILECAP or FOOTING to optimize design and enter refined results for each support"
Test Data
Run
READY
Save
Print
Ok
Cancel
Start
b-fond1 - Paint
SANS V.4.7 Program
4:09 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL
SANSPRO Foundation Design Module (For this new version: Model should be r...
Data
Result
SANSPRO V.4.7 Foundation Design Utility
(C) Nathan Madutujuh, 1988-2002
Code: ACI-318-99, PBI-91
Licensee: Ir. Nathan Madutujuh, M.Sc, Bandung
A. ASSUMPTIONS:
1. Reactions Forces
- Method 1: Reactions are taken from Factored Load Output
- Model is using factored loads
- Unfactored output are computed using 18 load combinations
automatically done by program using user defined live load reductions.
- Live Load Reduction Factor for Storey Number LLRF1 = 1.0
- Live Load Reduction Factor for Temporer Load LLRF2 = 0.3
- Load combinations included are for 1.0 DL and 0.9 DL for maximum tension
- 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 30% 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
Test Data
Run
READY
Save
Print
Ok
Cancel
Start
b-fond2 - Paint
SANS V.4.7 Program
4:09 PM

27. Compute Volume and Cost

SANSPRO provides easy and fast way to do volume and cost analysis. Just press a button and in seconds you will get a detailed report of volume and cost of all the elements in the building.

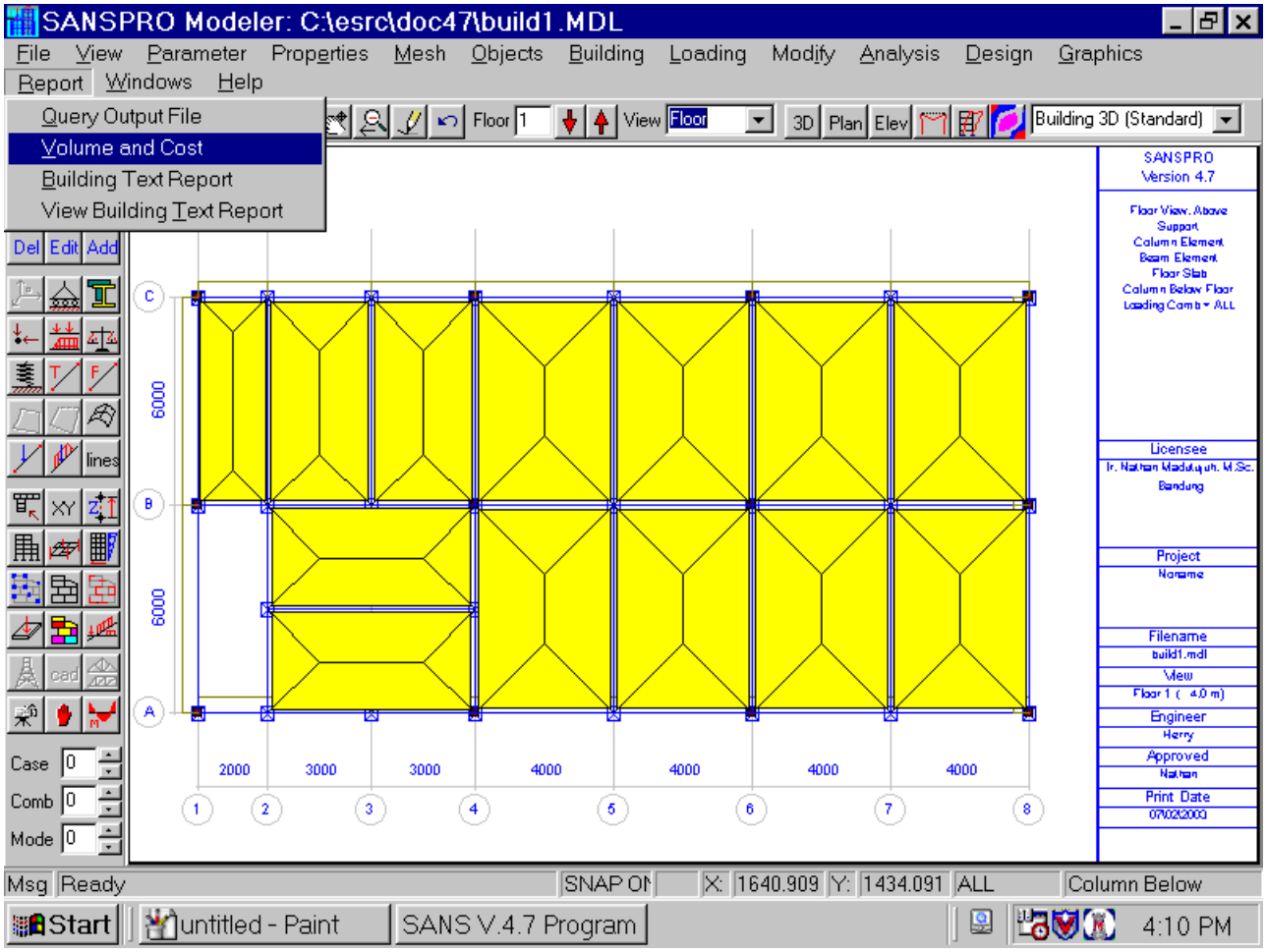
User only needs to provide unit price of concrete, rebar, formwork, and steel profile. SANSPRO will calculate the volume and cost automatically.

For concrete rebar, three options are available:

1. Use user defined rebar percentage (Select this for preliminary, no design results)
2. Use design results, different for each element
3. Use design results, maximum for same elset

Use the third method for better results.

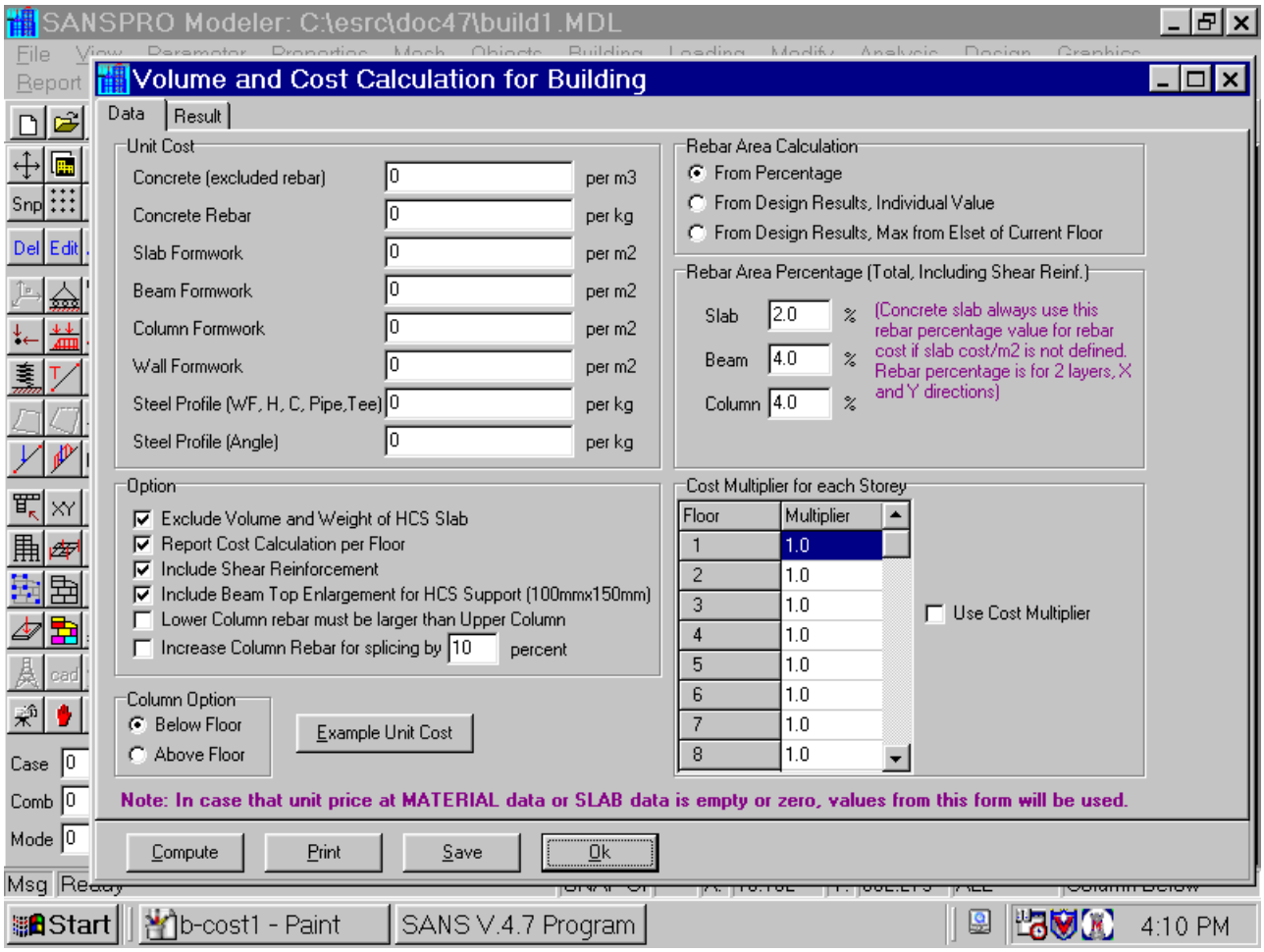
In any case, slab will be design only use the rebar percentage for slab.
If you want to overwrite this value, provide cost/m2 for all slab type at Floor Slab table.
This will give you a more accurate slab rebar weight and cost.
Exclude formwork for slab cost/m2 because SANSPRO can calculate the formwork cost automatically.

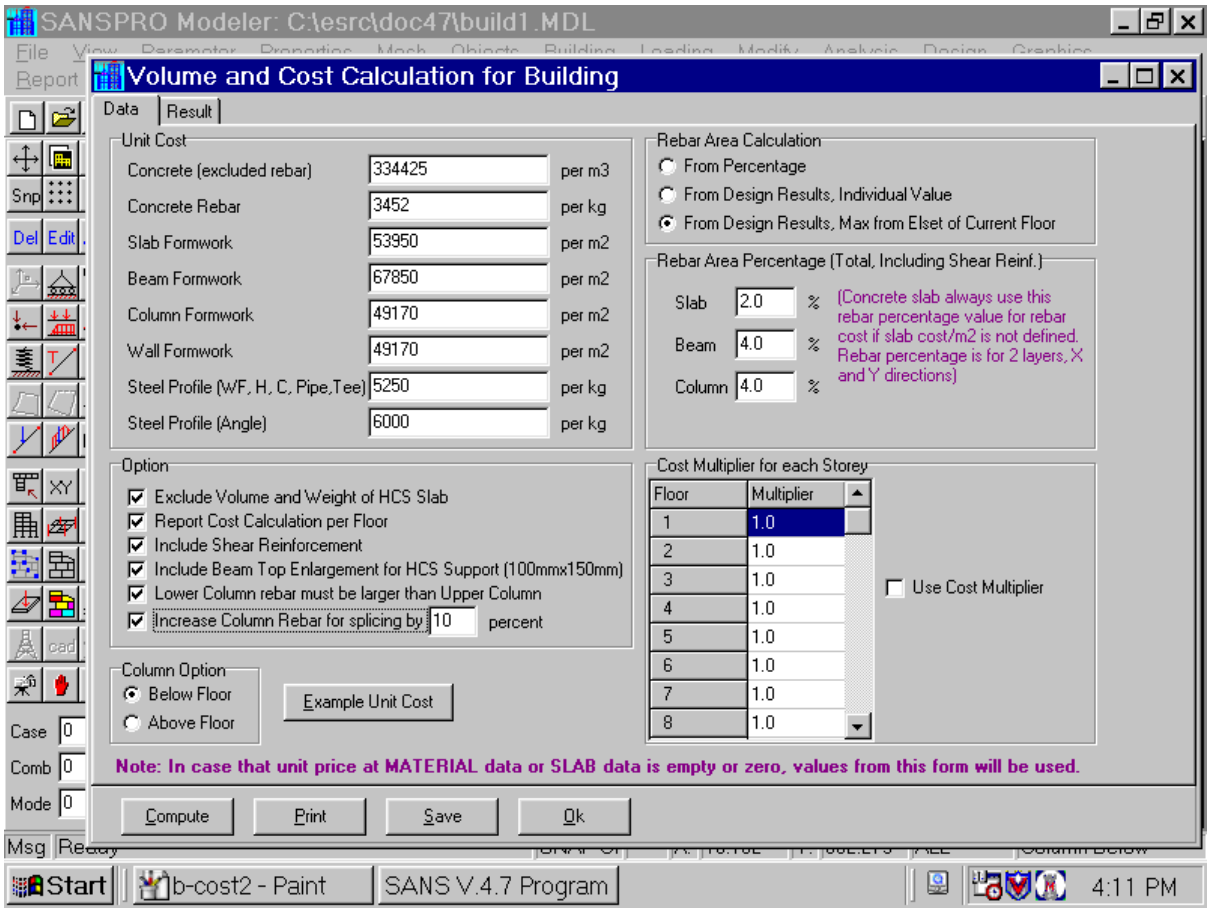


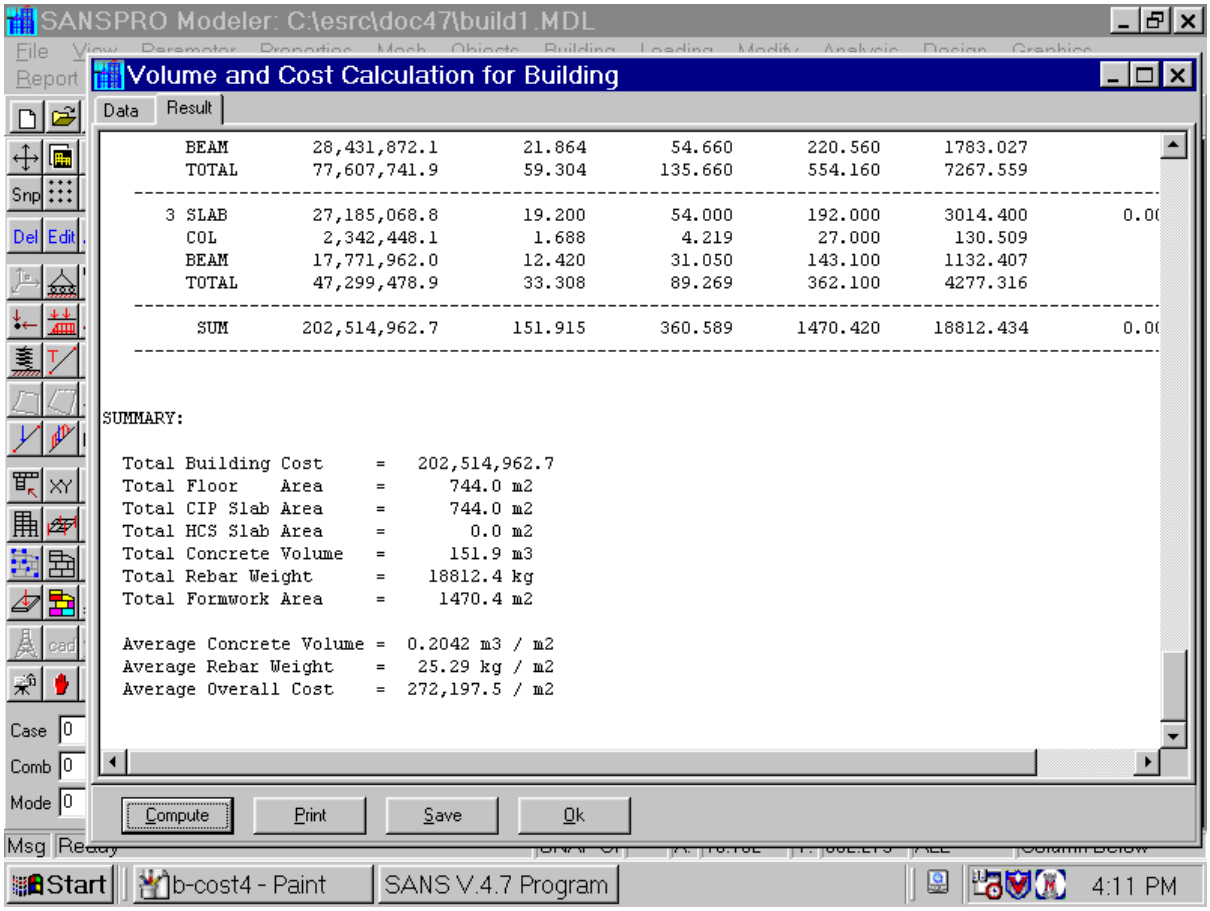
- Cost multiplier for each storey level may be used to increase material cost due to differences in handling cost for different floor level.

- Some additional options are also provided
- For cost example, click **Example Unit Cost**

- Cost reports are given per floor and per elset.
- Cost summary will be given including concrete density and average cost/m2.

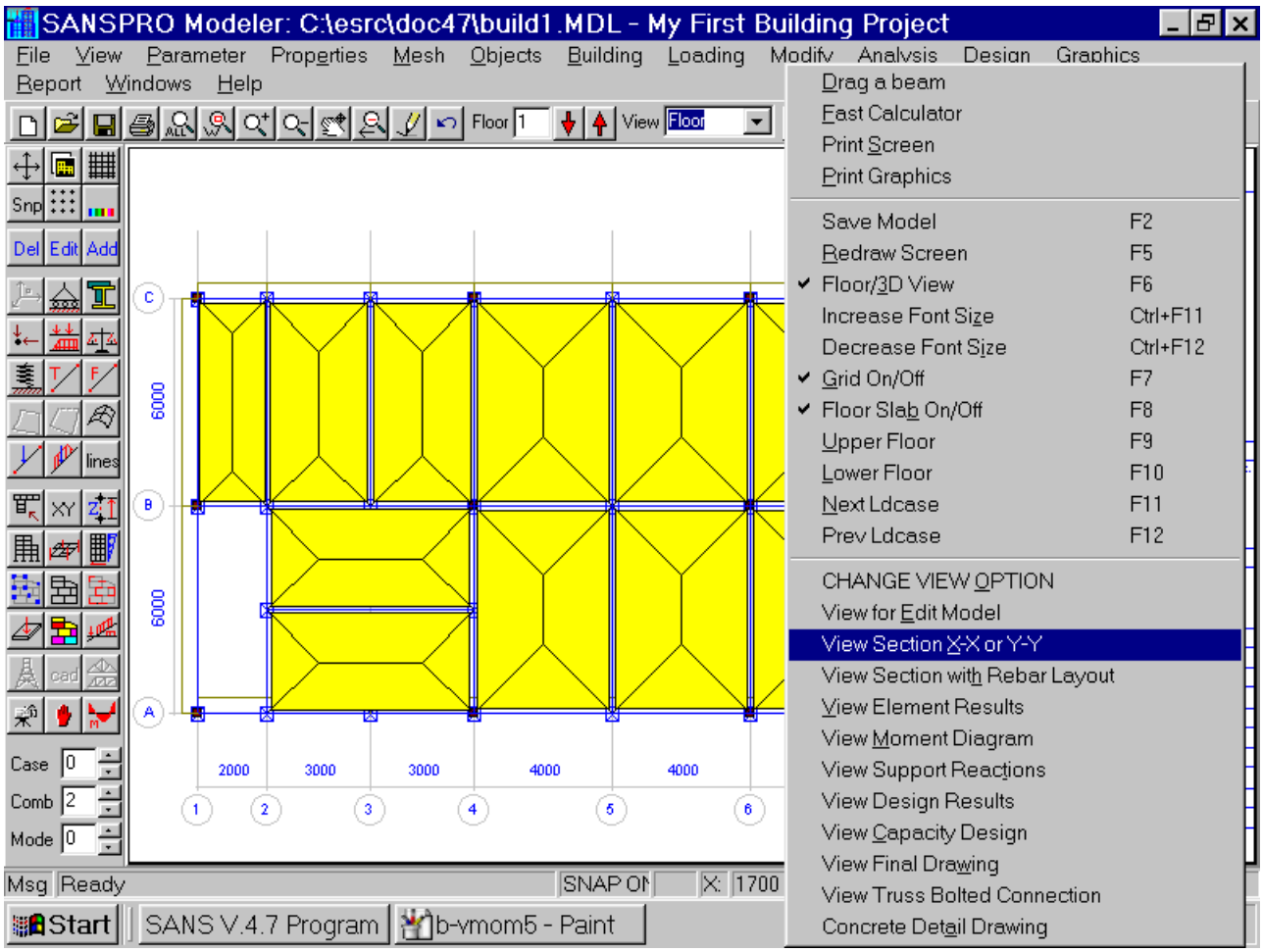


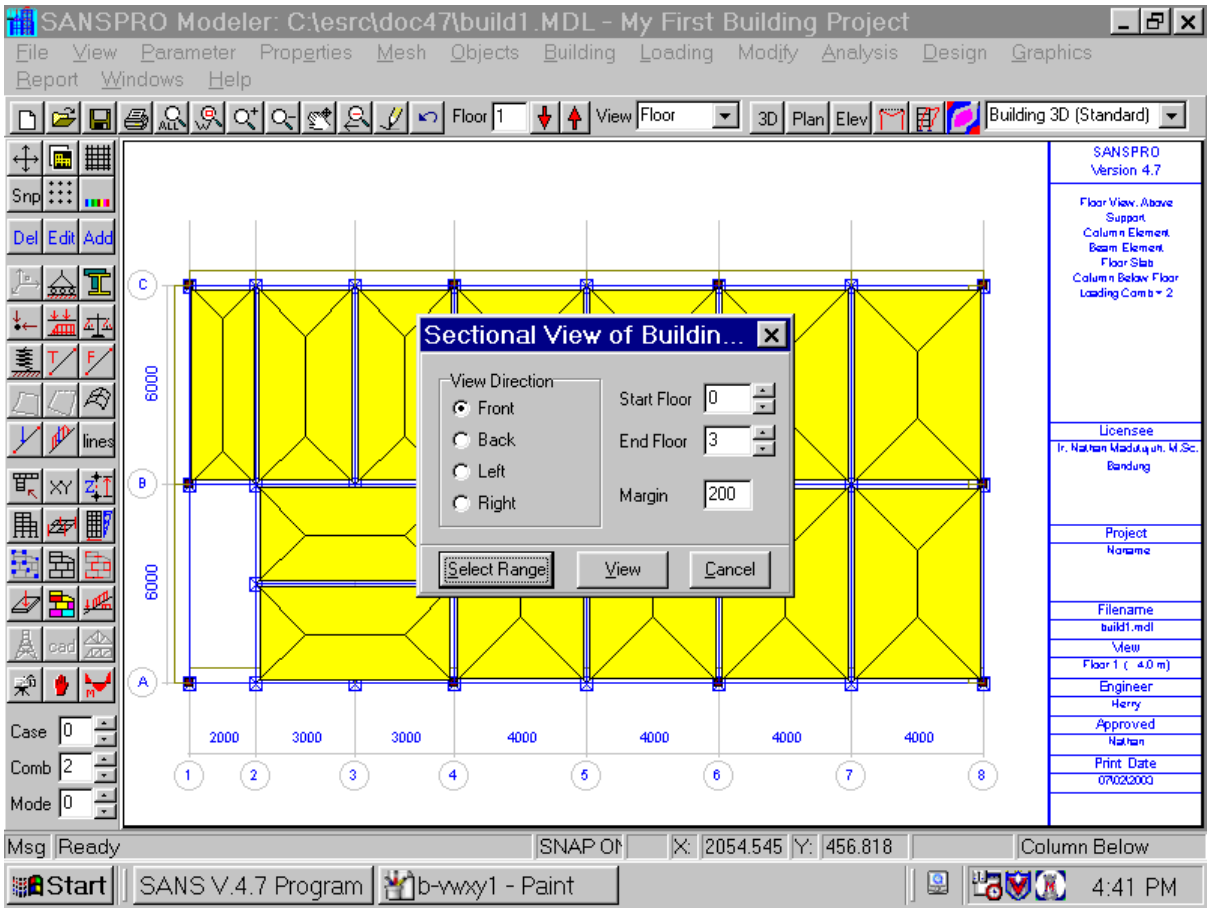

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
Volume and Cost Calculation for Building
Data
Result
Unit Cost
Concrete (excluded rebar) 334425 per m3
Concrete Rebar 3452 per kg
Slab Formwork 53950 per m2
Beam Formwork 67850 per m2
Column Formwork 49170 per m2
Wall Formwork 49170 per m2
Steel Profile (WF, H, C, Pipe,Tee) 5250 per kg
Steel Profile (Angle) 6000 per kg
Rebar Area Calculation
From Percentage
From Design Results, Individual Value
From Design Results, Max from Elset of Current Floor
Rebar Area Percentage (Total, Including Shear Reinf.)
Slab 2.0 %
Beam 4.0 %
Column 4.0 %
(Concrete slab always use this rebar percentage value for rebar cost if slab cost/m2 is not defined. Rebar percentage is for 2 layers, X and Y directions)
Option
Exclude Volume and Weight of HCS Slab
Report Cost Calculation per Floor
Include Shear Reinforcement
Include Beam Top Enlargement for HCS Support (100mmx150mm)
Lower Column rebar must be larger than Upper Column
Increase Column Rebar for splicing by 10 percent
Cost Multiplier for each Storey
Floor Multiplier
1 1.0
2 1.0
3 1.0
4 1.0
5 1.0
6 1.0
7 1.0
8 1.0
Use Cost Multiplier
Column Option
Below Floor
Above Floor
Example Unit Cost
Note: In case that unit price at MATERIAL data or SLAB data is empty or zero, values from this form will be used.
Compute
Print
Save
Ok
Case
Comb
Mode
Msg
Ready
Start
b-cost2 - Paint
SANS V.4.7 Program
4:11 PM

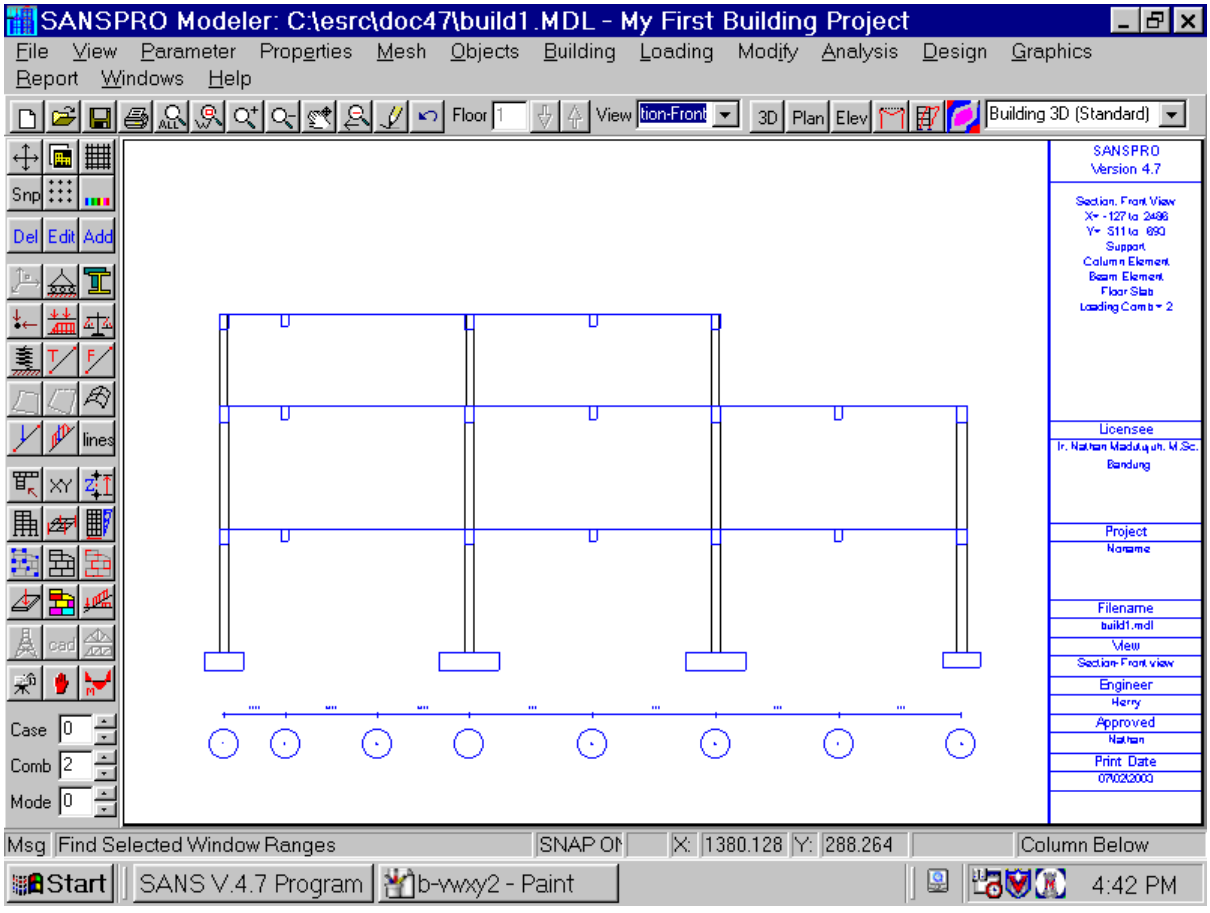

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
Volume and Cost Calculation for Building
Data
Result
BEAM 28,431,872.1 21.864 54.660 220.560 1783.027
TOTAL 77,607,741.9 59.304 135.660 554.160 7267.559
3 SLAB 27,185,068.8 19.200 54.000 192.000 3014.400 0.00
COL 2,342,448.1 1.688 4.219 27.000 130.509
BEAM 17,771,962.0 12.420 31.050 143.100 1132.407
TOTAL 47,299,478.9 33.308 89.269 362.100 4277.316
SUM 202,514,962.7 151.915 360.589 1470.420 18812.434 0.00
SUMMARY:
Total Building Cost = 202,514,962.7
Total Floor Area = 744.0 m2
Total CIP Slab Area = 744.0 m2
Total HCS Slab Area = 0.0 m2
Total Concrete Volume = 151.9 m3
Total Rebar Weight = 18812.4 kg
Total Formwork Area = 1470.4 m2
Average Concrete Volume = 0.2042 m3 / m2
Average Rebar Weight = 25.29 kg / m2
Average Overall Cost = 272,197.5 / m2
Compute
Print
Save
Ok
Case
Comb
Mode
Msg
Ready
Start
b-cost4 - Paint
SANS V.4.7 Program
4:11 PM

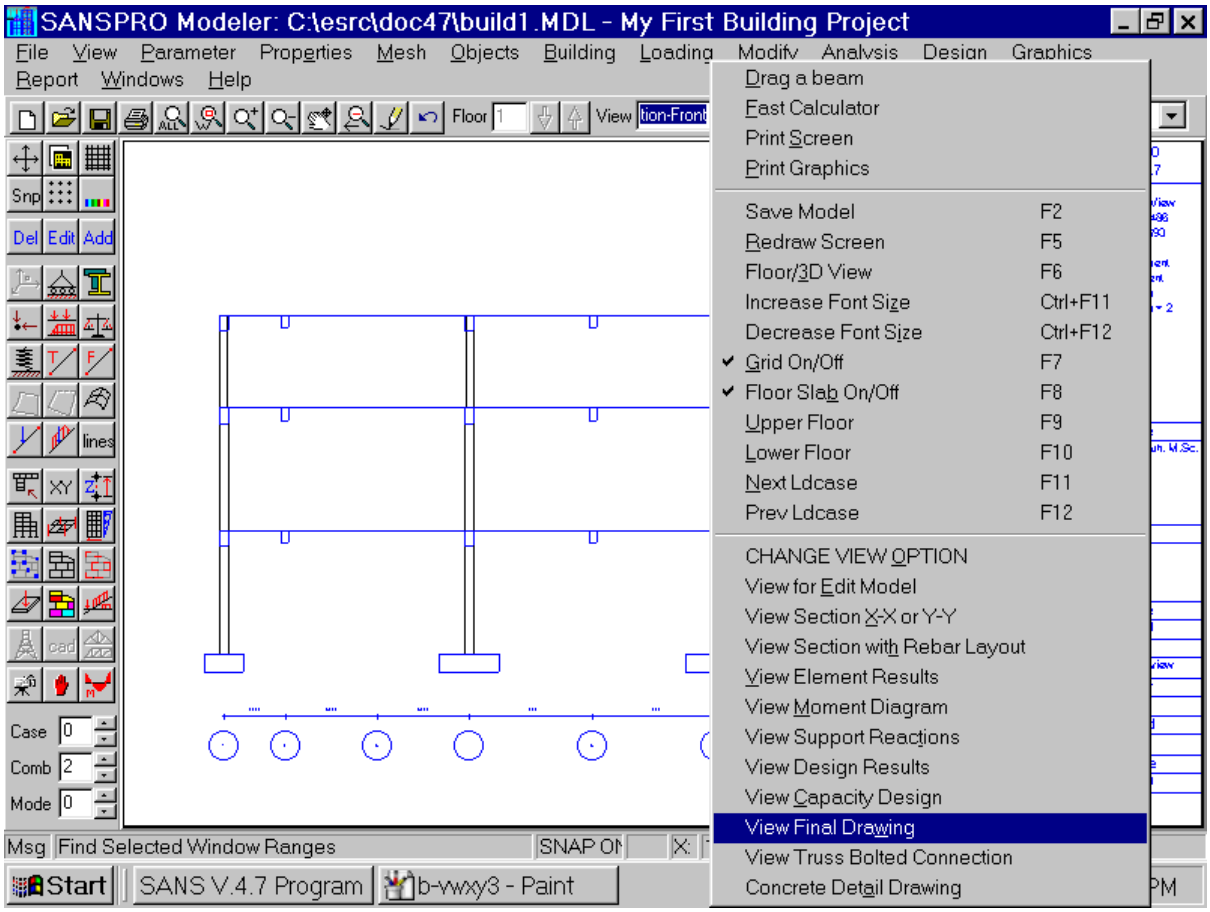
28. View Rebar Layout

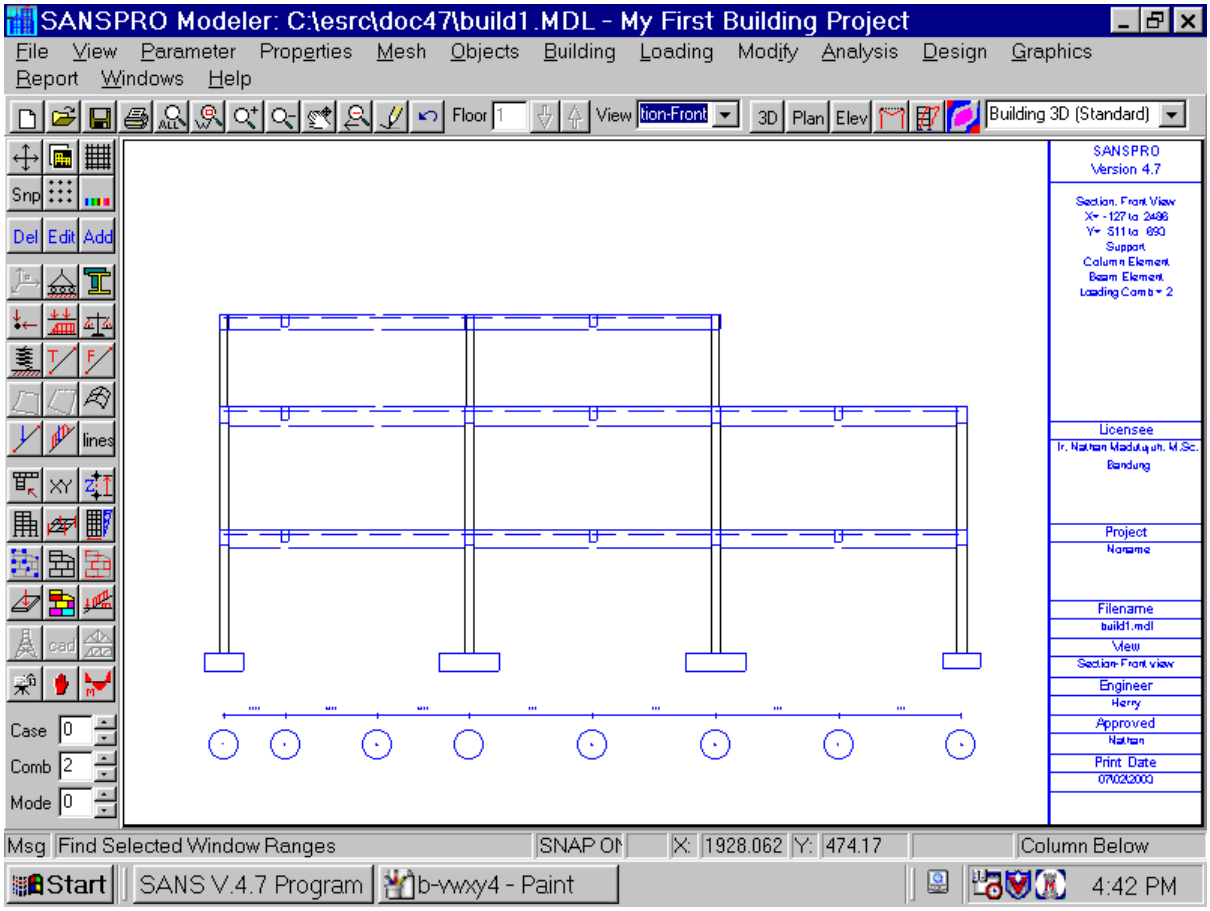
- SANSPRO also provides sectional view of rebar layout (portal view)
- Right-Click and select View Section
- Select the view range by dragging a window
- Select view direction (front or right)
- Select Ok
- Sectional View will be displayed
- Right-Click and select View Final Drawing/Size
- Right-Click and select View Section with Rebar Layout

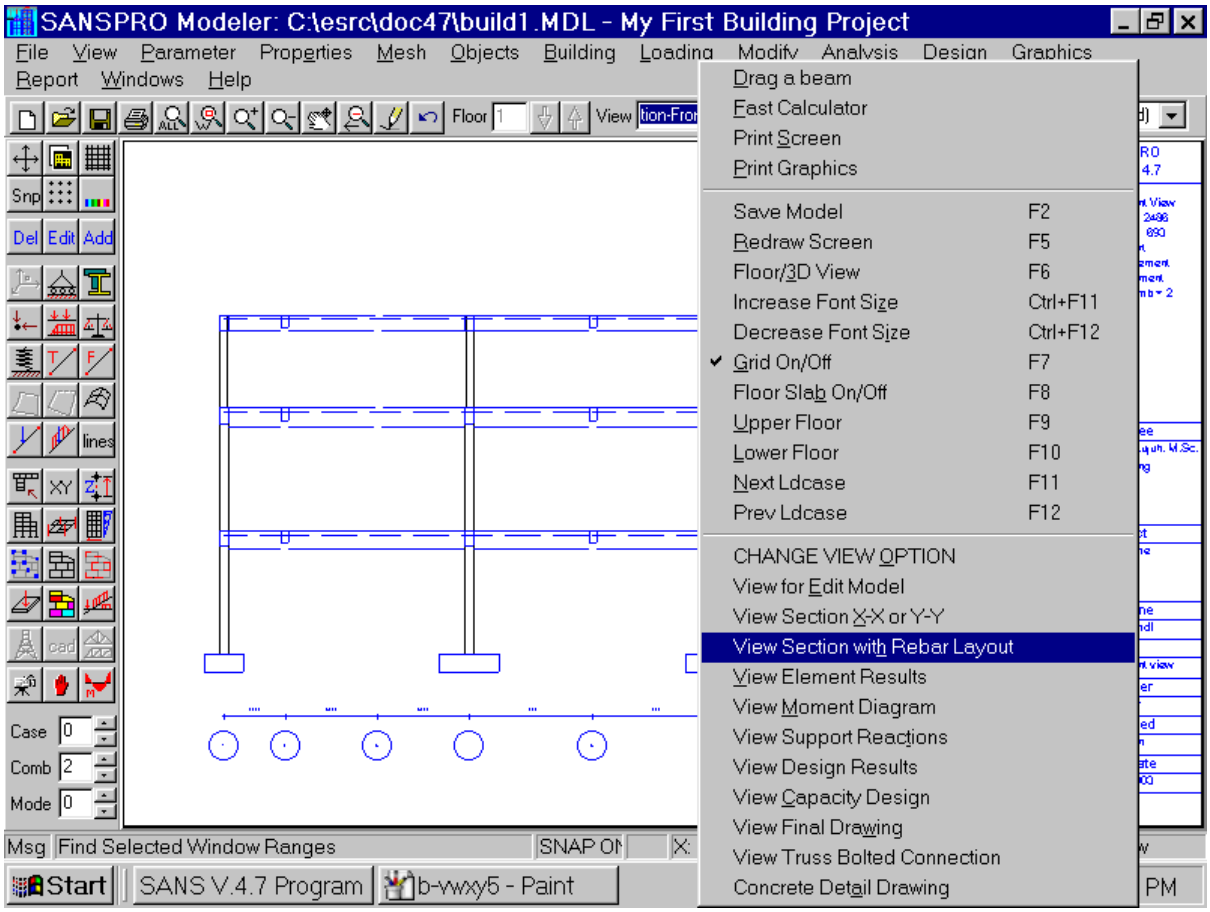


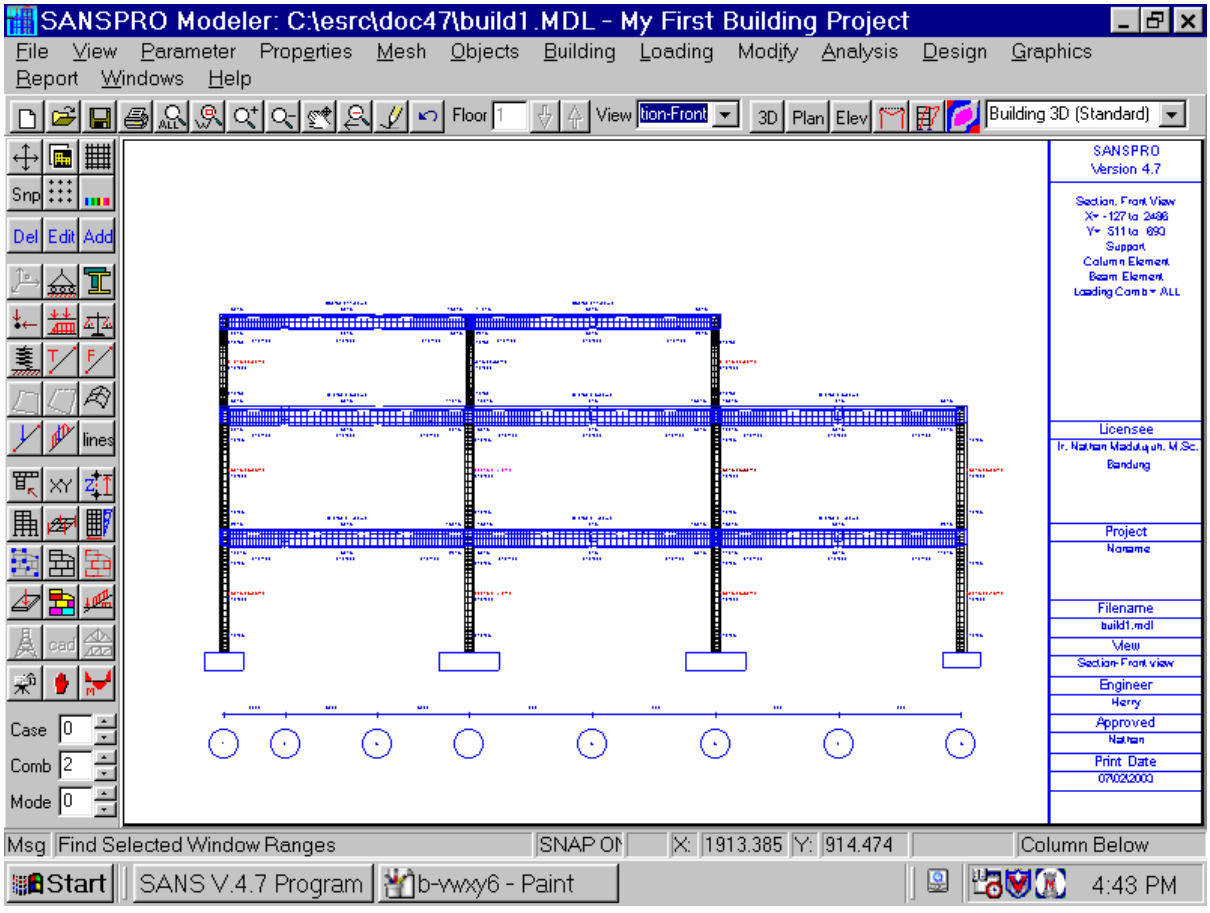

SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Sectional View of Buildin...
View Direction
Front
Back
Left
Right
Start Floor 0
End Floor 3
Margin 200
Select Range
View
Cancel
SANSPRO Version 4.7
Msg Ready
SNAP Off
X: 2054.545 Y: 456.818
Column Below
Start
SANS V.4.7 Program
b-vwxy1 - Paint
4:41 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
SANSPRO Version 4.7
Msg Find Selected Window Ranges
SNAP Off
X: 1380.128 Y: 288.264
Column Below
Start
SANS V.4.7 Program
b-vwxy2 - Paint
4:42 PM

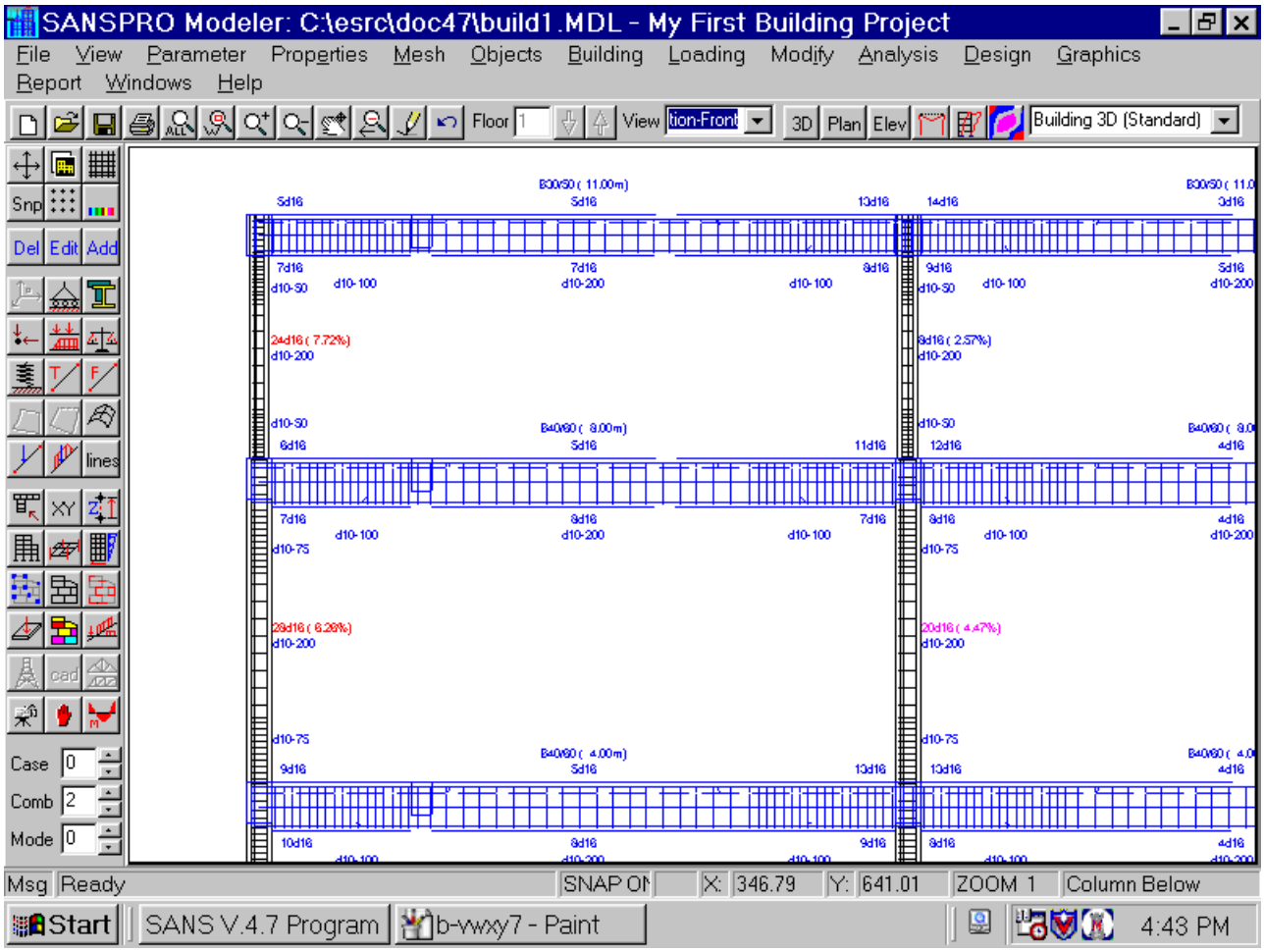

SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Drag a beam
Fast Calculator
Print Screen
Print Graphics
Save Model F2
Redraw Screen F5
Floor/3D View F6
Increase Font Size Ctrl+F11
Decrease Font Size Ctrl+F12
Grid On/Off F7
Floor Slab On/Off F8
Upper Floor F9
Lower Floor F10
Next Ldcase F11
Prev Ldcase F12
CHANGE VIEW OPTION
View for Edit Model
View Section X-X or Y-Y
View Section with Rebar Layout
View Element Results
View Moment Diagram
View Support Reactions
View Design Results
View Capacity Design
View Final Drawing
View Truss Bolted Connection
Concrete Detail Drawing
Msg Find Selected Window Ranges
Start
SANS V.4.7 Program
b-vwxy3 - Paint


SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Building 3D (Standard)
SANSPRO
Version 4.7
Licensee
Project
Filename
build1.mdl
Engineer
Approved
Print Date
Msg Find Selected Window Ranges
X: 1928.062 Y: 474.17
Column Below
Start
SANS V.4.7 Program
b-vwxy4 - Paint
4:42 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Drag a beam
Fast Calculator
Print Screen
Print Graphics
Save Model F2
Redraw Screen F5
Floor/3D View F6
Increase Font Size Ctrl+F11
Decrease Font Size Ctrl+F12
Grid On/Off F7
Floor Slab On/Off F8
Upper Floor F9
Lower Floor F10
Next Ldcase F11
Prev Ldcase F12
CHANGE VIEW OPTION
View for Edit Model
View Section X-X or Y-Y
View Section with Rebar Layout
View Element Results
View Moment Diagram
View Support Reactions
View Design Results
View Capacity Design
View Final Drawing
View Truss Bolted Connection
Concrete Detail Drawing
Msg Find Selected Window Ranges
Start
SANS V.4.7 Program
b-vwxy5 - Paint


SANSPRO Modeler: C:\esrc\doc47\build1.MDL - My First Building Project
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Building 3D (Standard)
SANSPRO
Version 4.7
Licensee
Project
Filename
build1.mdl
Engineer
Approved
Print Date
Msg Find Selected Window Ranges
X: 1913.385 Y: 914.474
Column Below
Start
SANS V.4.7 Program
b-vwxy6 - Paint
4:43 PM

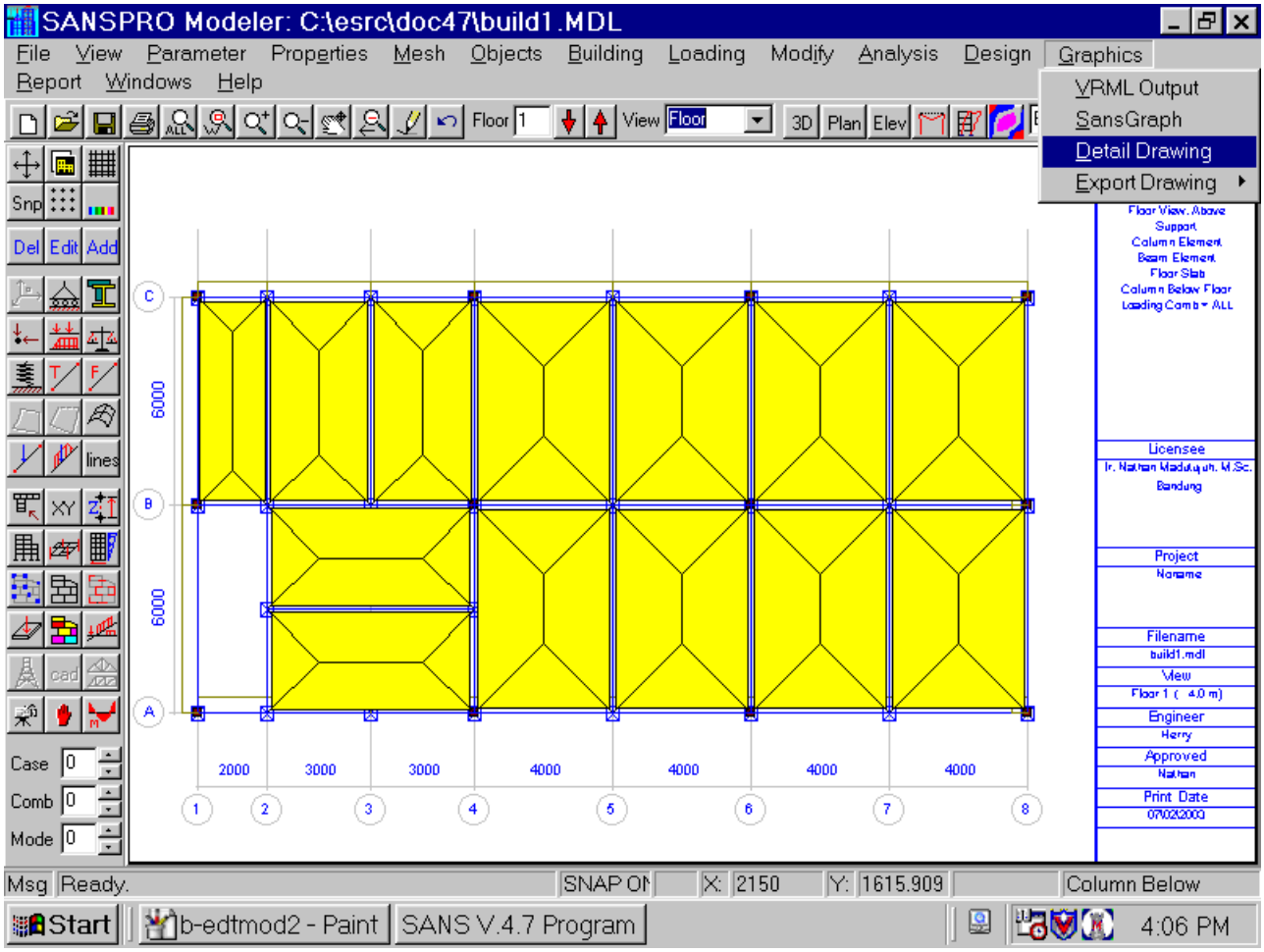
- To see larger drawing, use zoom in or zoom rectangle/window.
- The sectional drawing can be printed or saved to AutoCad DXF format to be editted inside AutoCad or other DXF compatible CAD programs.
- Increase/decrease font size by clicking Ctlr-F11 or Ctrl-F12

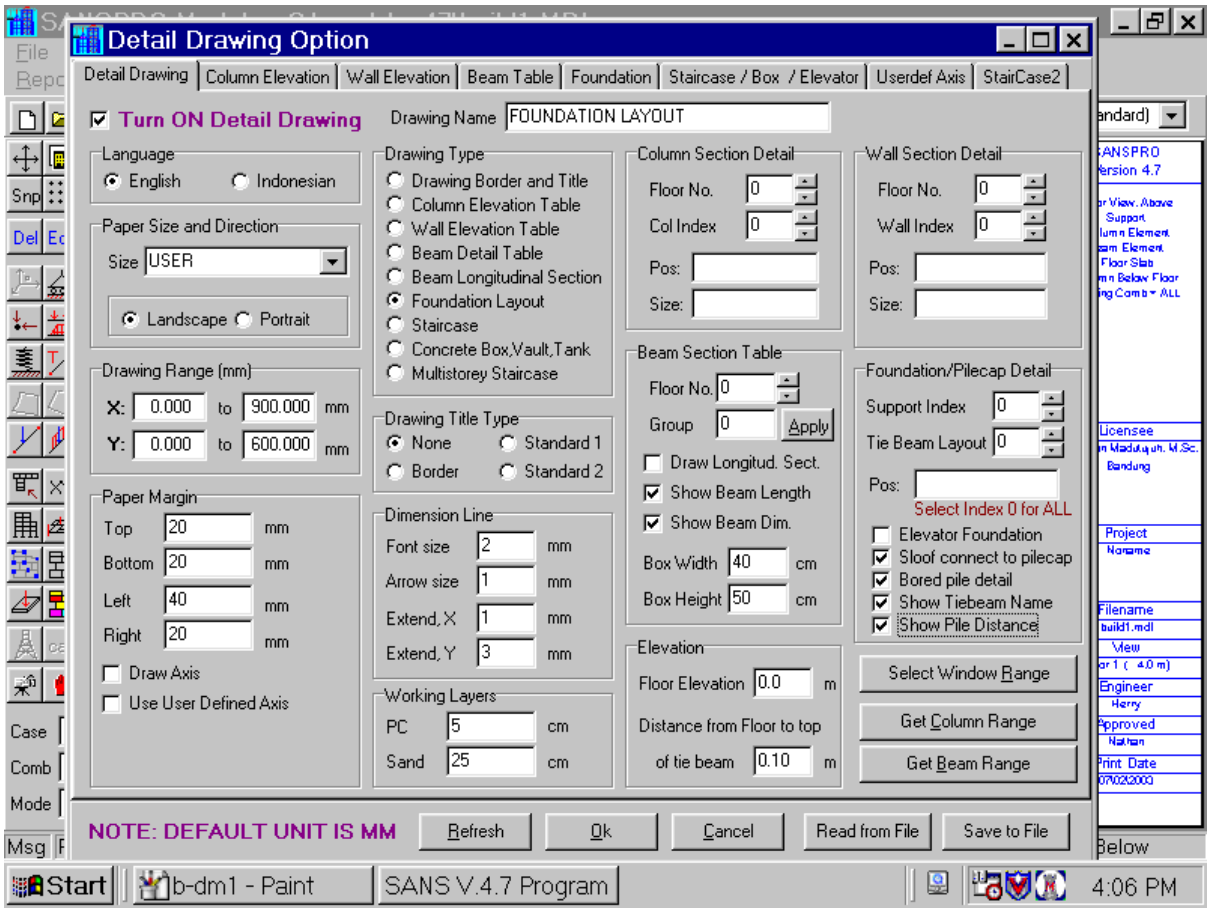


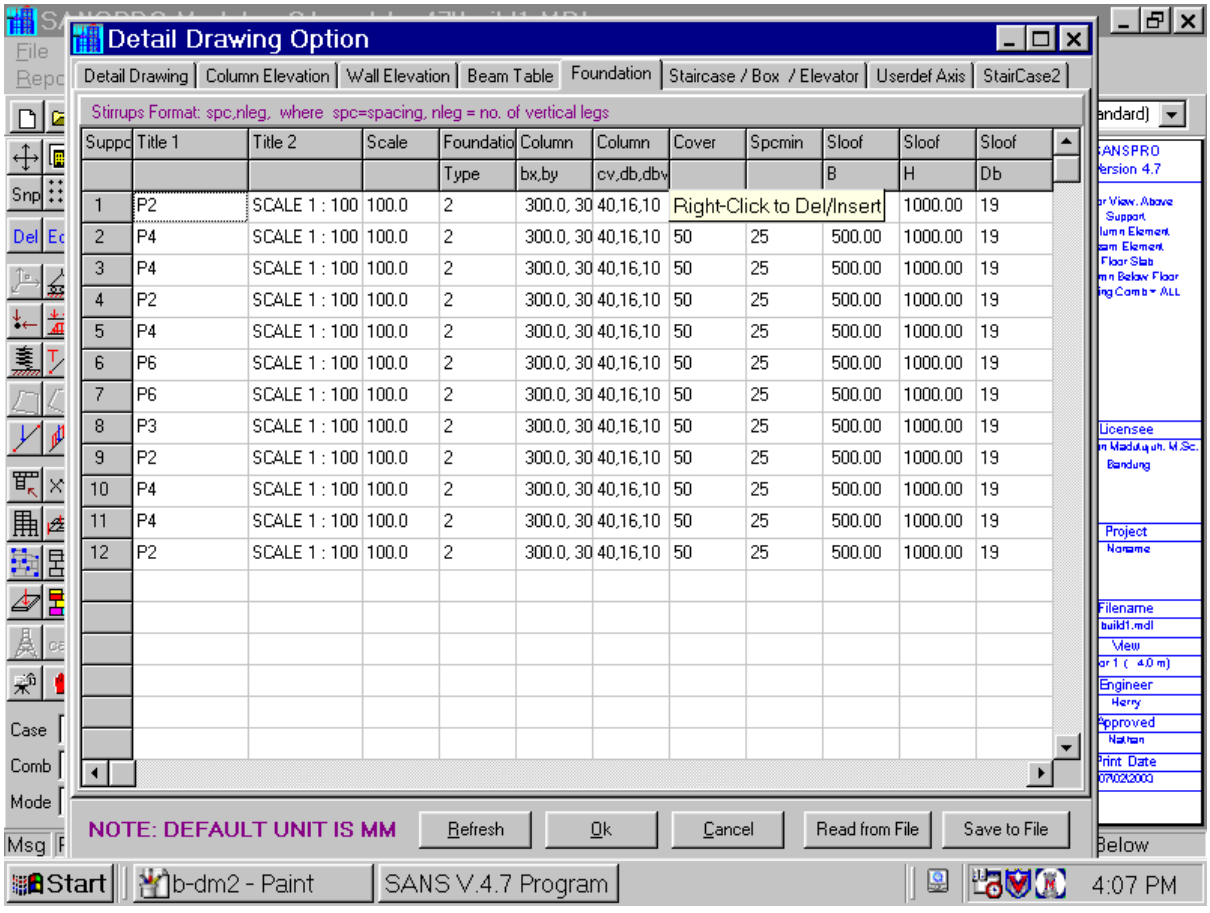
29. Generating Drawing

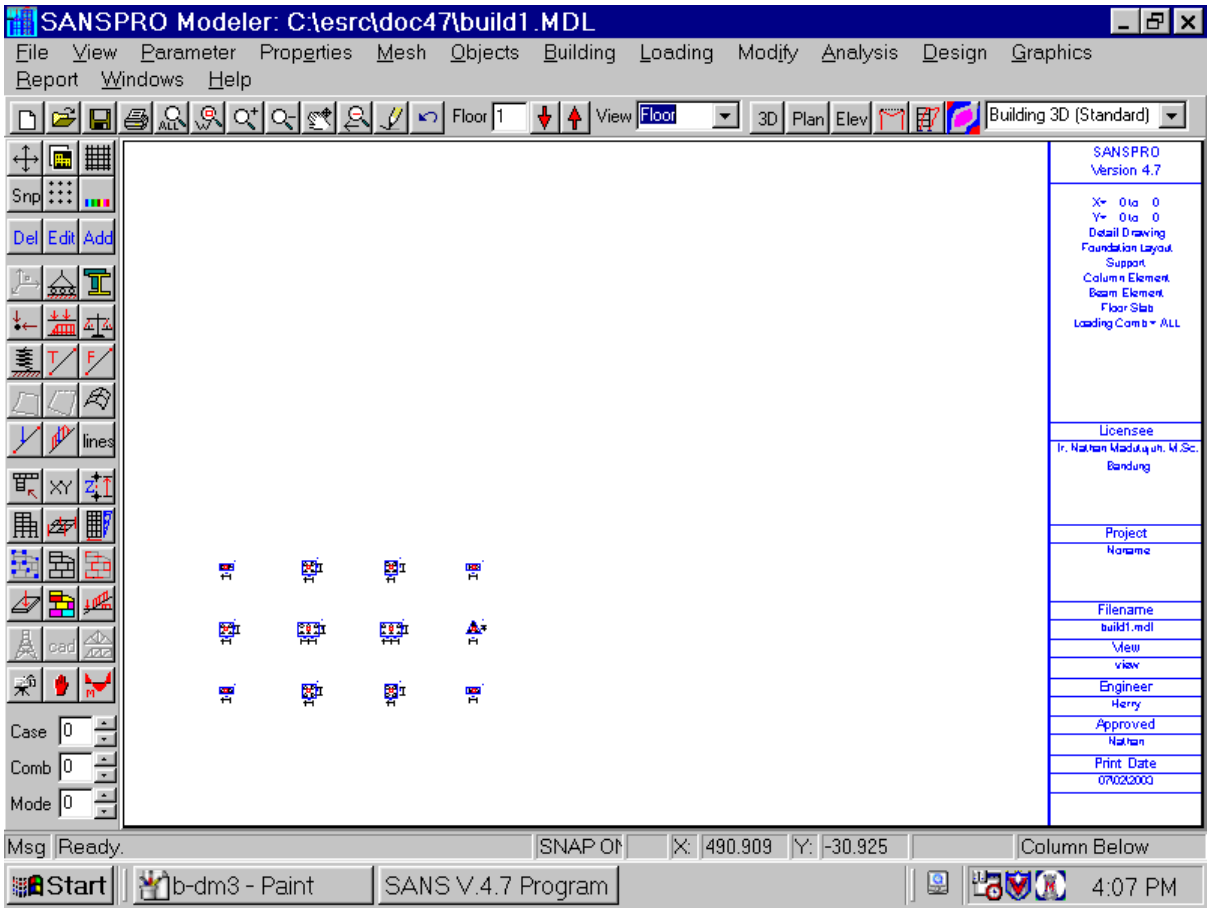
- SANSPRO is capable to generate detail structural drawing in seconds.
- Only SANSPRO with Drawing Module activated can be used to generate drawings
- Before drawn, the drawing parameters are stored to a table that can be editted by user
- This approach is called Parametric Drawing
- Objects that can be drawn by SANSPRO are:
    - Beam sectional detail
    - Beam longitudinal detail
    - Column elevation table
    - Foundation/Pile layout
    - Pilecap detail
    - Other objects

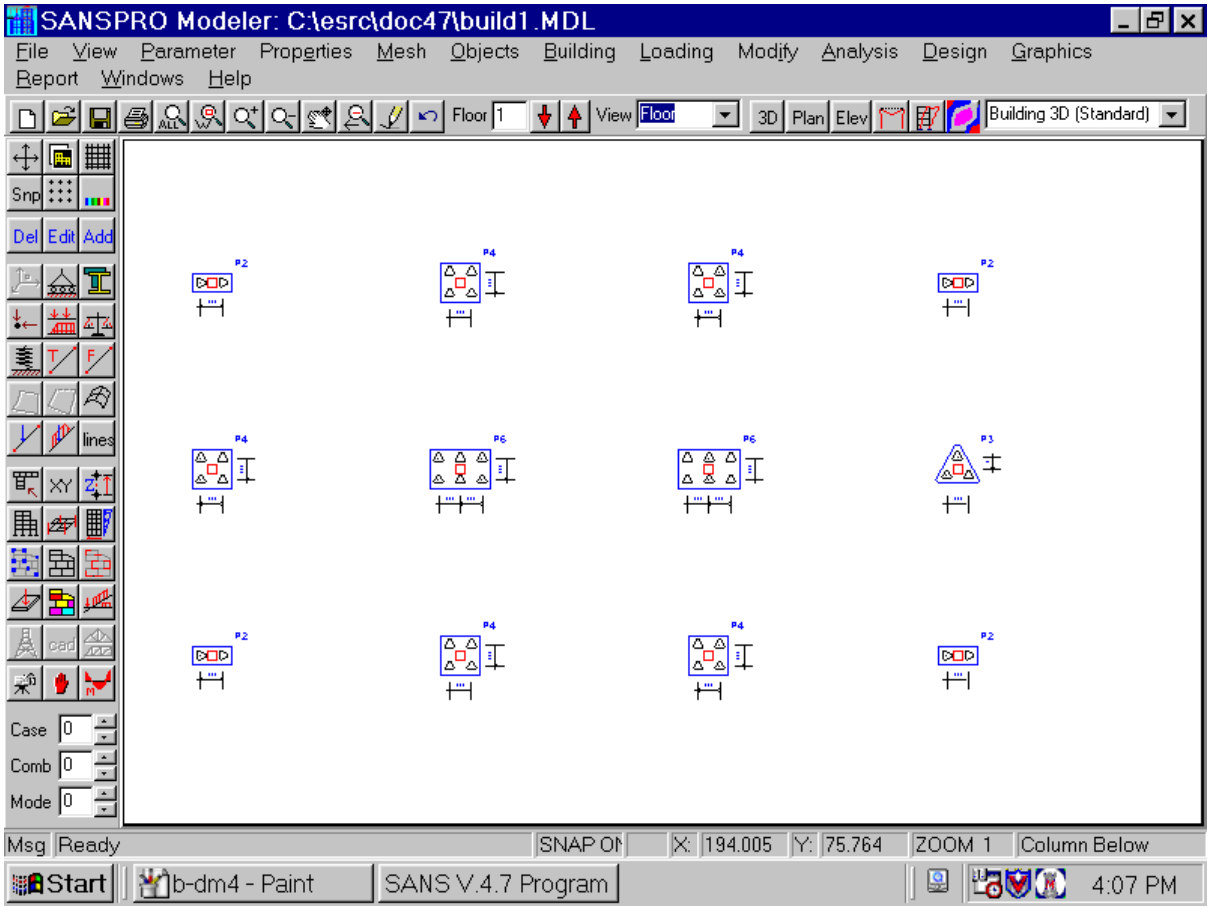
- Click **Graphics-Detail Drawing** to call the Drawing Generator Module
- Use Zoom to get a larger view




Detail Drawing Option
Detail Drawing
Column Elevation
Wall Elevation
Beam Table
Foundation
Staircase / Box / Elevator
Userdef Axis
StairCase2
Turn ON Detail Drawing
Drawing Name
FOUNDATION LAYOUT
Language
English
Indonesian
Paper Size and Direction
Size
USER
Landscape
Portrait
Drawing Range (mm)
X: 0.000 to 900.000 mm
Y: 0.000 to 600.000 mm
Paper Margin
Top 20 mm
Bottom 20 mm
Left 40 mm
Right 20 mm
Draw Axis
Use User Defined Axis
Drawing Type
Drawing Border and Title
Column Elevation Table
Wall Elevation Table
Beam Detail Table
Beam Longitudinal Section
Foundation Layout
Staircase
Concrete Box,Vault,Tank
Multistorey Staircase
Drawing Title Type
None
Standard 1
Border
Standard 2
Dimension Line
Font size 2 mm
Arrow size 1 mm
Extend, X 1 mm
Extend, Y 3 mm
Working Layers
PC 5 cm
Sand 25 cm
Column Section Detail
Floor No. 0
Col Index 0
Pos:
Size:
Beam Section Table
Floor No. 0
Group 0
Apply
Draw Longitud. Sect.
Show Beam Length
Show Beam Dim.
Box Width 40 cm
Box Height 50 cm
Elevation
Floor Elevation 0.0 m
Distance from Floor to top of tie beam 0.10 m
Wall Section Detail
Floor No. 0
Wall Index 0
Pos:
Size:
Foundation/Pilecap Detail
Support Index 0
Tie Beam Layout 0
Pos:
Select Index 0 for ALL
Elevator Foundation
Sloof connect to pilecap
Bored pile detail
Show Tiebeam Name
Show Pile Distance
Select Window Range
Get Column Range
Get Beam Range
NOTE: DEFAULT UNIT IS MM
Refresh
Ok
Cancel
Read from File
Save to File
Start
b-dm1 - Paint
SANS V.4.7 Program
4:06 PM


Detail Drawing Option
Detail Drawing
Column Elevation
Wall Elevation
Beam Table
Foundation
Staircase / Box / Elevator
Userdef Axis
StairCase2
Stirrups Format: spc,nleg, where spc=spacing, nleg = no. of vertical legs
Suppc
Title 1
Title 2
Scale
Foundatio Type
Column bx,by
Column cv,db,dbv
Cover
Spcmin
Sloof B
Sloof H
Sloof Db
1 P2 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 Right-Click to Del/Insert 1000.00 19
2 P4 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
3 P4 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
4 P2 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
5 P4 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
6 P6 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
7 P6 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
8 P3 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
9 P2 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
10 P4 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
11 P4 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
12 P2 SCALE 1 : 100 100.0 2 300.0, 30 40,16,10 50 25 500.00 1000.00 19
NOTE: DEFAULT UNIT IS MM
Refresh
Ok
Cancel
Read from File
Save to File
Start
b-dm2 - Paint
SANS V.4.7 Program
4:07 PM


SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
Snp Del Edit Add lines Case 0 Comb 0 Mode 0
SANSPRO Version 4.7
Detail Drawing
Foundation Layout
Support
Column Element
Beam Element
Floor Slab
Loading Comb = ALL
Licensee
Ir. Nathan Madutujuh, M.Sc.
Bandung
Project
Noname
Filename
build1.mdl
View
view
Engineer
Herry
Approved
Nathan
Print Date
07/02/2003
Msg Ready. SNAP Of X: 490.909 Y: -30.925 Column Below
Start b-dm3 - Paint SANS V.4.7 Program 4:07 PM

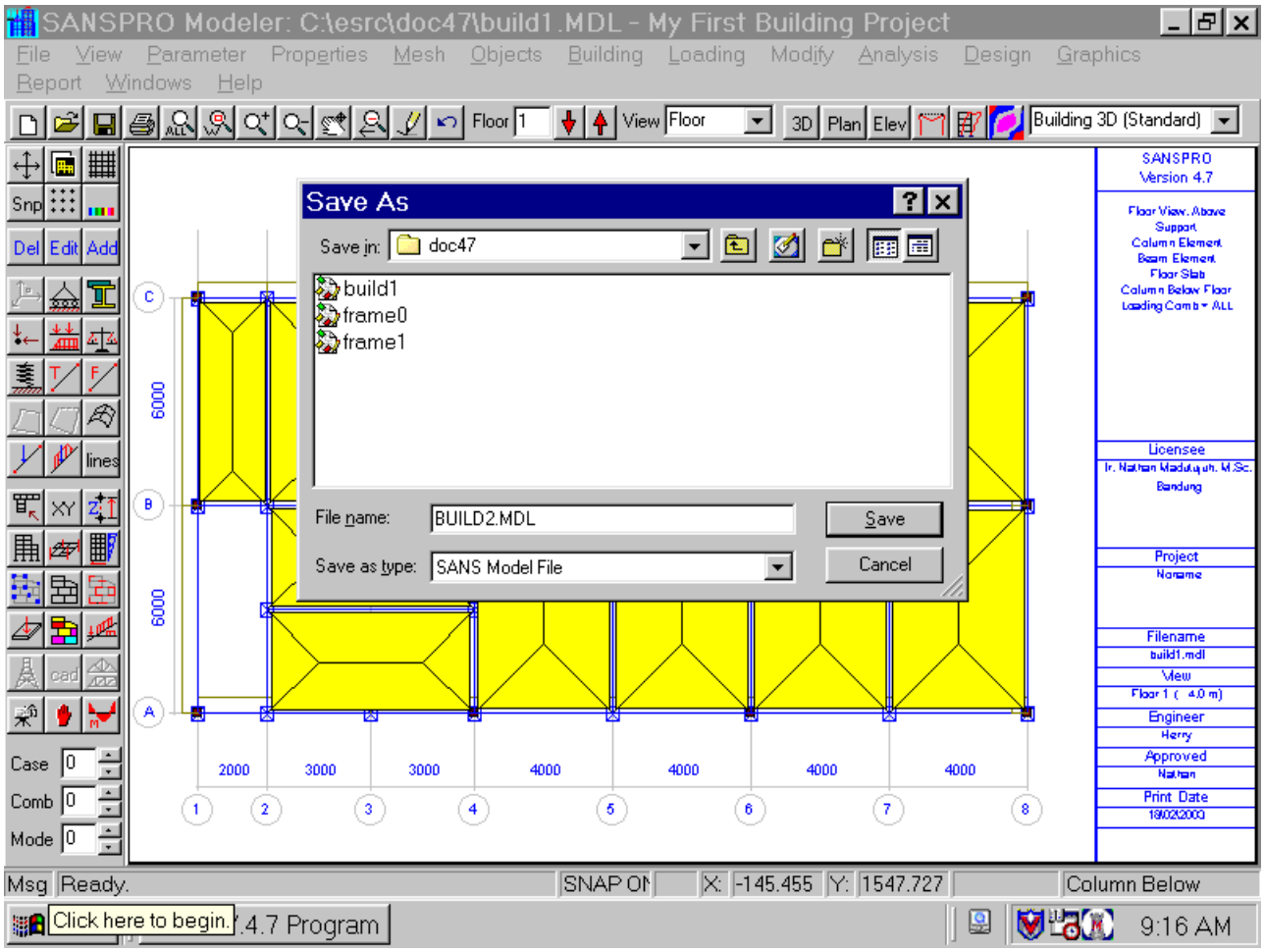

SANSPRO Modeler: C:\esrc\doc47\build1.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Floor 1 View Floor 3D Plan Elev Building 3D (Standard)
Snp Del Edit Add lines Case 0 Comb 0 Mode 0
P2 P4 P4 P2
P4 P6 P6 P3
P2 P4 P4 P2
Msg Ready SNAP Of X: 194.005 Y: 75.764 ZOOM 1 Column Below
Start b-dm4 - Paint SANS V.4.7 Program 4:07 PM

Dynamic Analysis:

SANSPRO can be used for dynamic analysis by adding few model changes:

- Mass Contribution Factor
- Spectrum Response
- Dynamic Analysis Option

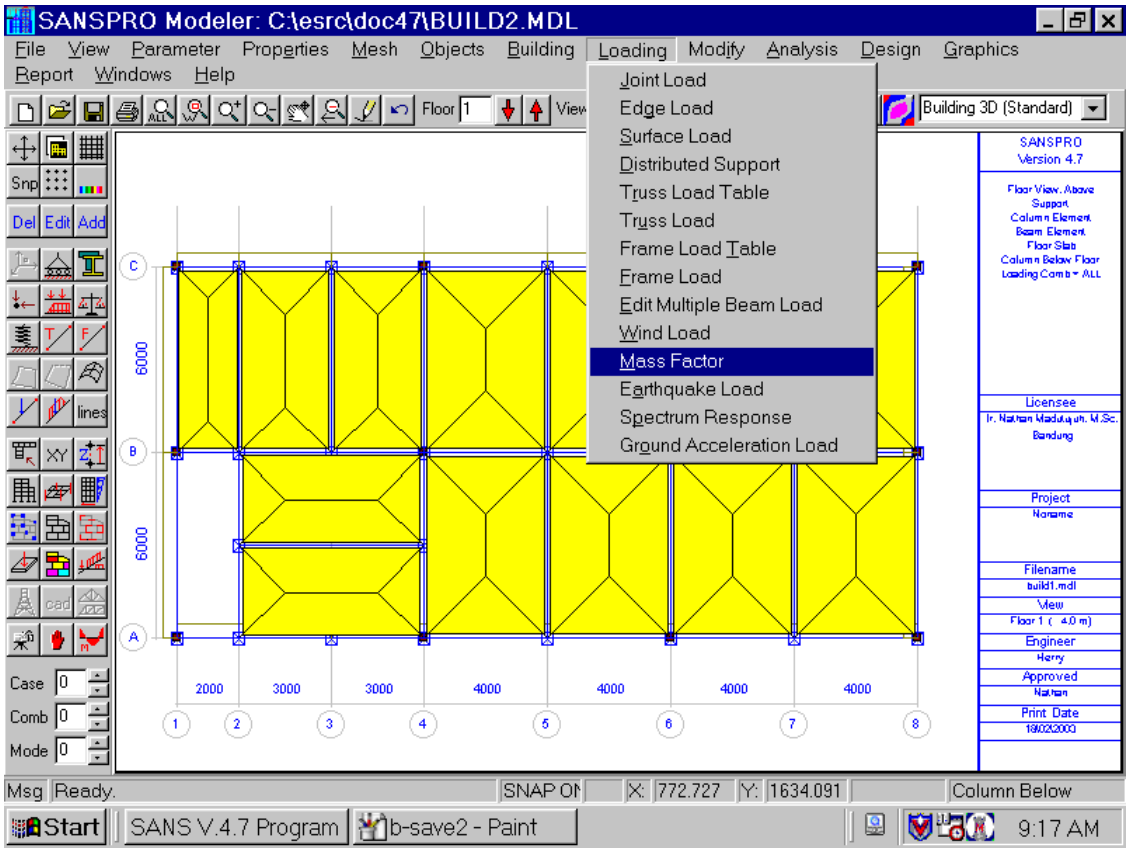
First, we must **save** the prevous model file into a new file named BUILD2.MDL so that we will have two models and two **incremental design** results.

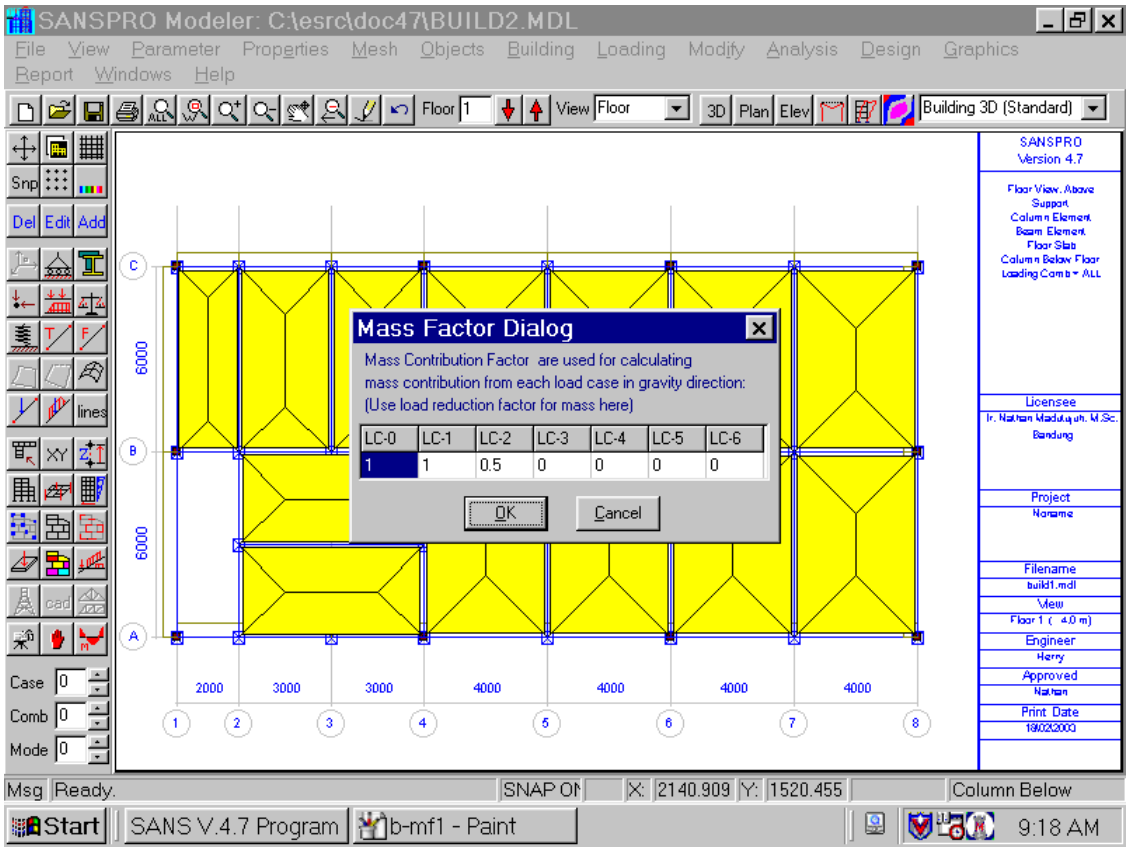


After saving to BUILD2.MDL, do the following:

1. Define Mass Factor

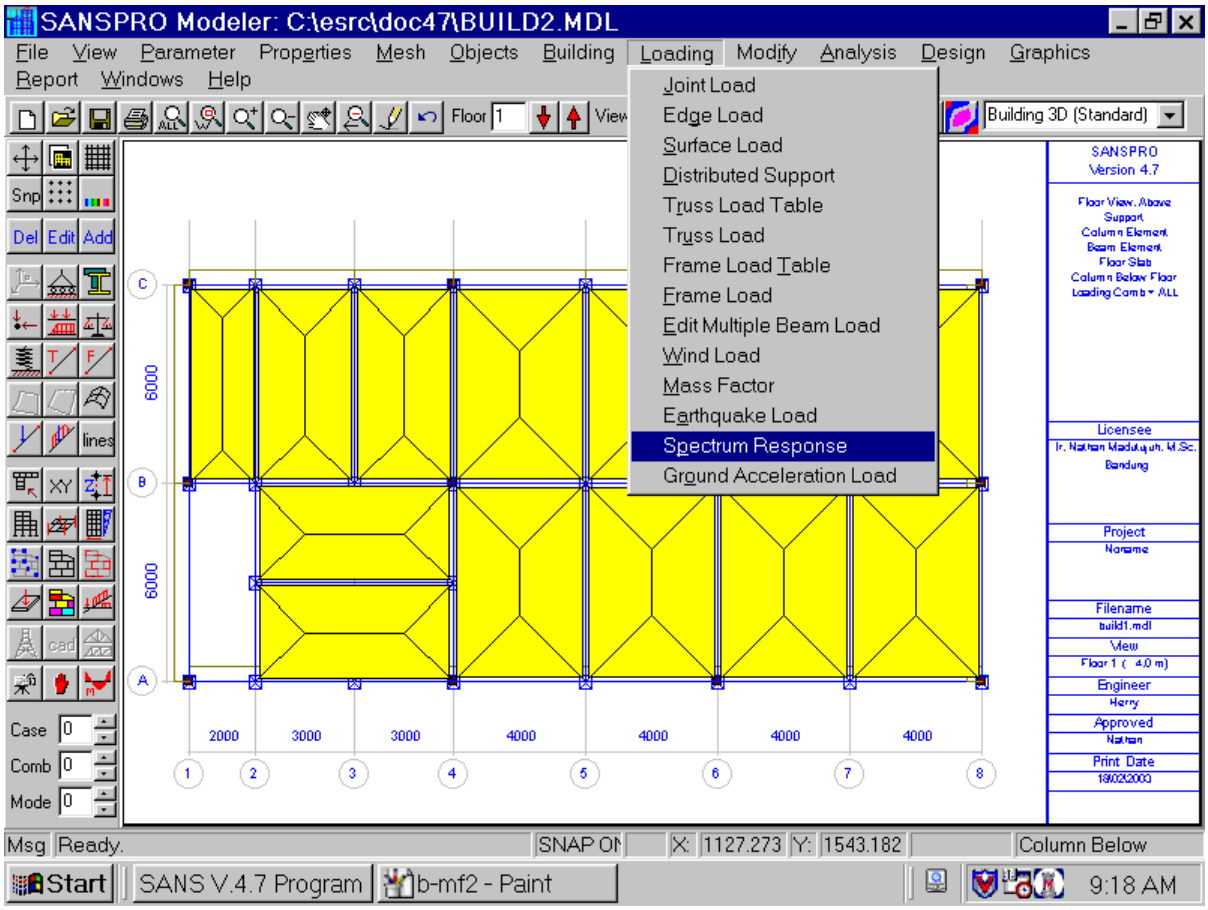
Mass Contribution Factors are multipliers for each load case for calculating nodal masses. Usually for self-weight (load case 0) and dead load (load case 1) we use 1.0 (100%), for live load (load case 2) = 0.3 to 0.8, while for earthquake/lateral load case, mass factor = 0.

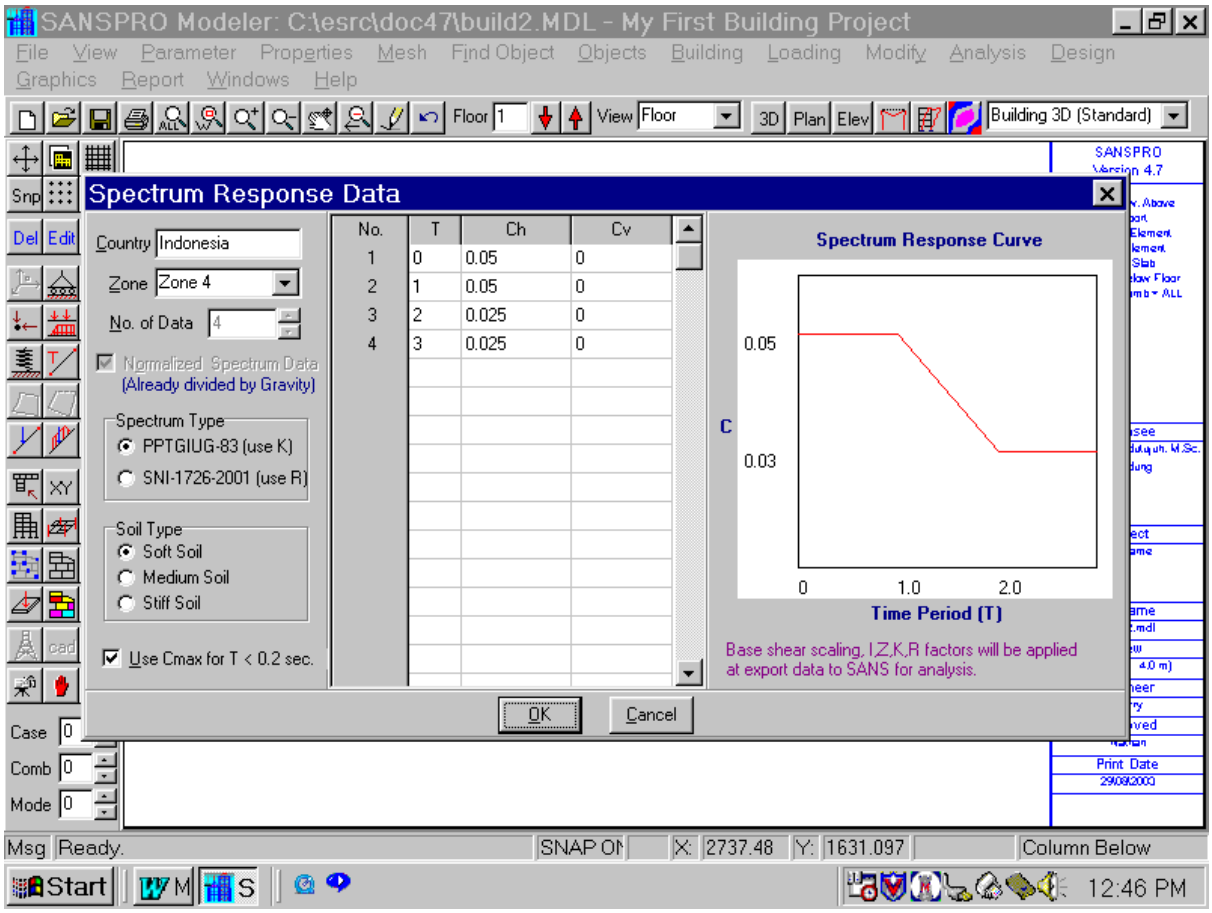




2. Define Spectrum Response

Dynamic Analysis Method that will be used is Dynamic Response Spectrum Analysis. The method needs Response Spectrum Data that depends on the earthquake zone and soil stiffness. Most country Building Regulators already have the response spectrum data.



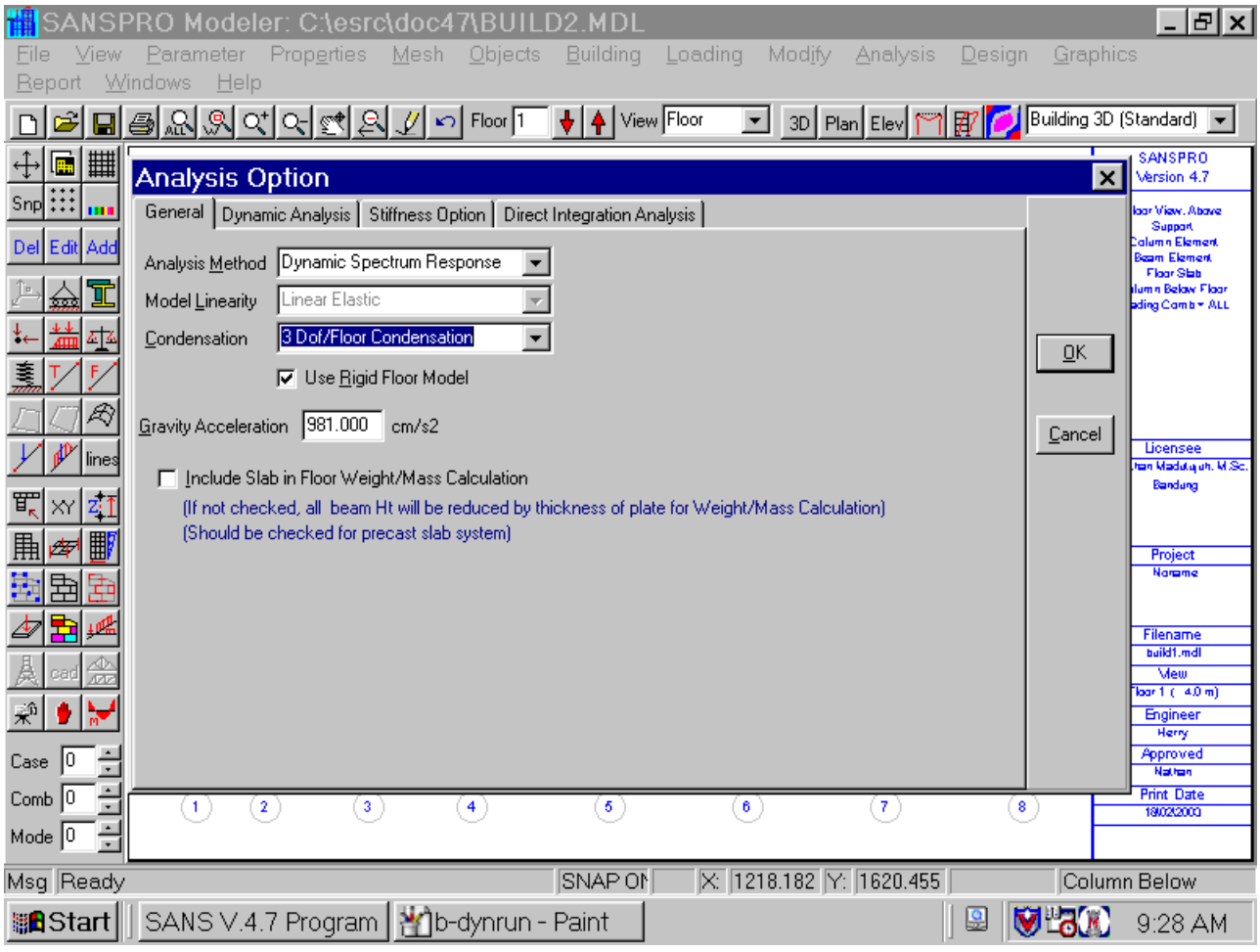

SANSPRO Modeler: C:\esrc\doc47\build2.MDL - My First Building Project
File View Parameter Properties Mesh Find Object Objects Building Loading Modify Analysis Design
Graphics Report Windows Help
Spectrum Response Data
Country Indonesia
Zone Zone 4
No. of Data 4
Normalized Spectrum Data
(Already divided by Gravity)
Spectrum Type
PPTGIUG-83 (use K)
SNI-1726-2001 (use R)
Soil Type
Soft Soil
Medium Soil
Stiff Soil
Use Cmax for T < 0.2 sec.
No. T Ch Cv
1 0 0.05 0
2 1 0.05 0
3 2 0.025 0
4 3 0.025 0
Spectrum Response Curve
C
0.05
0.03
0
1.0
2.0
Time Period (T)
Base shear scaling, I,Z,K,R factors will be applied at export data to SANS for analysis.
OK
Cancel
Case
Comb
Mode
Msg Ready.
SNAP ON
X: 2737.48
Y: 1631.097
Column Below
Start
12:46 PM

3. Dynamic Analysis Option

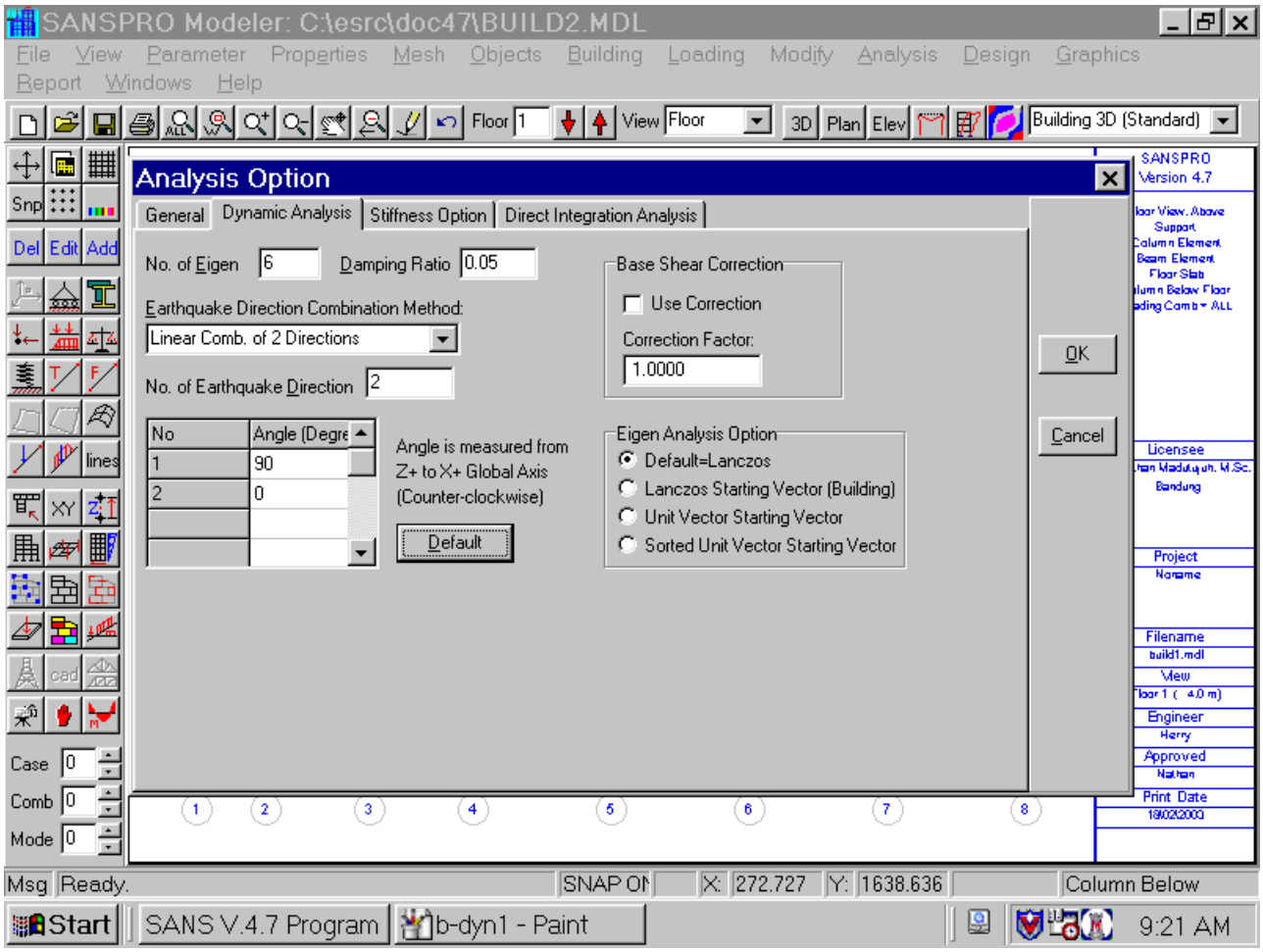
We need also to select option for Dynamic Spectrum Response Analysis.
Enter the following parameters:

- Number of Eigen NE = 3*MO <= 3*NF (MO = Mode shape order, 1..NF)
- Earthquake Direction = 2 (X and Z direction, measured from Z+)
- Earthquake Direction = 90 and 0 degrees measured from Z+
- Static Condensation = 3 Dof/Floor
- Rigid Floor = YES

- MO should be >= 2 or 3, <= NF
- Rigid Floor = Yes will make dynamic analysis faster and give almost same eigen values.
- Static Condensation is a method used by SANSPRO to reduce the linear equation size:
  - Static Condensation = None : No DOF reductions
  - Static Condensation = Zero Mass : DOF with zero mass will be reduced
  - Static Condensation = 3 Dof/Floor : For lateral load case, NDOF = 3 dof/flr
- For building, using Static Condensation = 3 Dof/Floor will give a good results.



- Number of Eigen = 6
- Damping Ratio = 0.05 (5%)
- Earthquake Direction = Linear Combination of two perpendicular Earthquakes Dir.
- Direction Angles: 90, 0 degrees
- Base Shear Correction: Unchecked
- Eigen Analysis Option: Default = Lanczos Method



Explanation:

- Most concrete buildings have damping ratio between 2% to 5% (0.02 to 0.05)

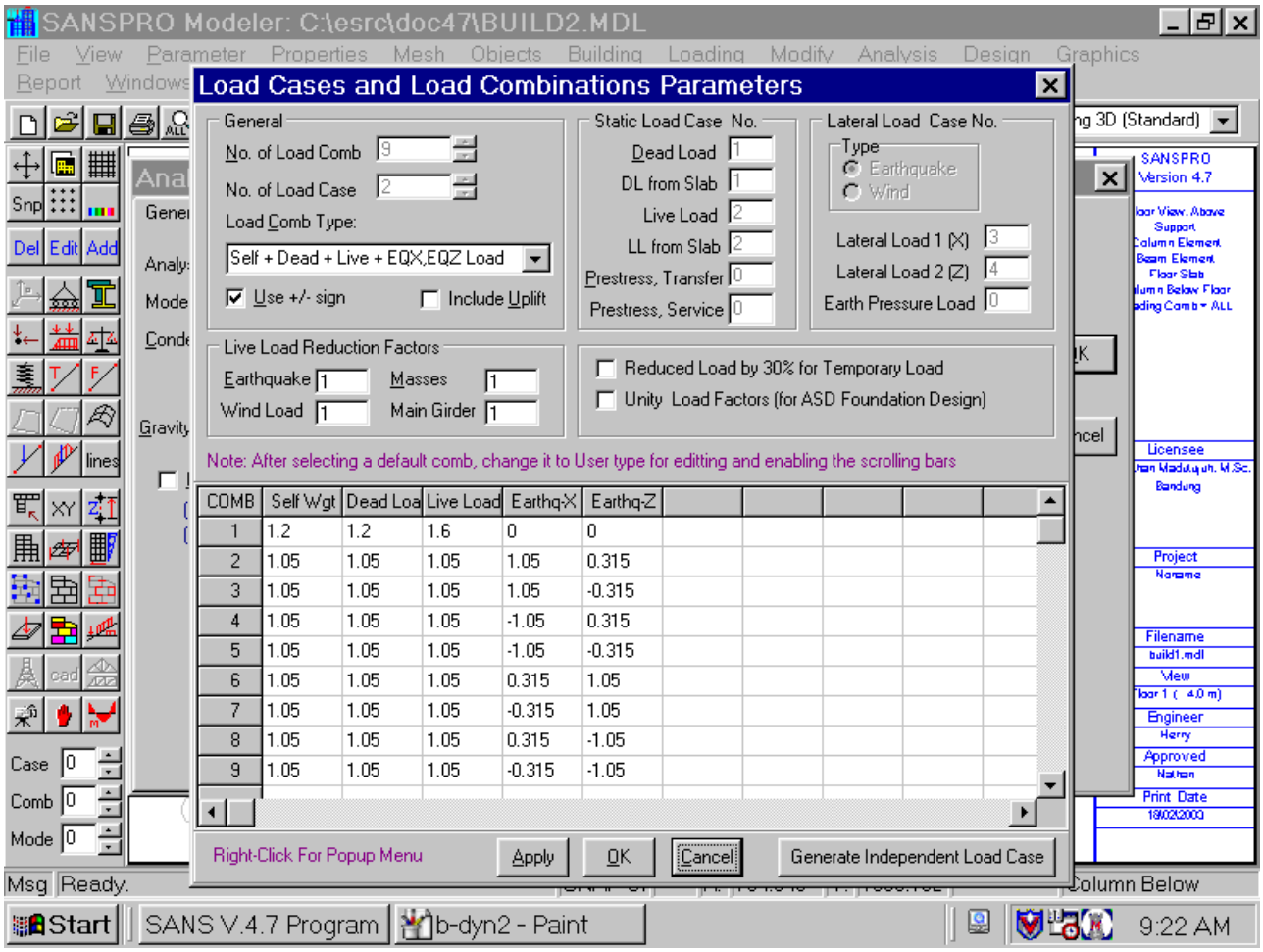
- SANSPRO can deal with several earthquake directions at once.
- Most regular buildings only require two perpendicular earthquake directions.
- Earthquake direction angles are measured from Z+ to X+ (counter-clockwise)

- Eigen Solver uses Lanczos Method to find starting vector for Modified Subspace Iteration
- Sometimes the default starting vector fails to converge, in this case user can select other option.

SANSPRO load cases are only for external load.

Earthquake load is not counted as load case, so the number of load case is automatically reduced to 2, for dead load and live load.

But, load combination requires load factors also for earthquake loading.

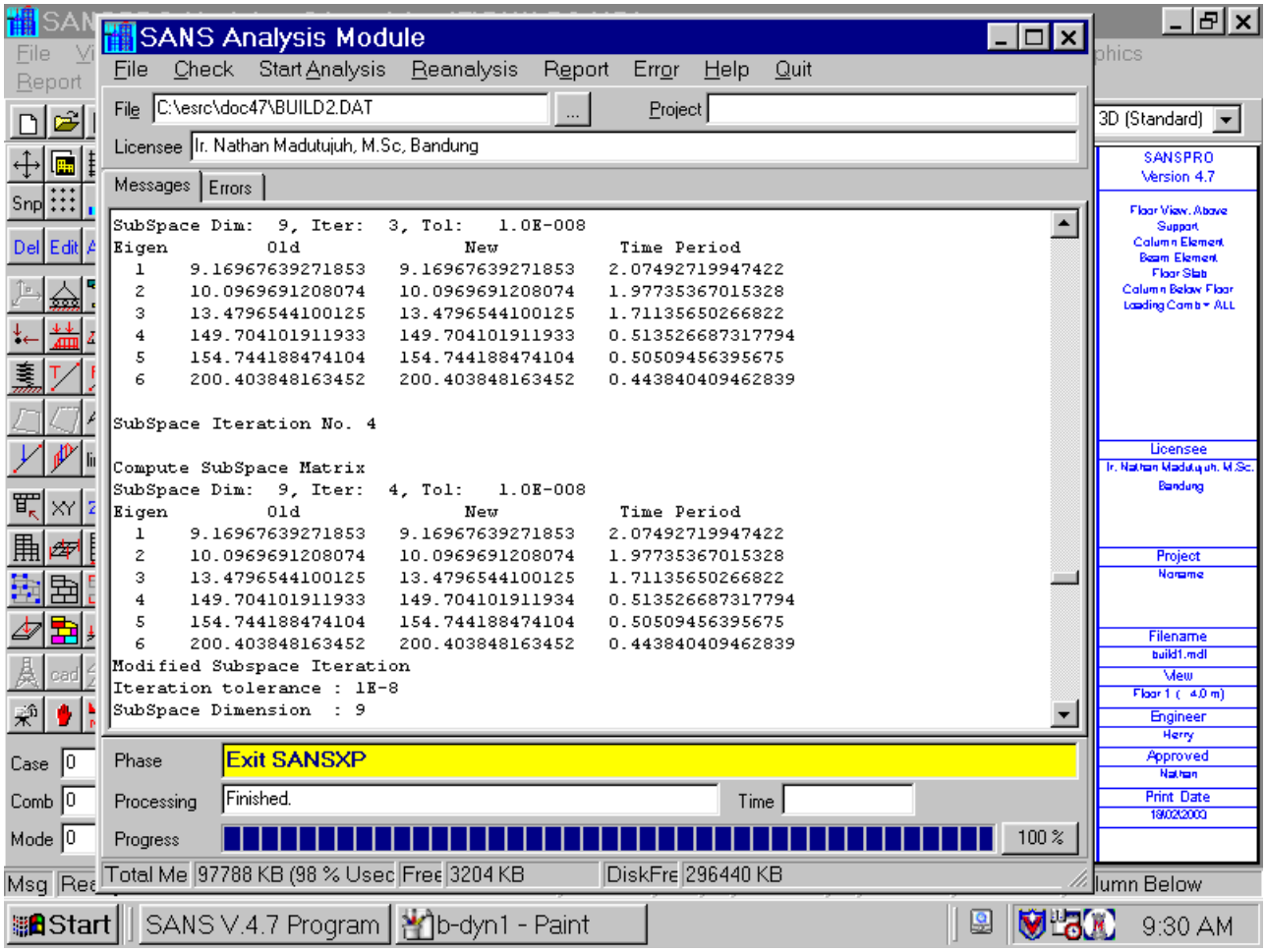


4. Run Dynamic Analysis

After defining needed parameters, the process to run dynamic analysis is exactly the same as for static analysis:

Compute Floor Weight to calculate center of mass (if you have not done it yet)
F2 – Save the model
F4 – Export the model
Analysis-Menu-Start Analysis
Output files from Dynamic Analysis:

BUILD2.OUT	- Deformations, Reactions, Element Forces (Text)
BUILD2.BIN	- Deformations, Reactions, Element Forces (Binary)
BUILD2.GRA	- Deformations and Mode Shapes (SANSGRAPH)
BUILD2.GEN	- Expanded Data
BUILD2.ROT	- Floor Individual Eccentricity
BUILD2.BSH	- Base Shear, Effective Mass, MPF, MDF, Scale
BUILD2.RSP	- Eigen Values, Mode Shape
BUILD2.SSH	- Building Story Shear
BUILD2.DSP	- Building Story Displacement



5. Check Dynamic Output

Some checkings need to be done for dynamic analysis output:

- Natural Period

To should be approximately 0.1 NF (depends on structural type and layout)
Natural period or dominant period To should be less than the maximum requirements, usually To << Tmax = 0.17 * NF

- First Mode/Dominant Mode

First mode must be translational mode.
It can be seen from the Mode Shape graphics that the shape of the building does not twist. It this is not the case, one must rearrange building layout.

- Mass Participation Factor

Effective Mass Factor or Mass Contribution Factor for the results should be larger than 90%. If this is not the case, one must increase number of eigen.

- Effective Mass Factor

- Modal Direction Factor

Determine in which direction a mode shape is dominant
One should avoid rotational mode as first mode

- Base Shear

Dynamic Base Shear should be larger than 80% Static Base Shear (Vd >= 0.8 Vs)
If this is not the case, check the following:

1. Is the structure too flexible, To near Tmax, or column/wall to few or small, that cause the earthquake coefficient taken from Response Spectrum Chart is very small. If so, increase column/wall stiffness

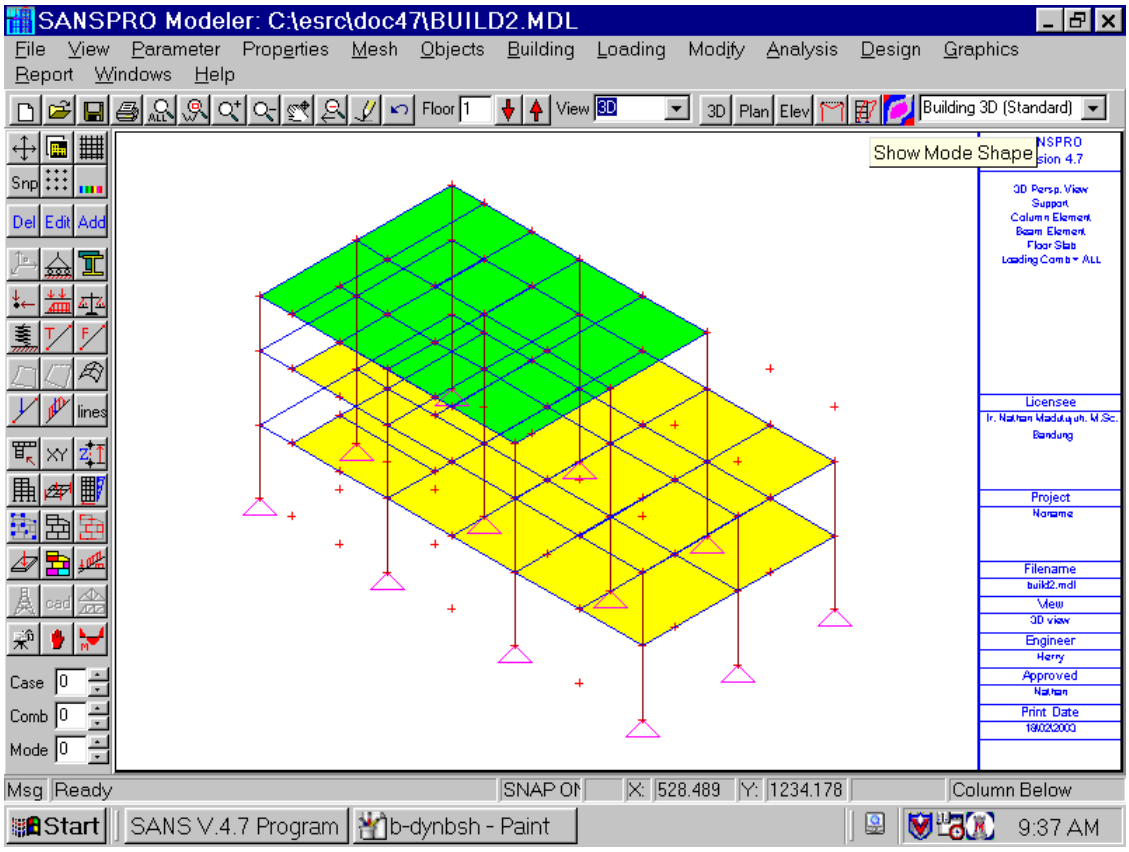
2. Is the mass too large, To will be large also, causing small earthquake coefficient. If so, check mass reduction factors being used.

3. Is the dominant mode not translational but rotational. In this case most of the earthquake energy is absorbed as twisting/rotational deformation, causing small base shear. Rearrange the size and location of columns/walls to achieve smaller eccentricity.

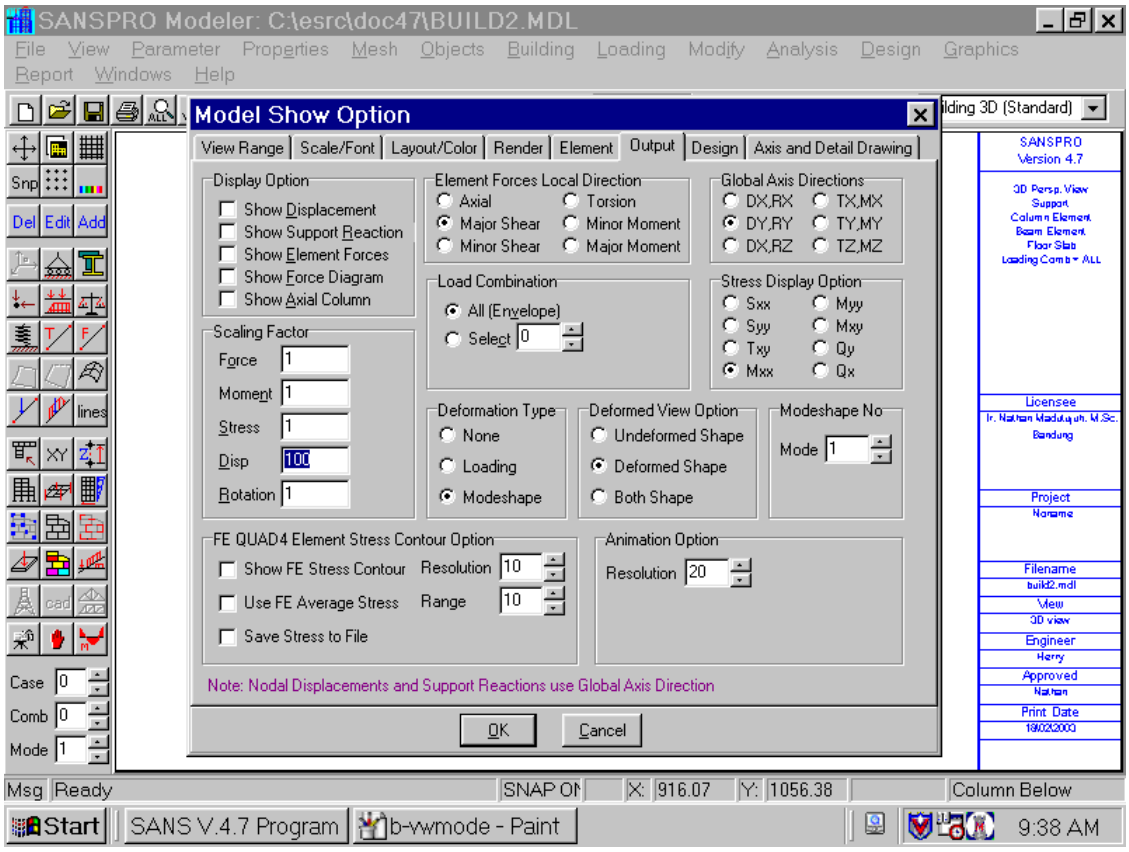
- Story Shear

- Story Displacement and Drift

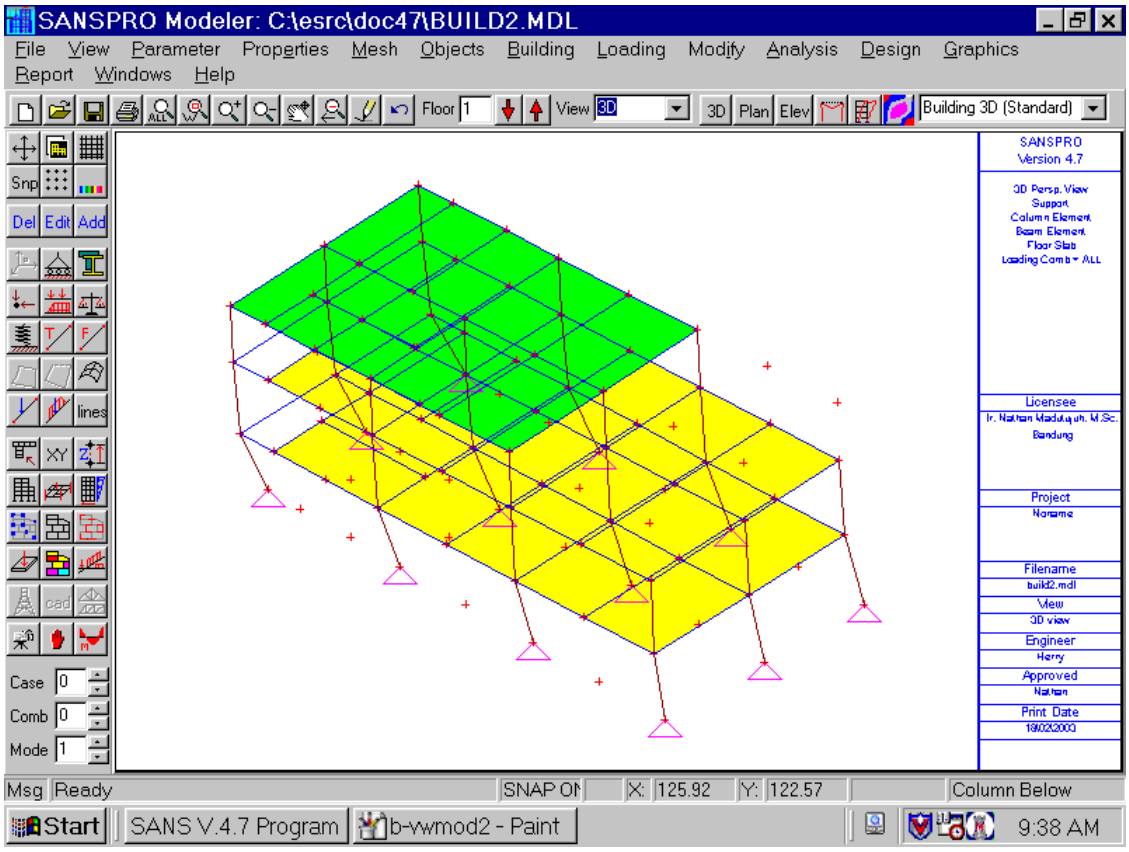
- To view mode shape:



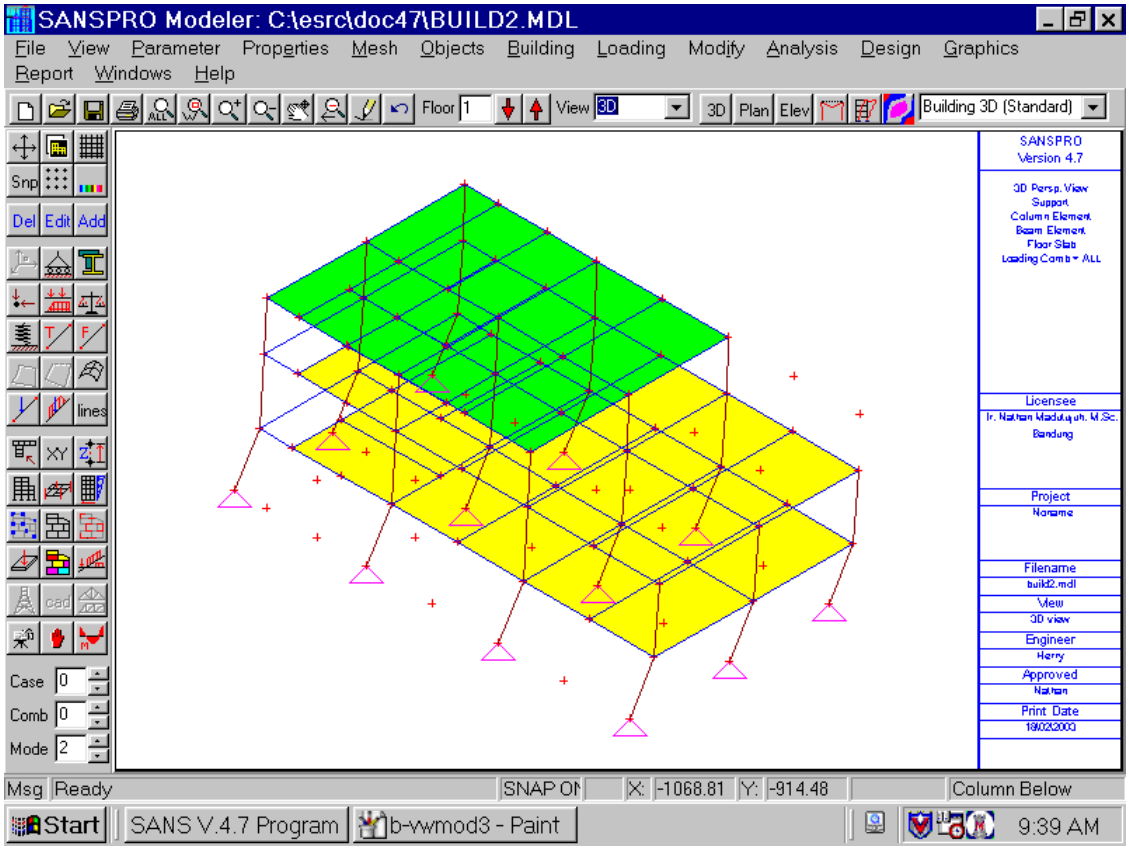
- Mode shape deformation/ displacements must be scaled up, here we use 100



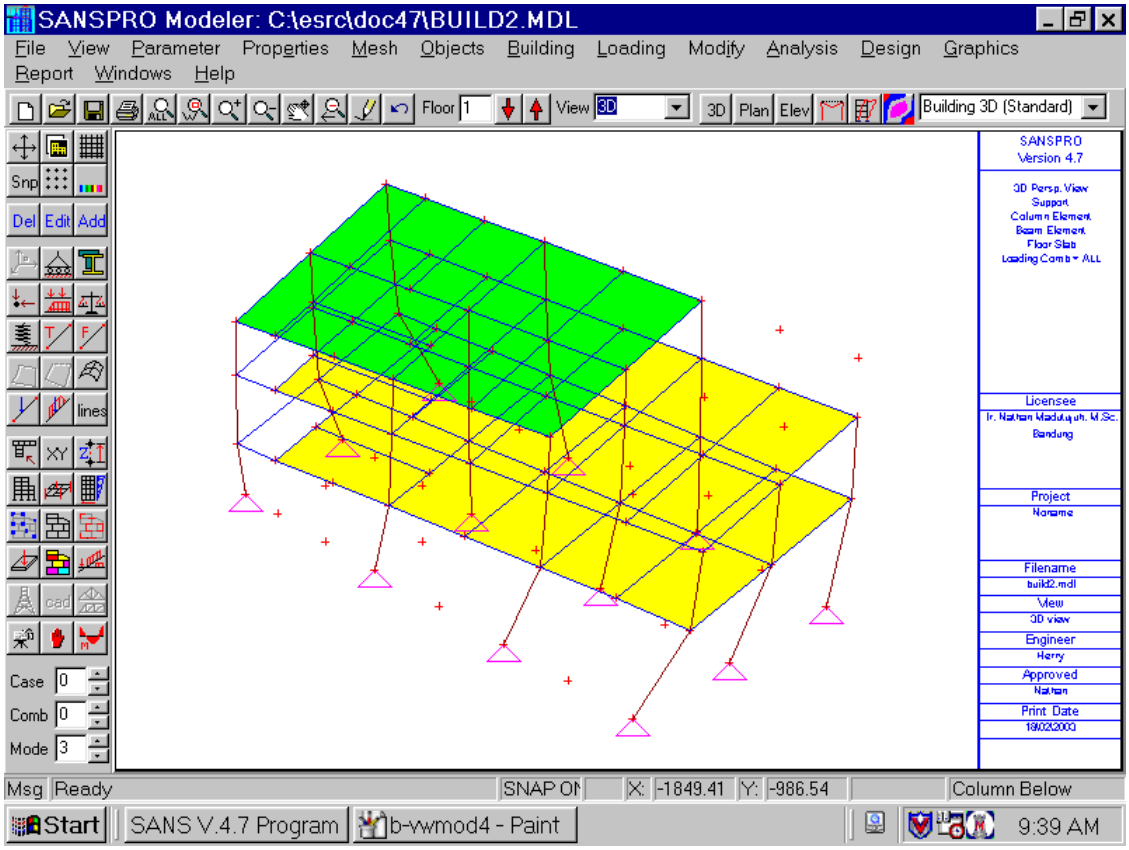
- First Mode (apparently a translational mode)



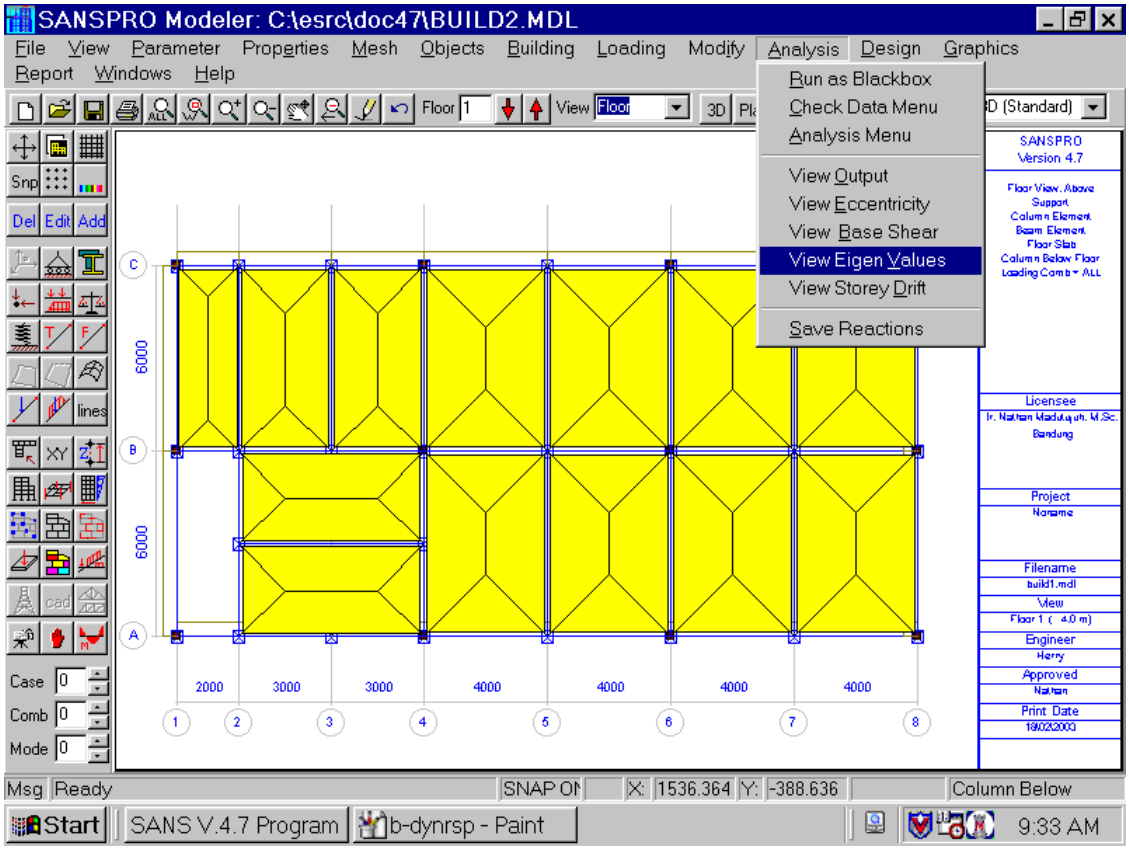
- Second Mode (apparently a translational mode)

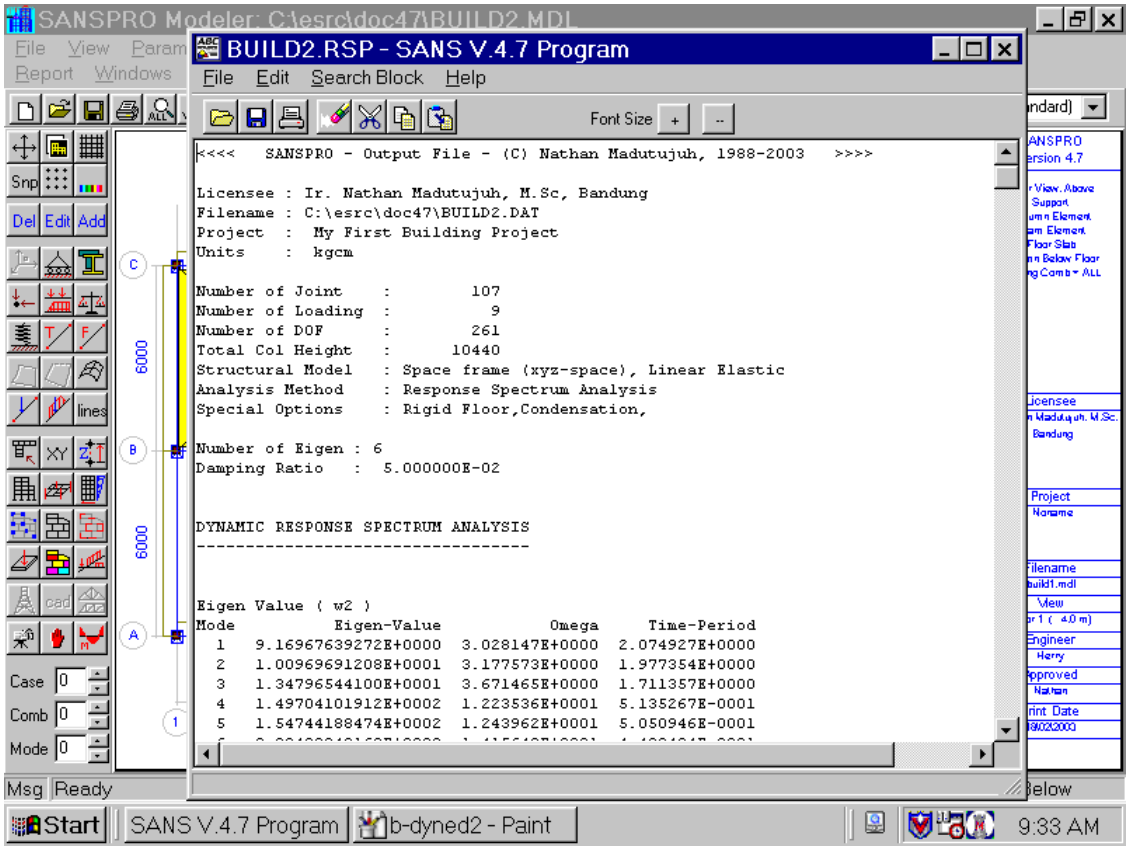


- Third Mode (apparently a rotational mode)

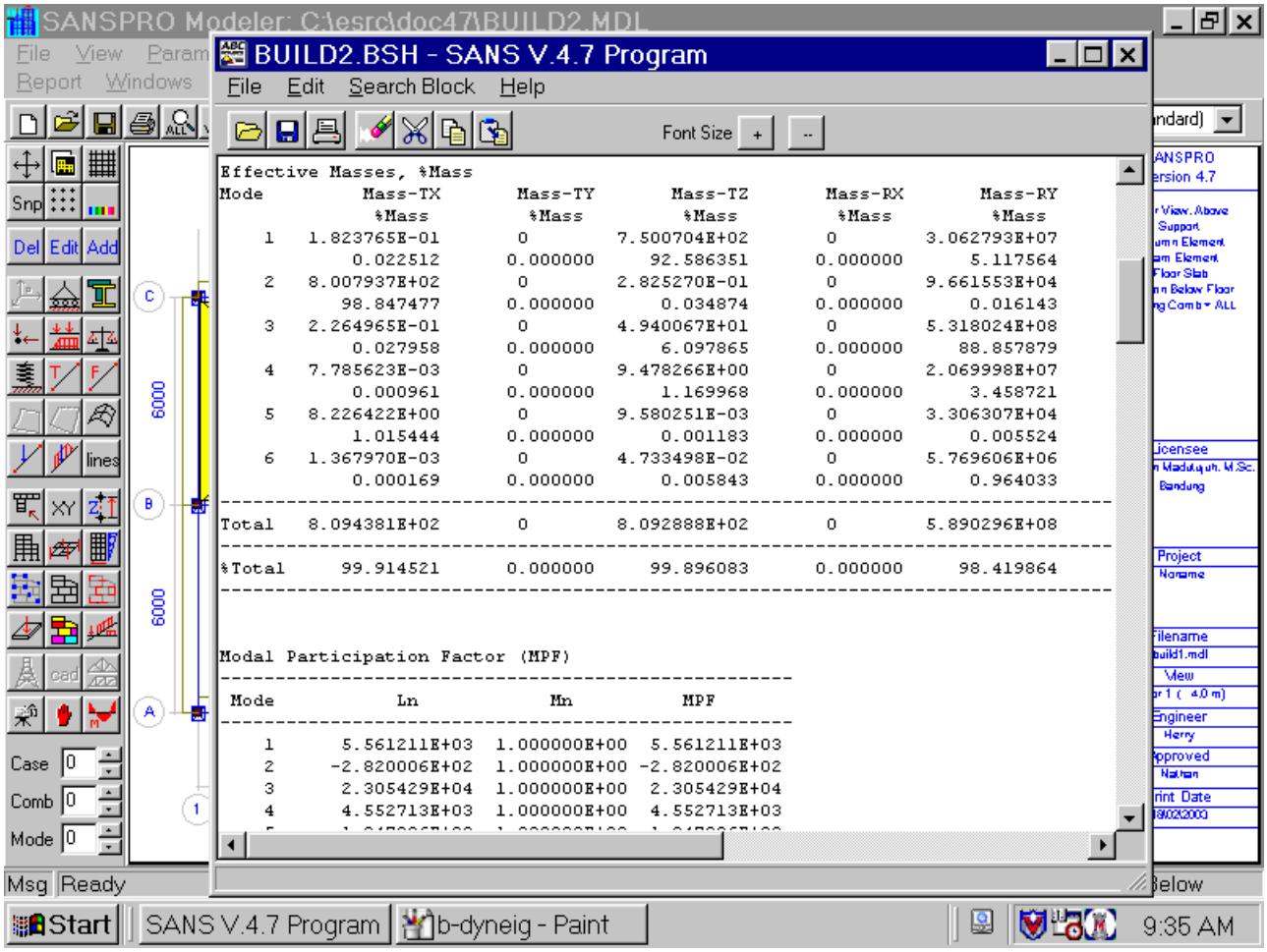


- Eigen values and Eigen Vectors

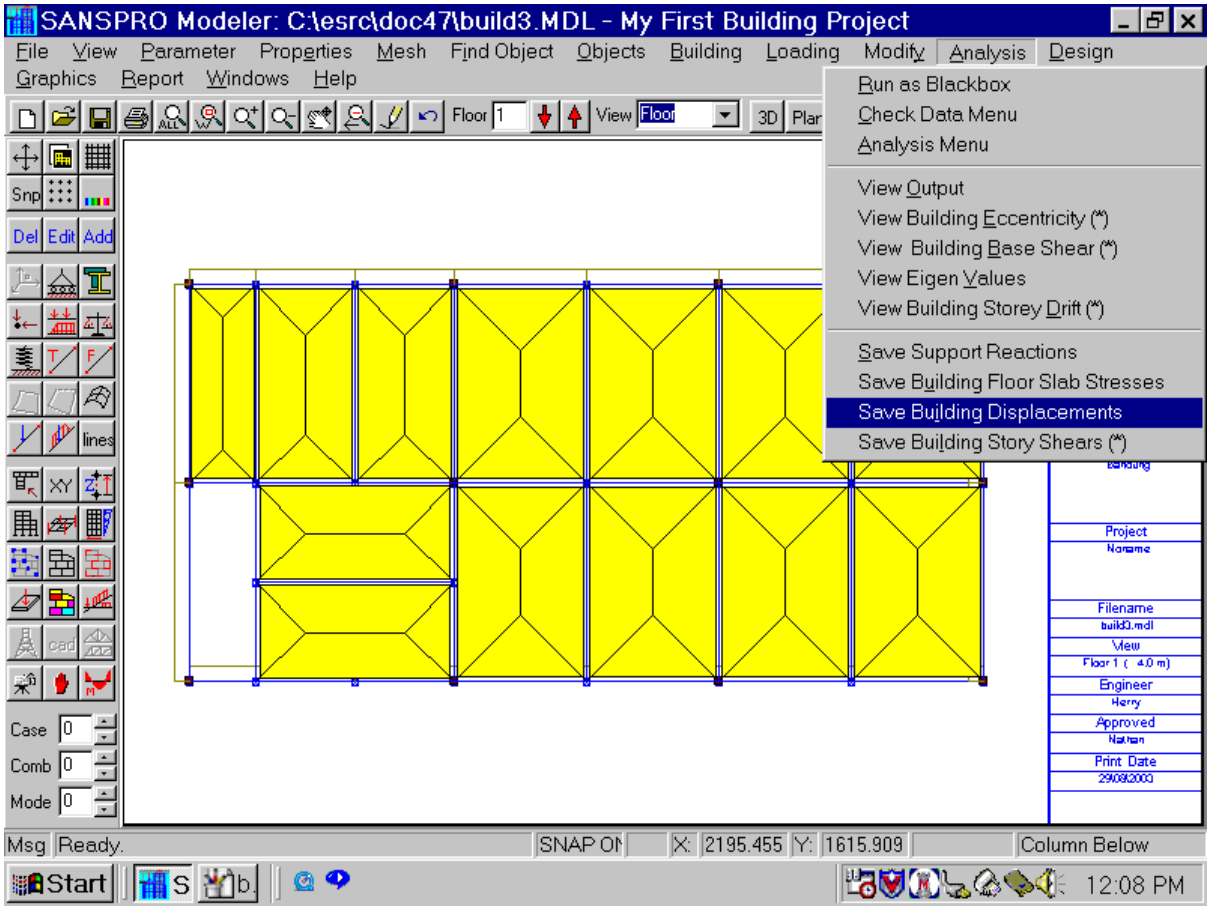


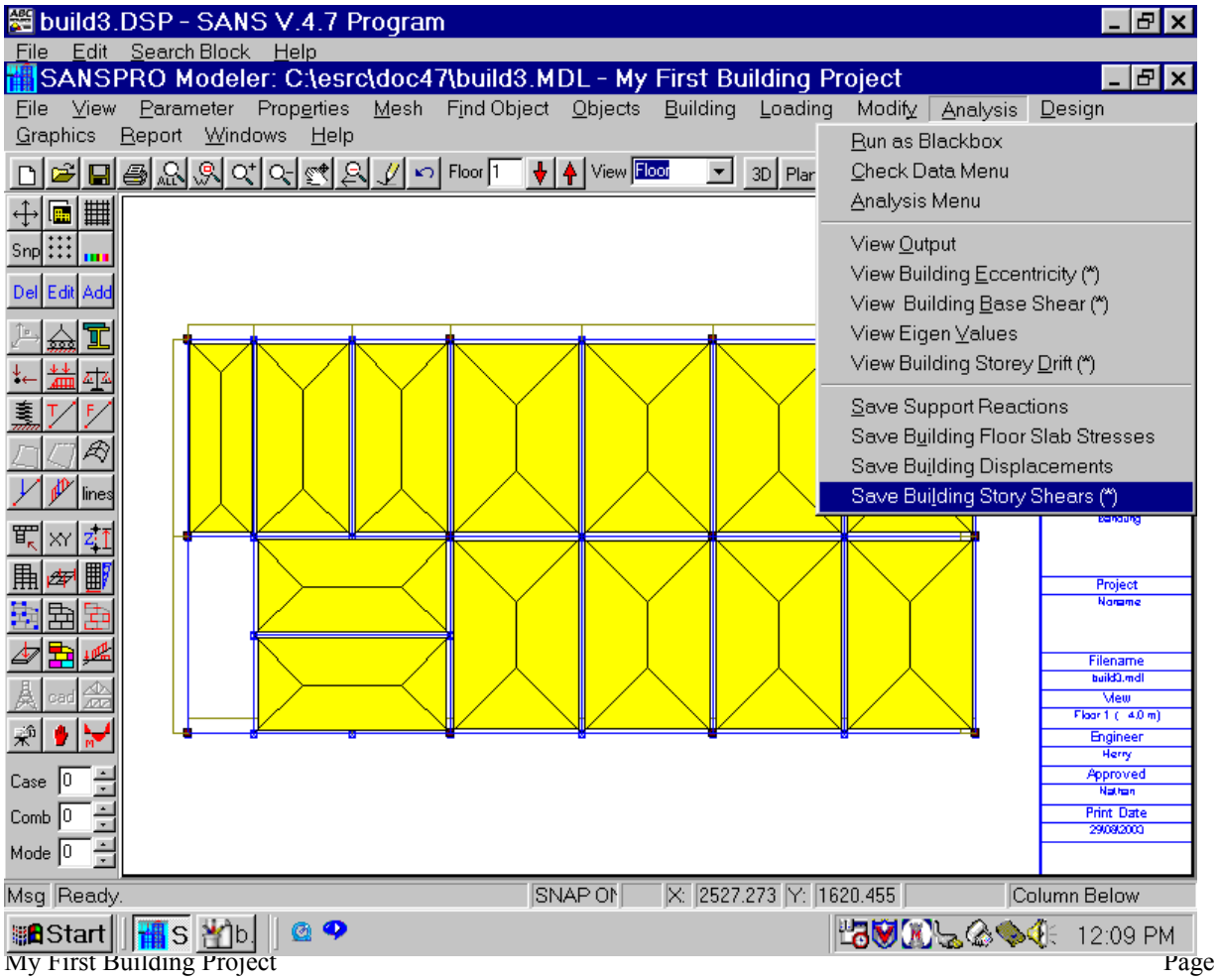


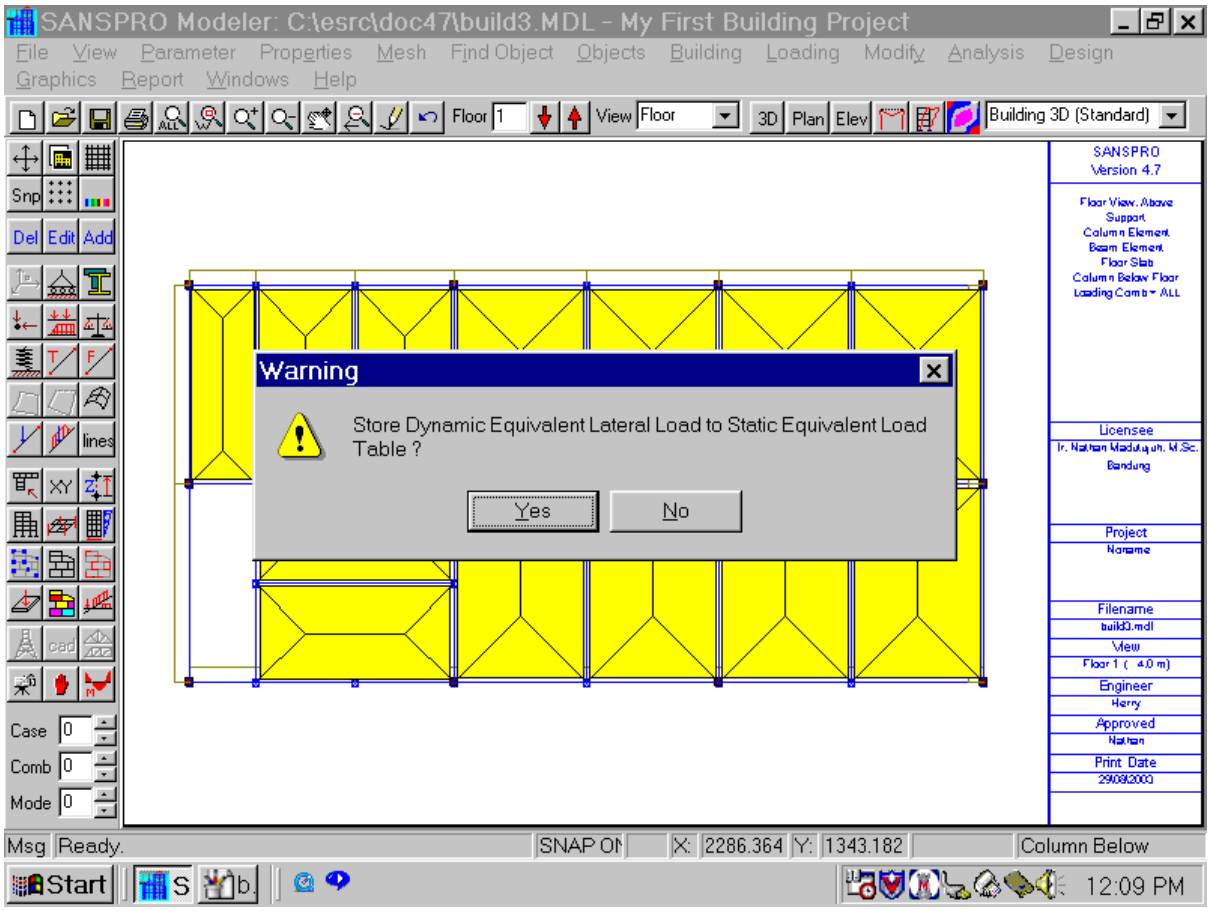
- Effective Mass and Mass Participation Factor



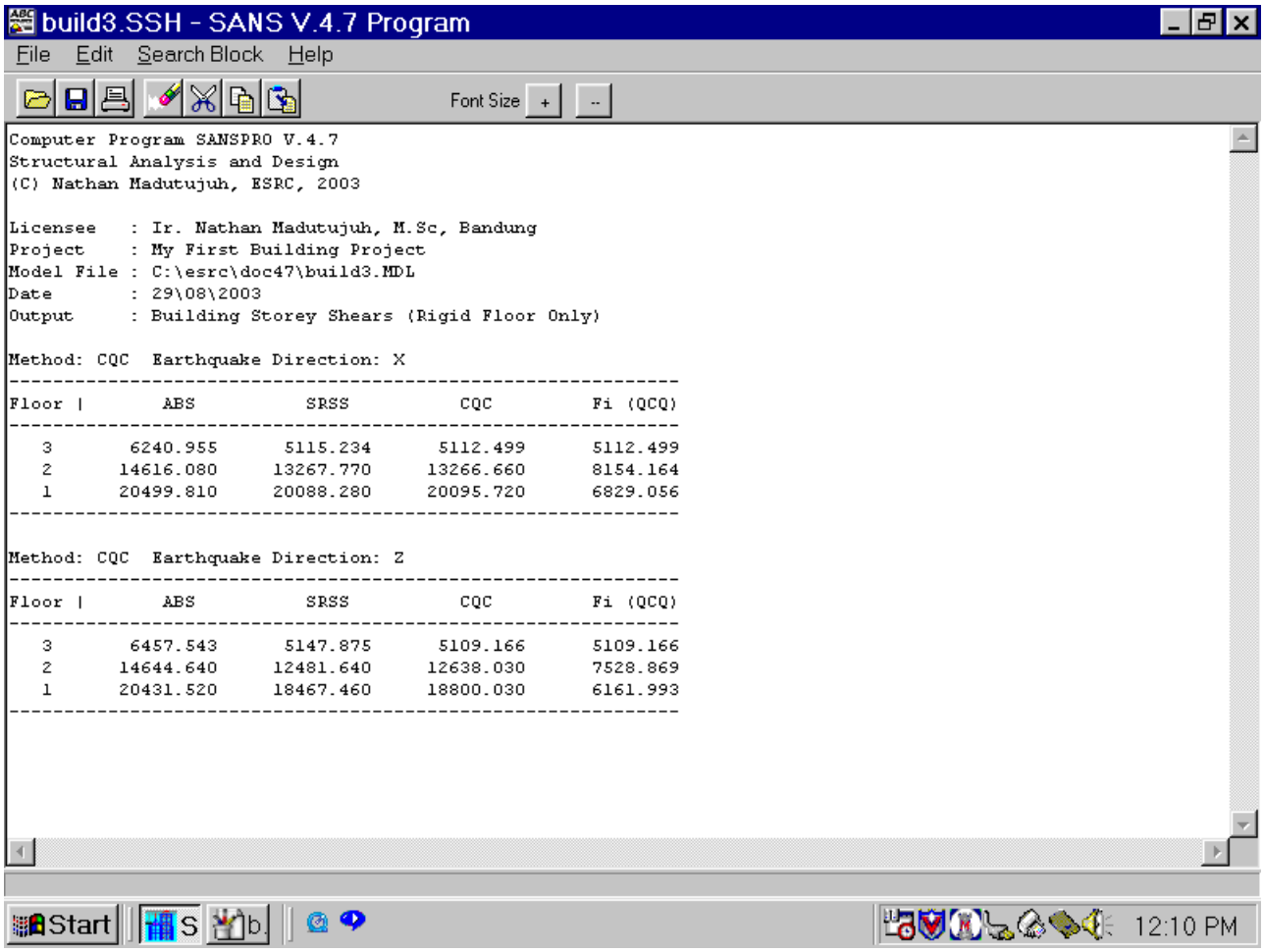
View Building Displacements/Drift






SANSPRO Modeler: C:\esrc\doc47\build3.MDL - My First Building Project
File View Parameter Properties Mesh Find Object Objects Building Loading Modify Analysis Design
Graphics Report Windows Help
Floor 1
View Floor
3D Plan Elev
Building 3D (Standard)
Snp
Del Edit Add
lines
XY
Case 0
Comb 0
Mode 0
SANSPRO
Version 4.7
Floor View, Above
Support
Column Element
Beam Element
Floor Slab
Column Below Floor
Loading Comb = ALL
Licensee
Ir. Nathan Madutujuh, M.Sc.
Bandung
Project
Noname
Filename
build3.mdl
View
Floor 1 (4.0 m)
Engineer
Harry
Approved
Nathan
Print Date
29/08/2000
Warning
Store Dynamic Equivalent Lateral Load to Static Equivalent Load Table ?
Yes
No
Msg Ready.
SNAP Of
X: 2286.364 Y: 1343.182
Column Below
Start
12:09 PM

Building Story Shear



SANSPRO will display story shear using ABS (Absolute Sum), SRSS (Square Root Sum) and CQC (Comple Quadratic Sum).

Usually values from CQC will be selected.
If Damping = 0 then SRSS and CQC will give equal values.

Fi is computed as difference between two stories.

6. Eccentricity

Floor Eccentricity should not be larger than the code requirements. If so, existing eccentricities should be magnified by scaling factor to get design eccentricity.
There are at least 5 conditions that will give 5 sets of output.

1. No magnification
2. Magnify Ex, Magnify Ey
3. Magnify Ex, Reduce Ey
4. Reduce Ex, Magnify Ey
5. Reduce Ex, Reduce Ey
6. Reduce Ex,Ey if negative, Enlarge if positive

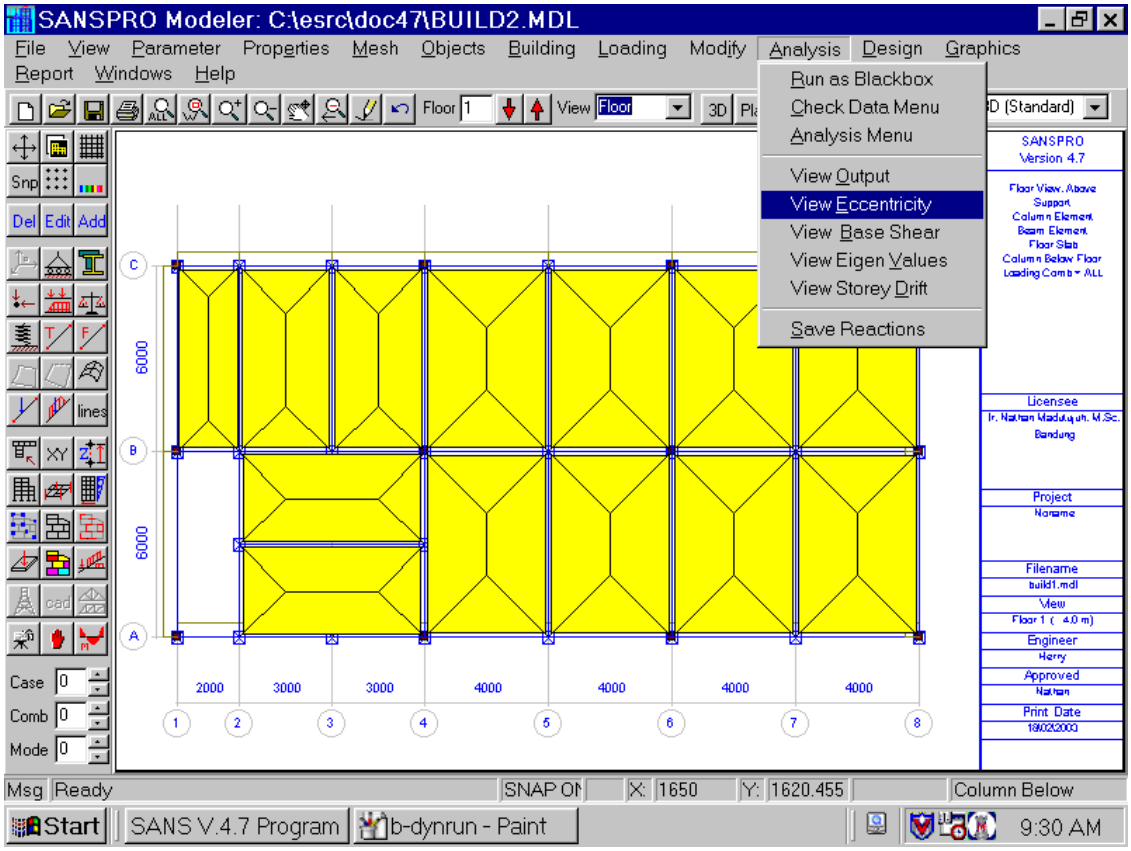
Correction of Eccentricity is done by shifting the floor center of mass. SANSPRO can do this automatically by selecting the magnification option at Building Storey Data and at Export Dialog.

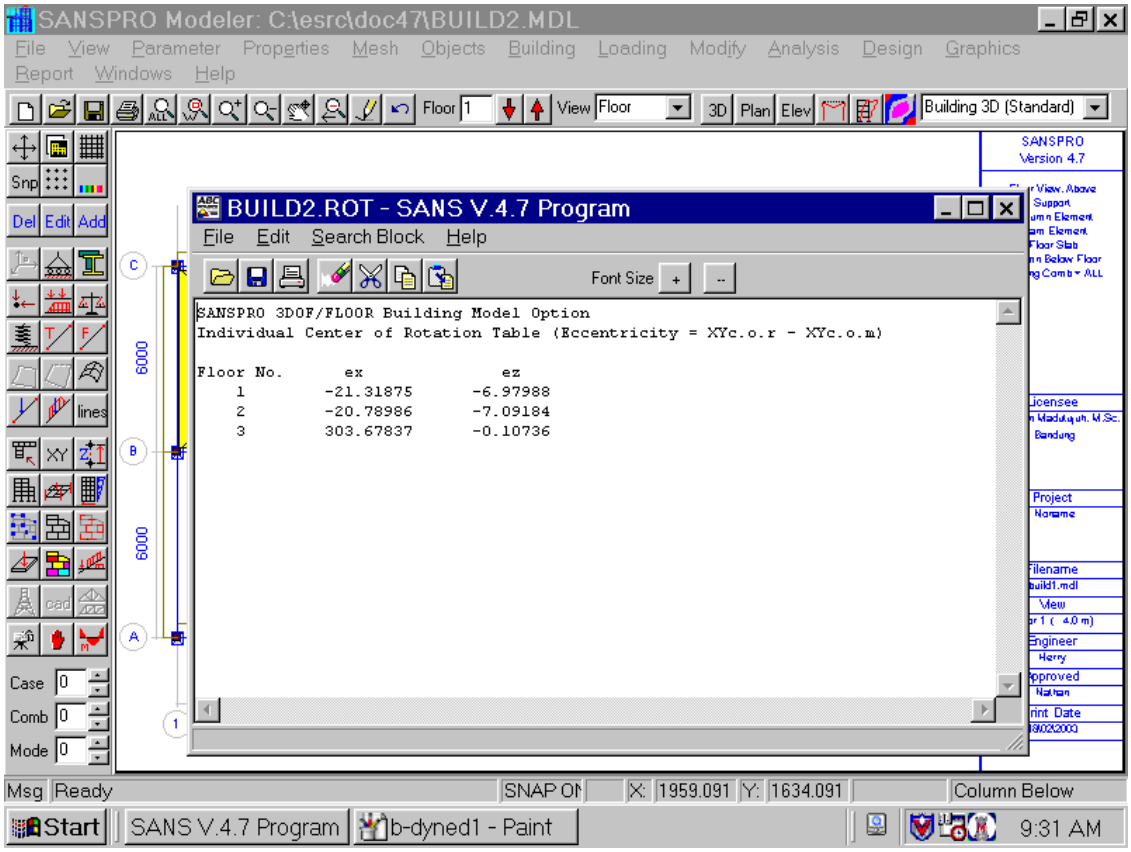
The original Center of Mass can be restored by unchecking the magnification option.

Currently, user must check one by one the separate output files from each magnification option, to get the maximum response.

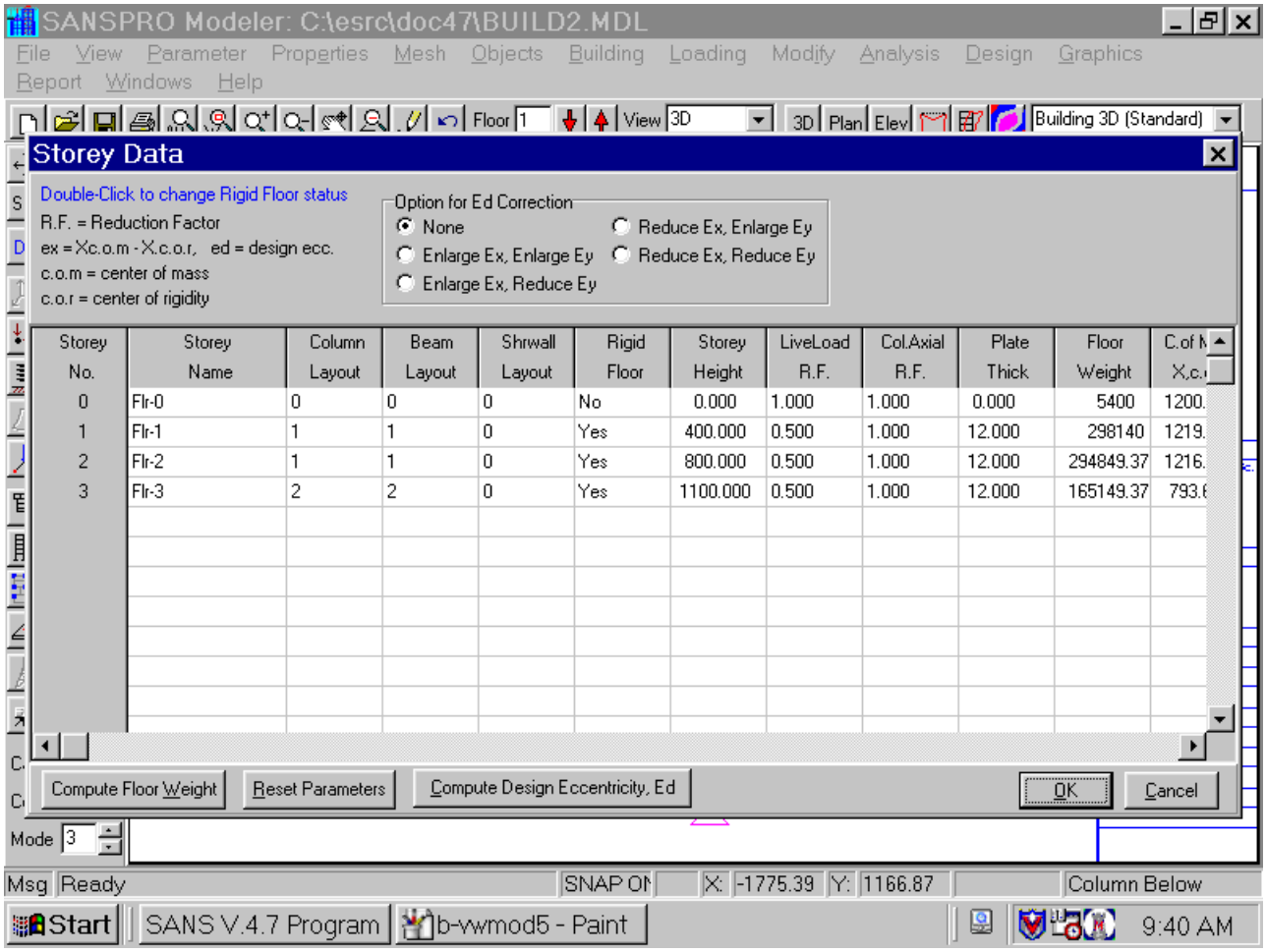
Alternatively, user can check option 6, and get the maximum design eccentricity.

- Existing Eccentricity (file BUILD2.ROT)

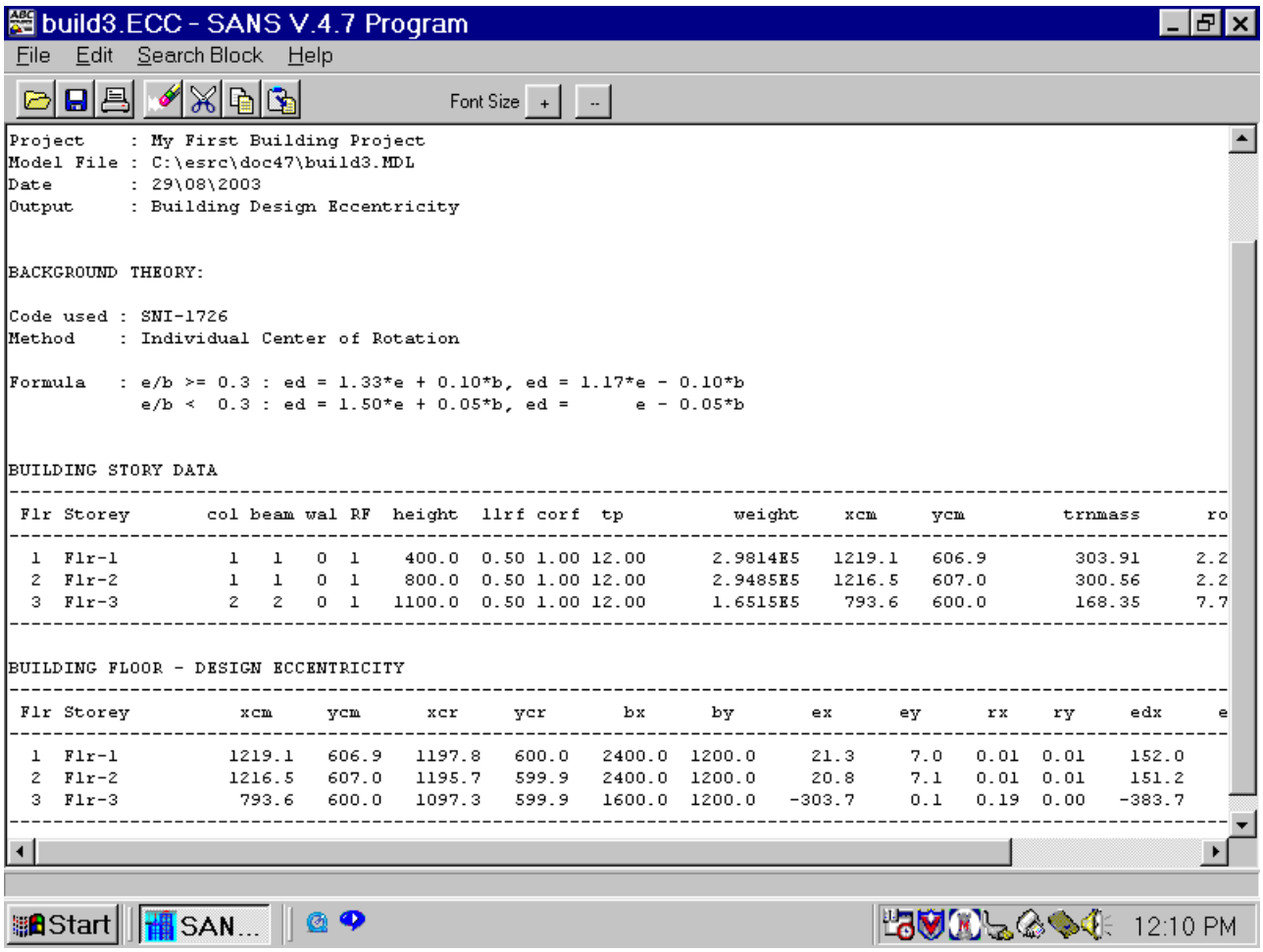




- Design Eccentricity can be calculated after first run because it needs file BUILD2.ROT

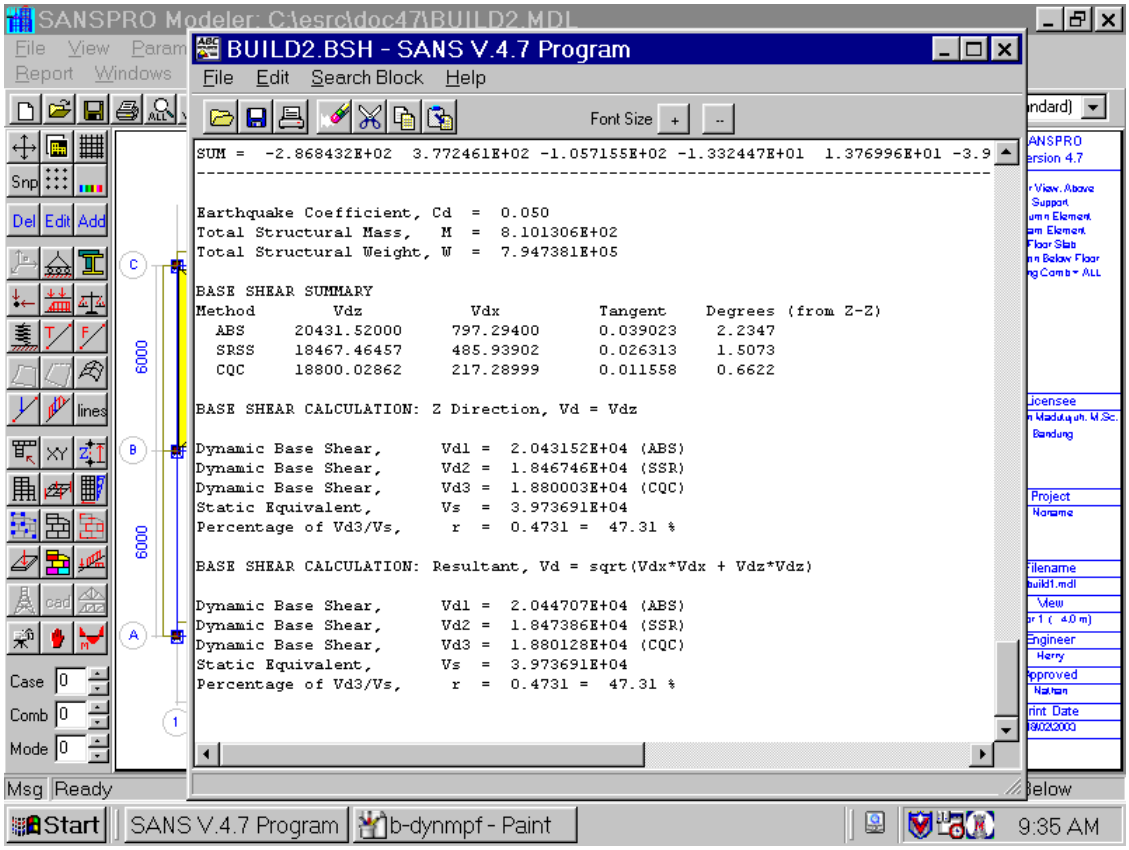


Computed Design Eccentricity using Abolute Maximum:

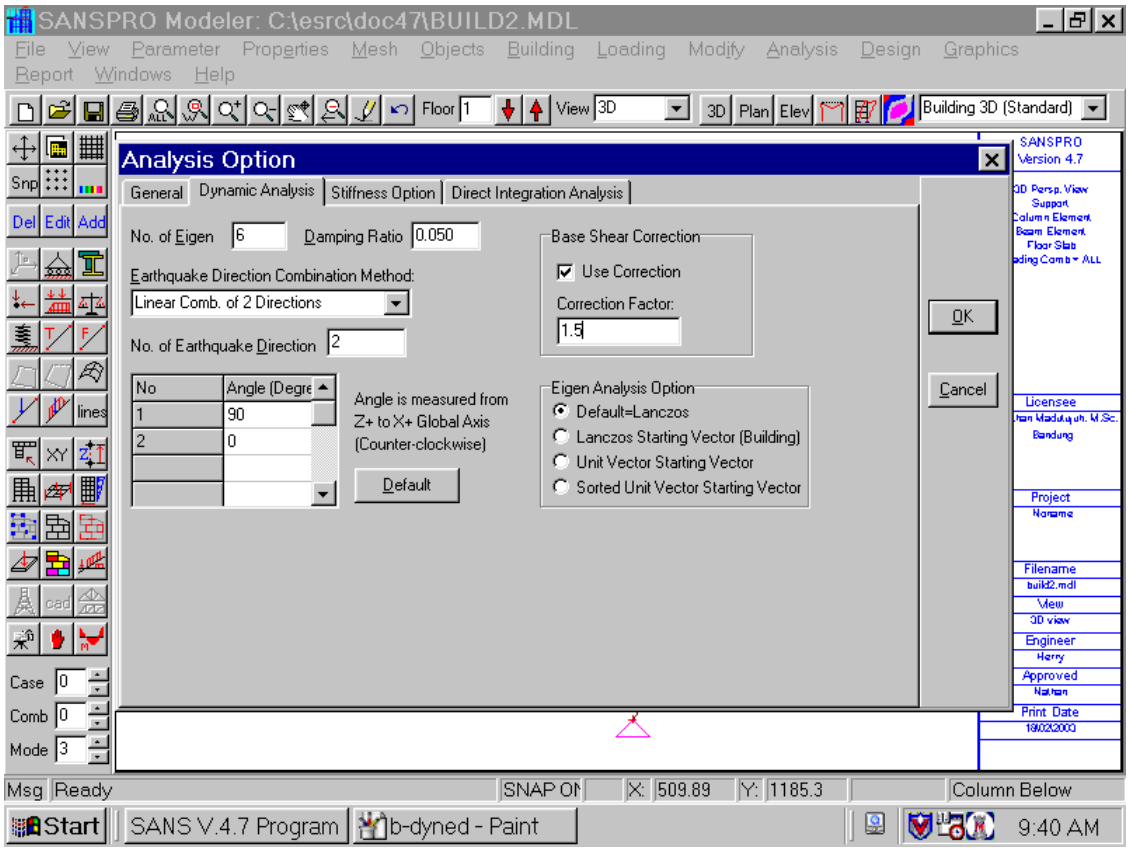


7. Base Shear

- Base Shear can be seen from file BUILD2.BSH



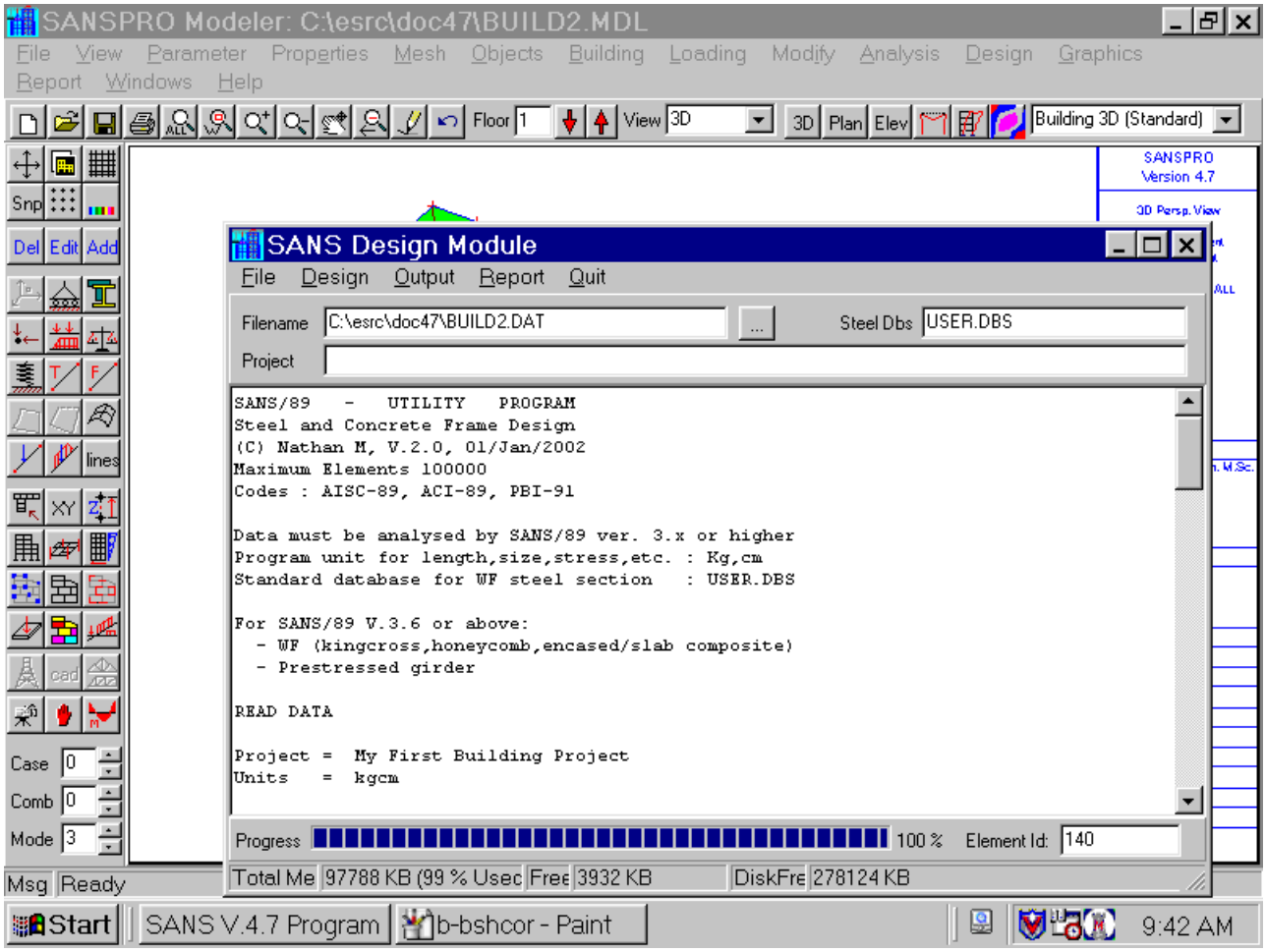
- Base Shear Correction can be done from Analysis Option:



8. Design for Dynamic Analysis

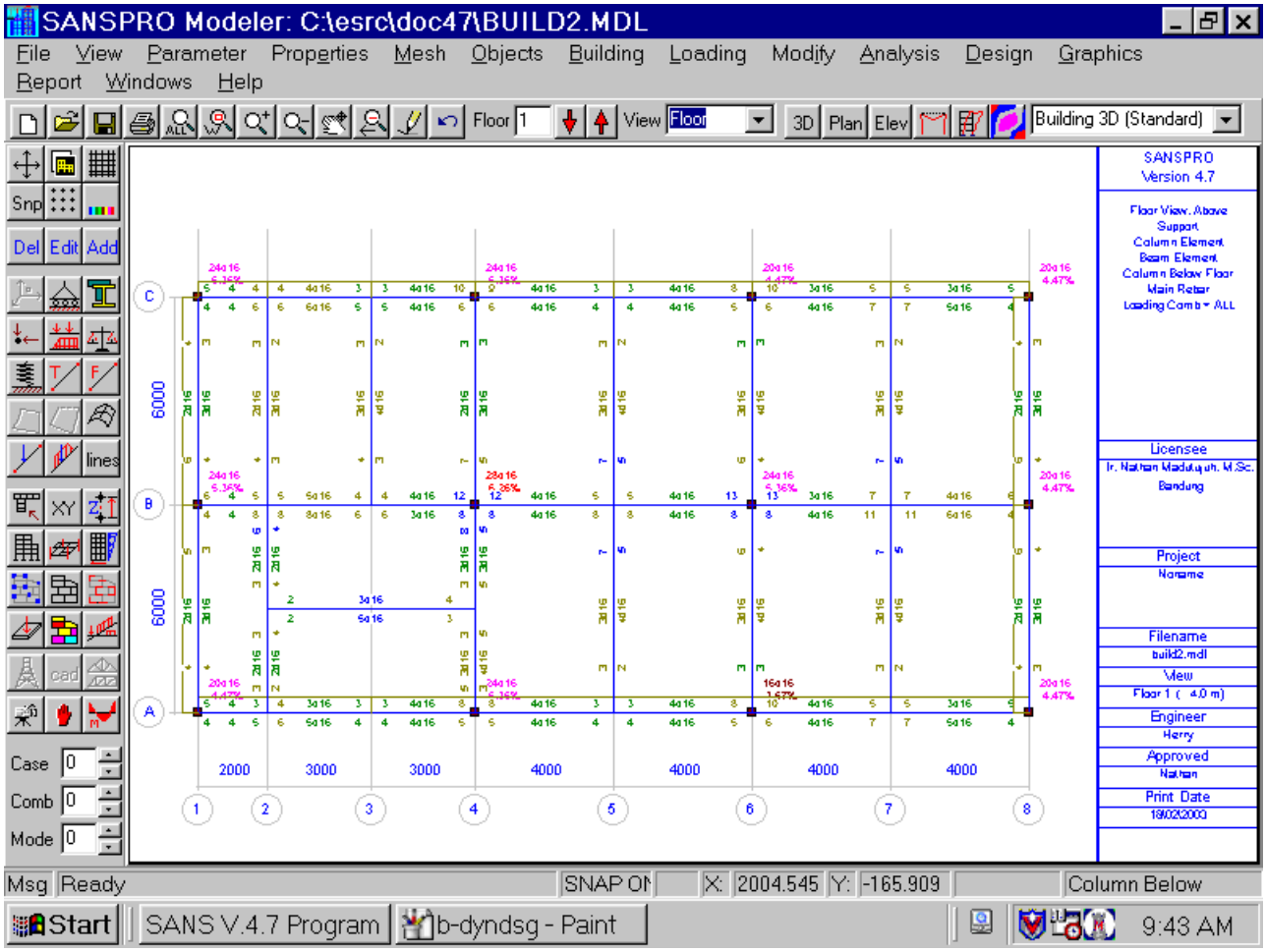
Element Design process for Response Spectrum Dynamic Analysis is exactly the same as for static analysis.

Click Design – Truss/Frame Design – Concrete Frame
Right-Click – View Design Results to view the design as in static analysis



9. Visual Design Report

- Right-Click – View Design Results to view the design as in static analysis
- Move up/down floor to see other floors
- Select different load combinations too see result from different load combinations



Capacity Design

1. Basic Concepts

SANSPRO uses Capacity Design Concepts to design Seismic Resistant Concrete Buildings.

Design a building to be able to resist full earthquake load will give a very strong structure and also not economical design. To get an economical solution, the capacity of the building to absorb earthquake energy should be increased, so that the building will be able to resist large earthquake in certain amount of time before total failure. The time delay will be used for evacuation of people and other actions.

In other words, buildings are only designed to resist moderate earthquake without severe damaged, and are allowed to be damaged by large earthquake without sudden failure.

Sudden failure usually comes from shear failure, so the design shear strength should be larger than the capacity of bending strength.

Capacity Design assumes that most of the earthquake energy will be absorbed by plastic hinges due to bending forces, so it requires that the structure dominant vibration mode should be in translational mode. If not so, user should design manually the capacity of the elements using other methods.

Currently SANSPRO can deal with full ductile design only.

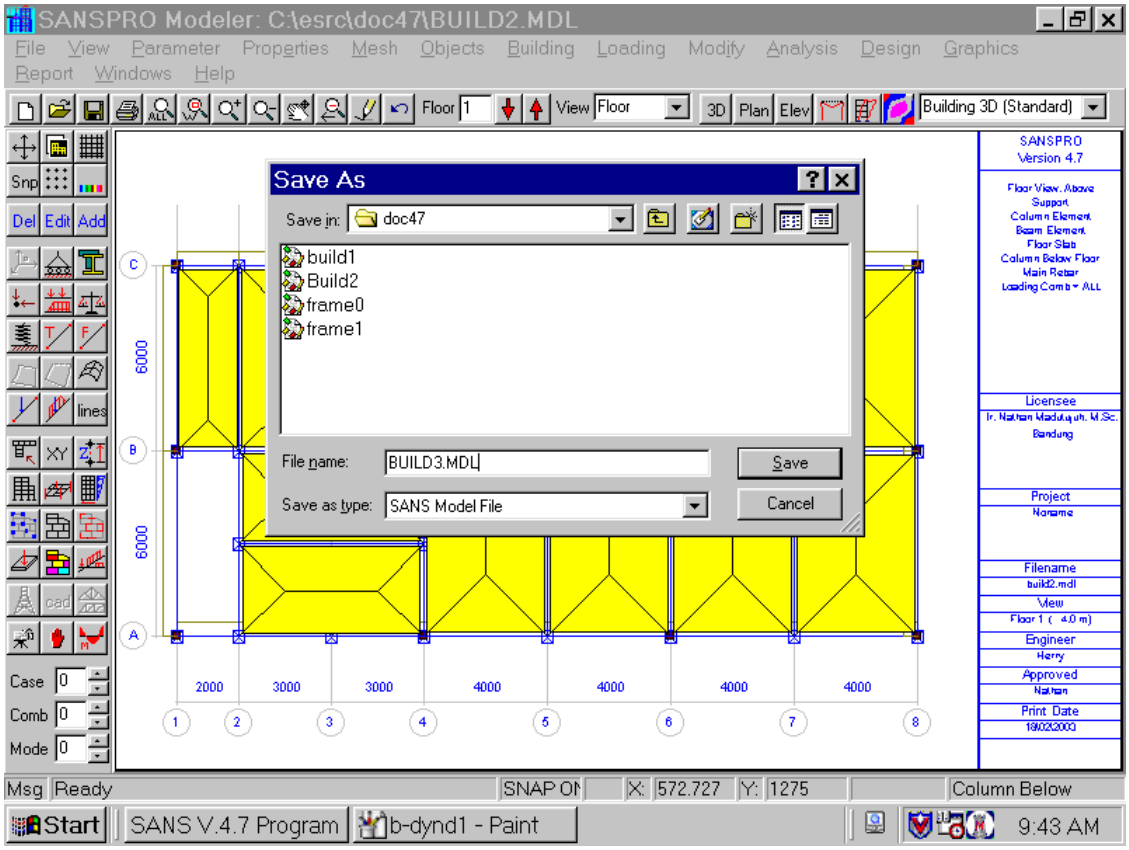
SANSPRO Capacity Design Procedures:

1. Design Beam Rebar using Factored Load
2. Calculate Beam Bending Capacity for each direction of column face
3. Compute Beam Shear from Beam Bending Capacity
4. Compute Additional Column Axial from Beam Moment Capacity
5. Compute Column Design Bending Forces
6. Compute Column Bending Rebar
7. Compute Column Bending Capacity
8. Compute Column Shear from Column Bending Capacity

Output of SANSPRO Capacity Design (Text Files):

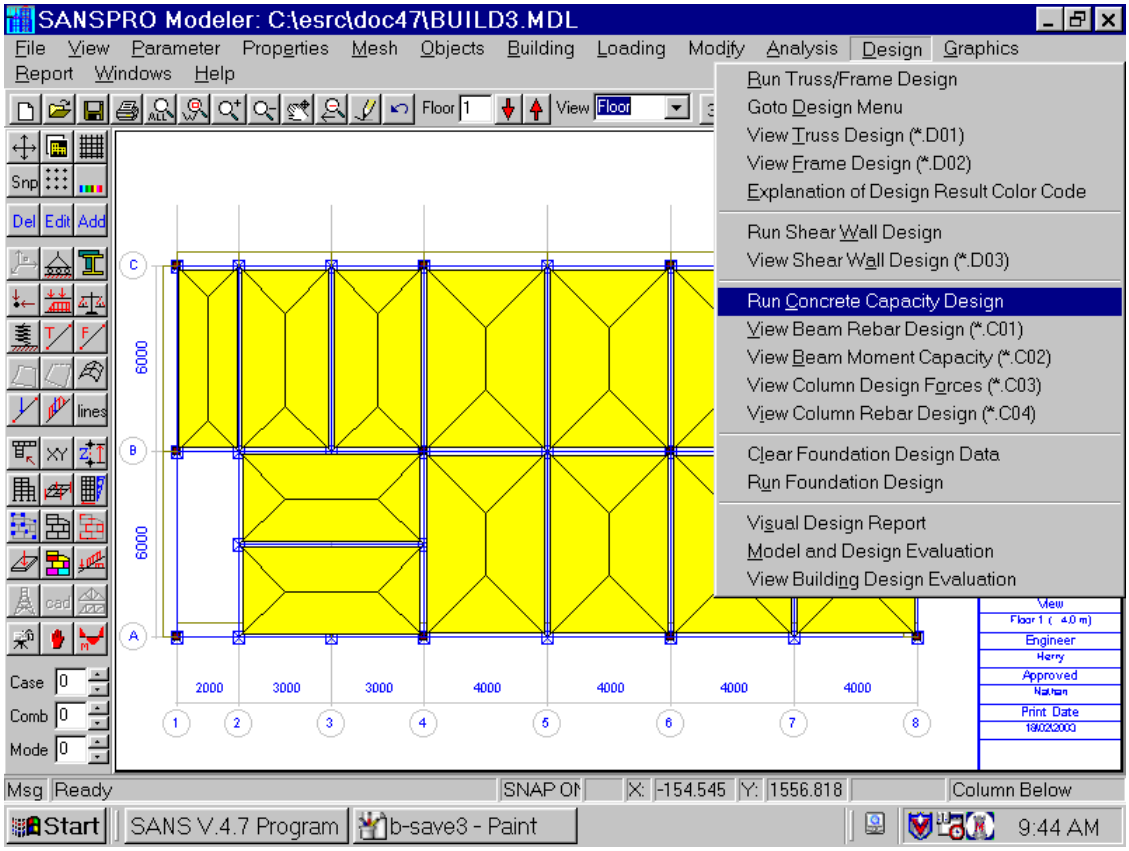
Beam Rebar Design	BUILD3.C01
Beam Moment Capacity	BUILD3.C02
Column Design Forces	BUILD3.C03
Column Rebar Design	BUILD3.C04

- To get incremental design results, save the current file to new file : BUILD3.MDL



2. Define Parameters

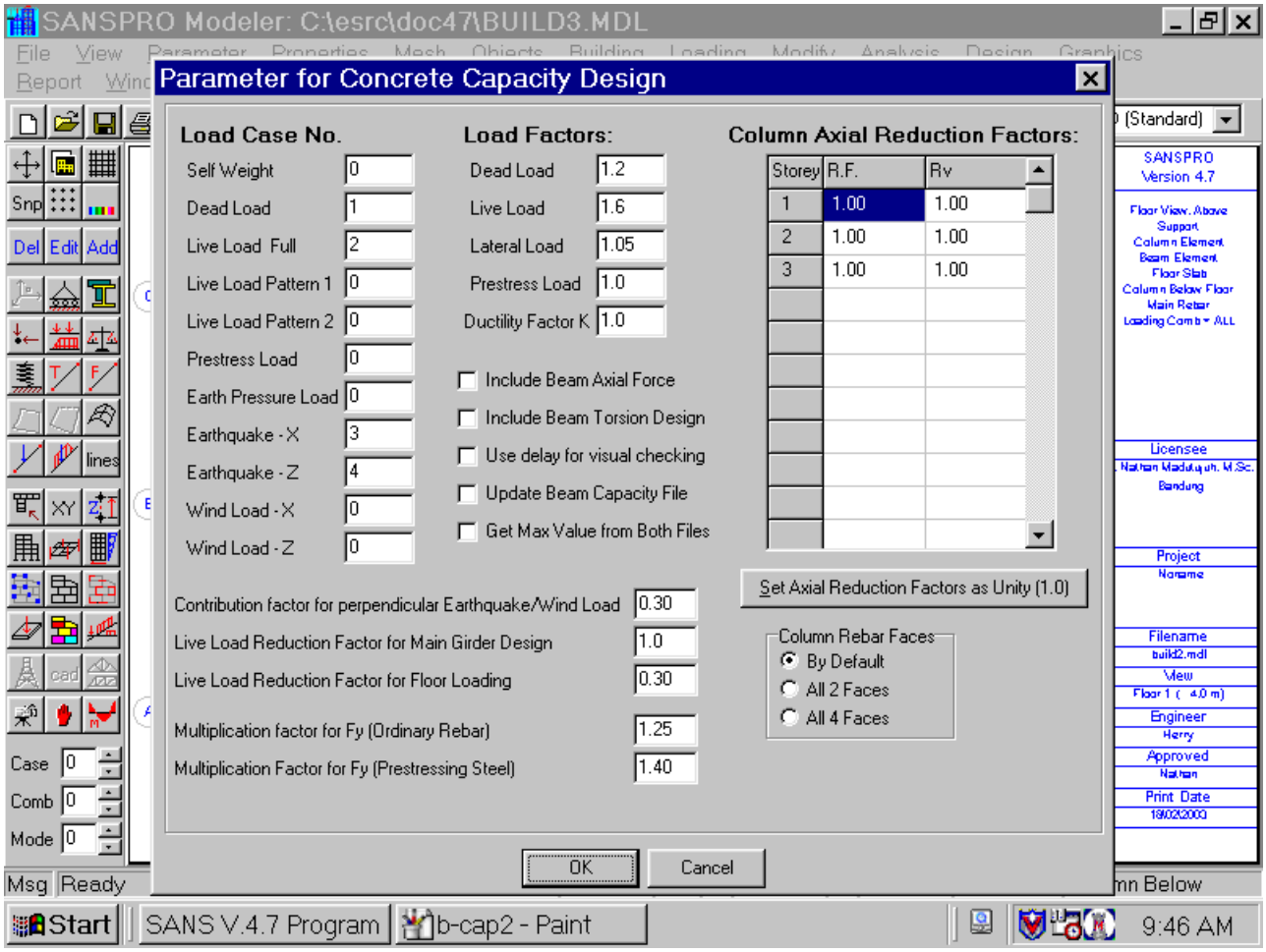
- SANSPRO Capacity Design needs several control parameters:



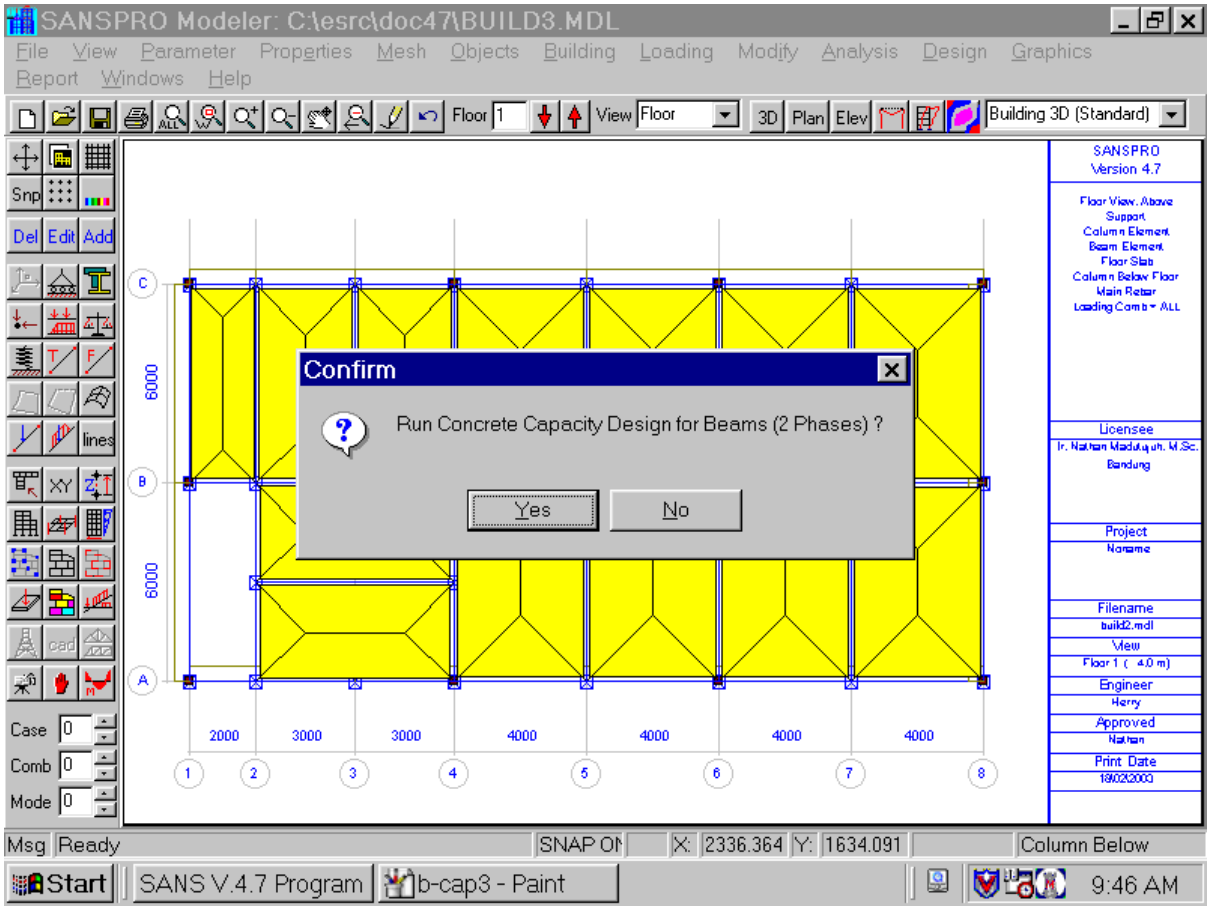
Parameters Explanation:

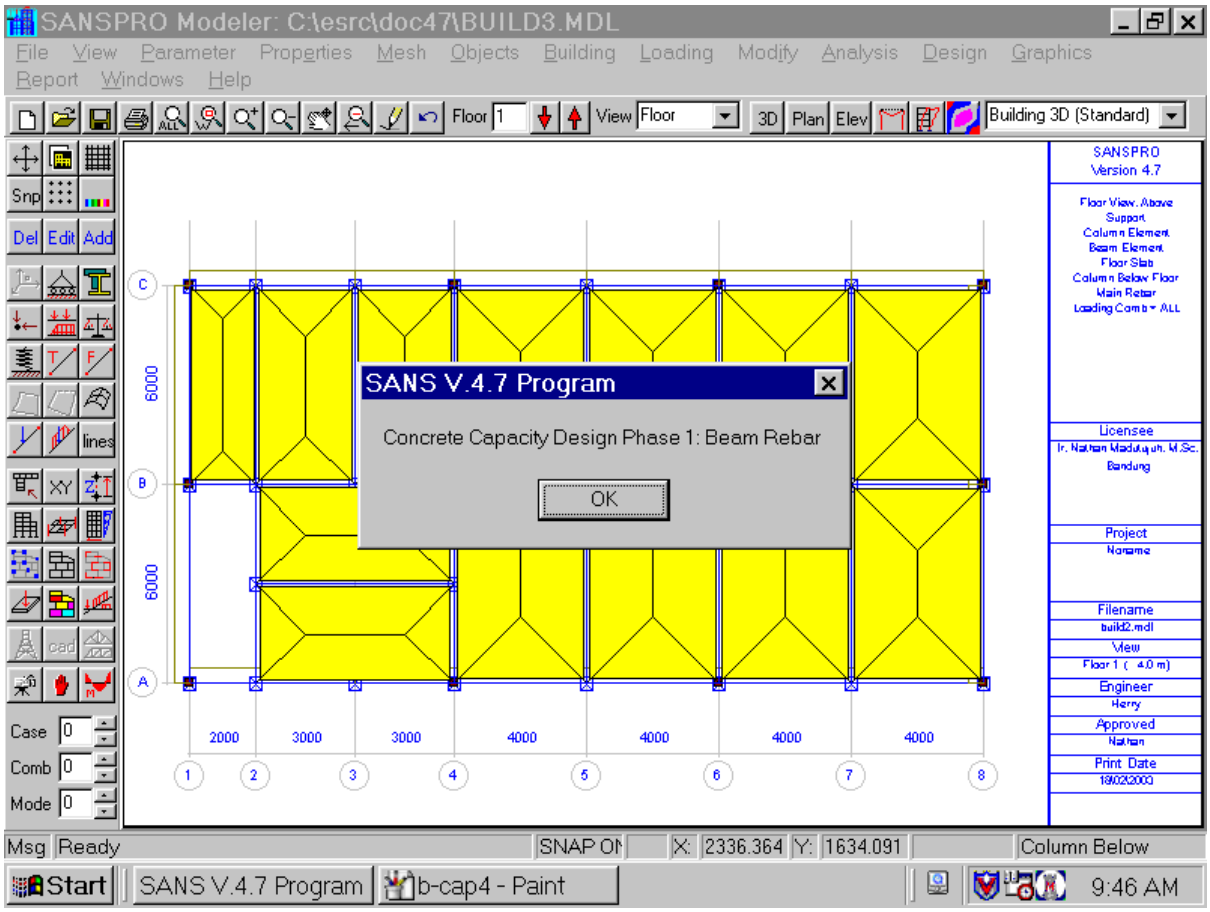
- Capacity Design Module will use its own load combination because of the use of different load factors and load reduction. Enter the load case number for dead load, live load, etc.
- Load Factors also can be entered for flexible usage.
- Contribution of perpendicular earthquake (usually 30%) should be entered
- Steel Rebar Overstrength Factor should be entered
- Column axial reduction factors for each floor should be entered
- Structural Ducktility Factor (1.0 for full ductile) should be entered
- Beam can be designed for torsional forces and also for axial forces.
- In case you have changed the Beam Moment Capacity file, please check the Update Beam Capacity File flag

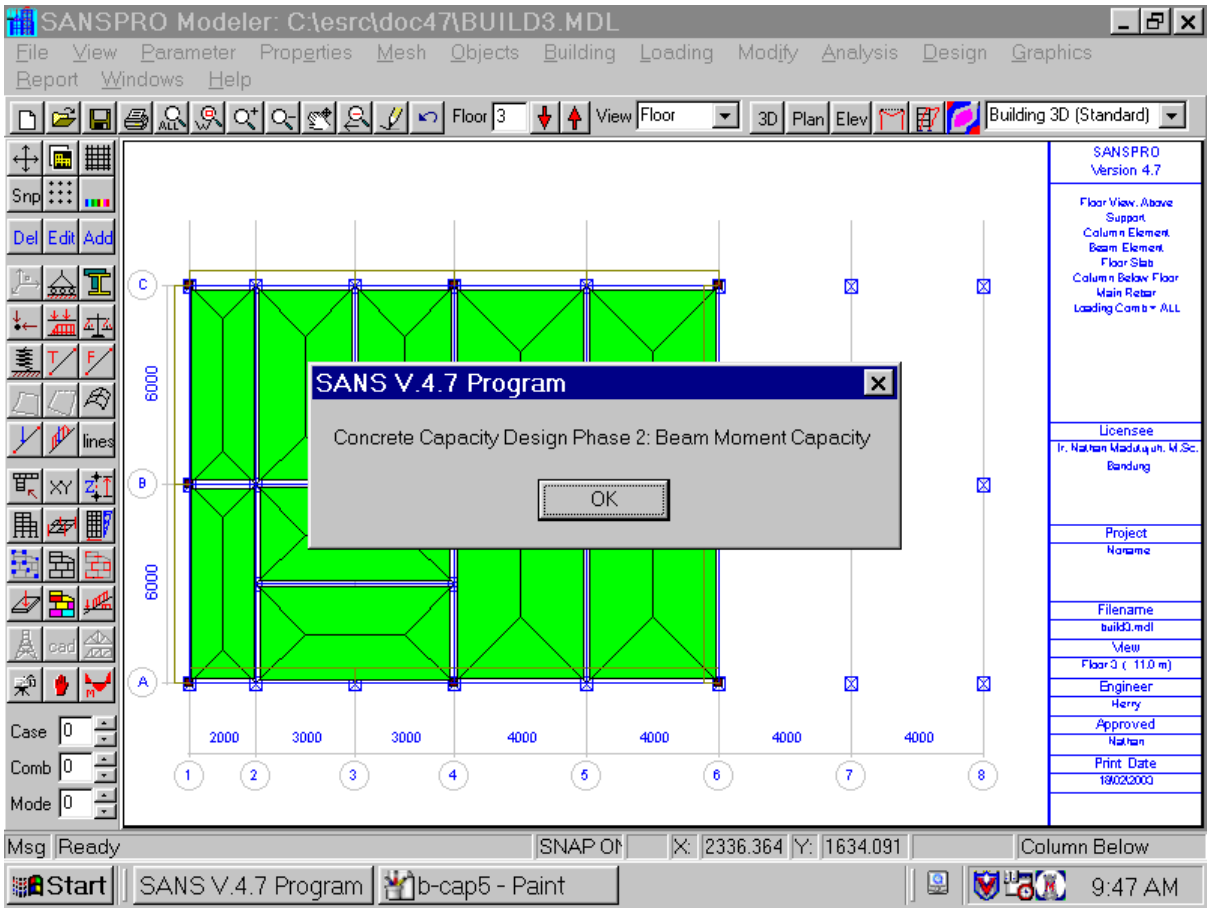
(For example, if the moment capacity of prestressed beams have been calculated manually or outside the SANSPRO, the values can be entered into file BUILD3.C02 to be used for column capacity design)

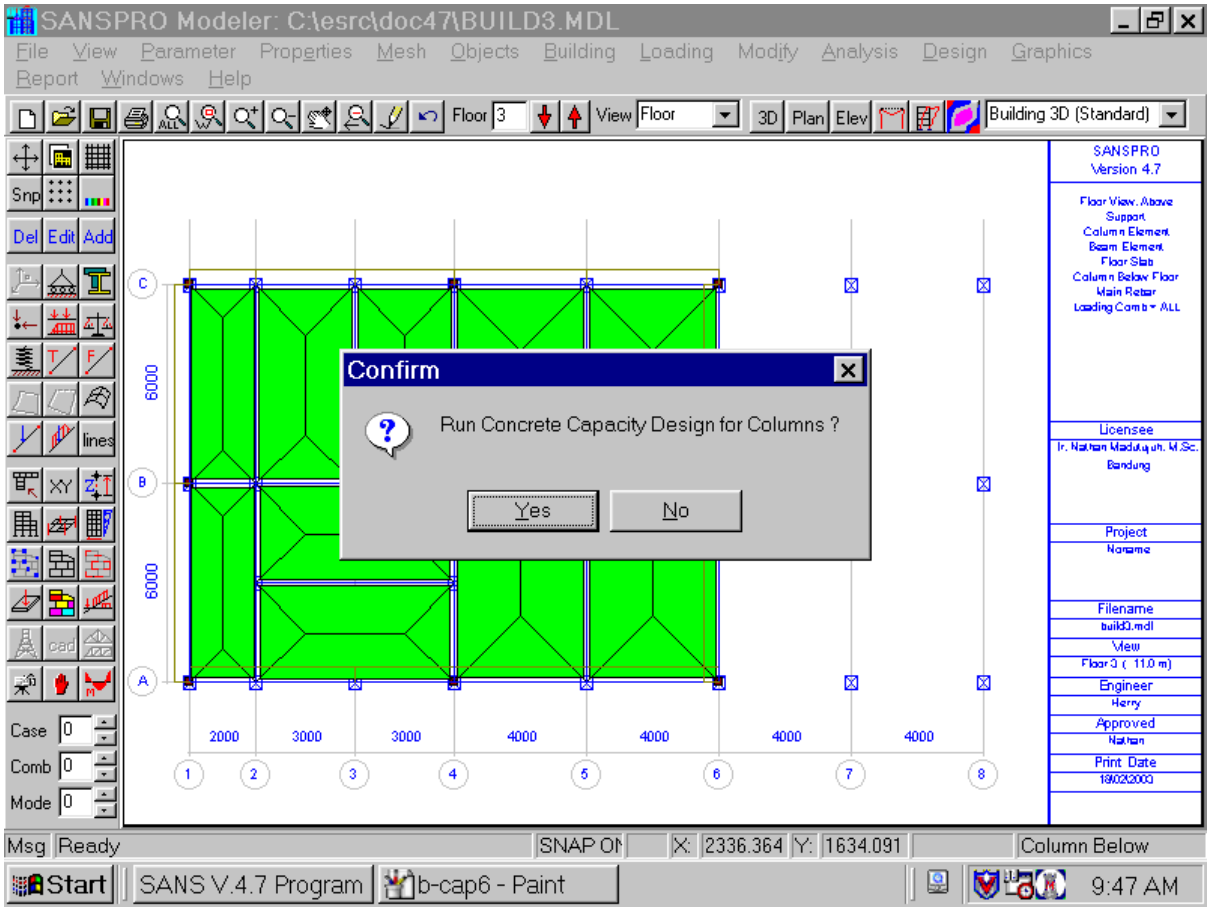


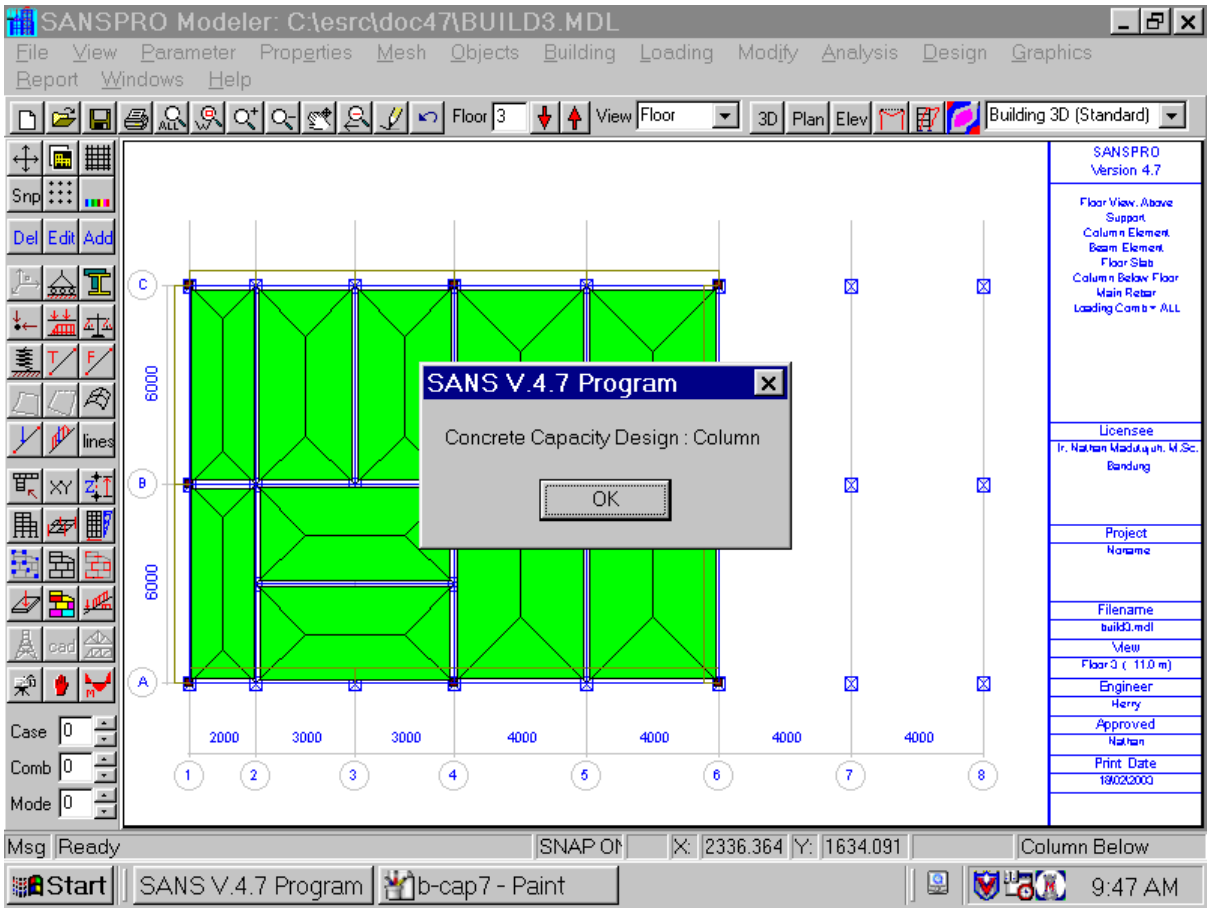
3. Run Capacity Design





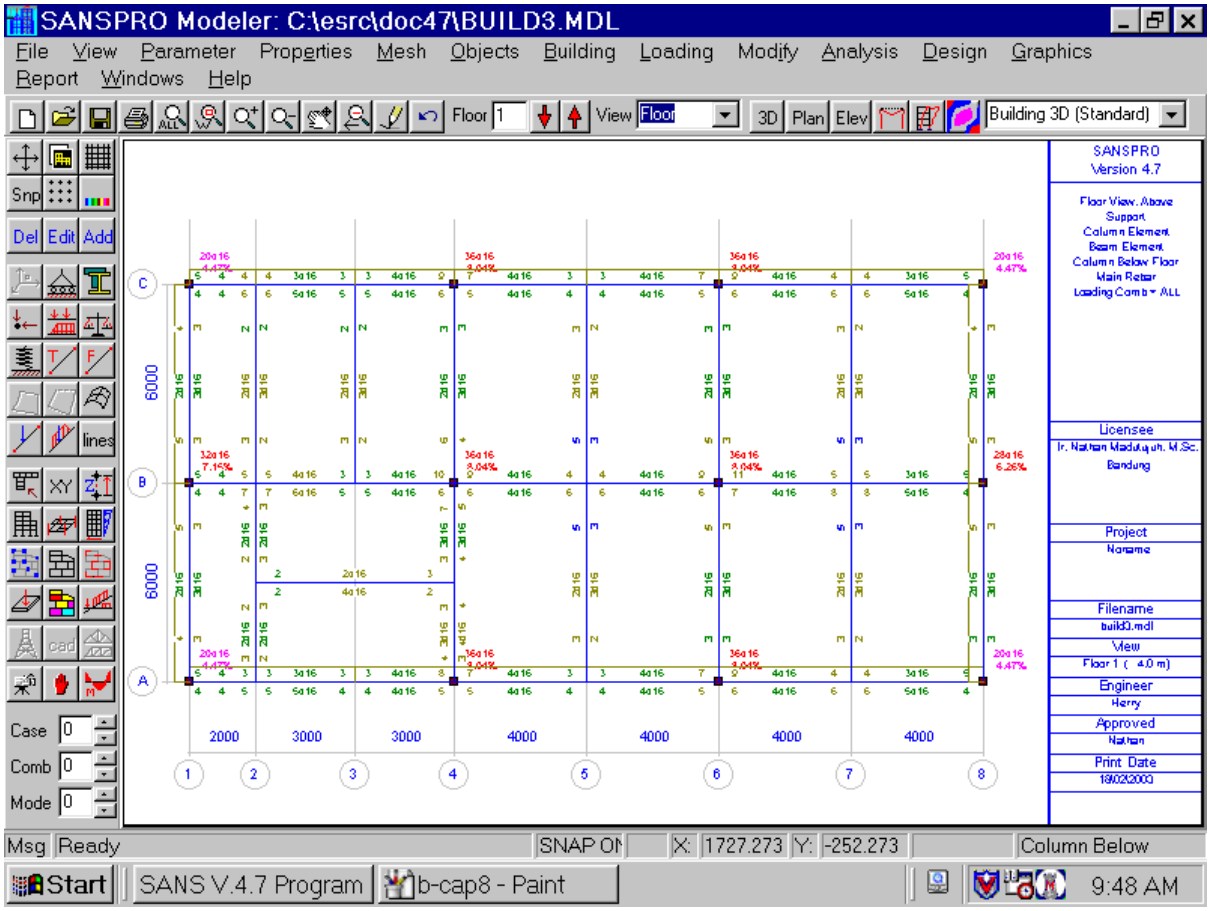

SANSPRO Modeler: C:\esrc\doc47\BUILD3.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
SANS V.4.7 Program
Concrete Capacity Design Phase 2: Beam Moment Capacity
OK
SANSPRO Version 4.7
Msg Ready
Column Below
Start
SANS V.4.7 Program
b-cap5 - Paint
9:47 AM


SANSPRO Modeler: C:\esrc\doc47\BUILD3.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
Confirm
Run Concrete Capacity Design for Columns ?
Yes
No
SANSPRO Version 4.7
Msg Ready
Column Below
Start
SANS V.4.7 Program
b-cap6 - Paint
9:47 AM

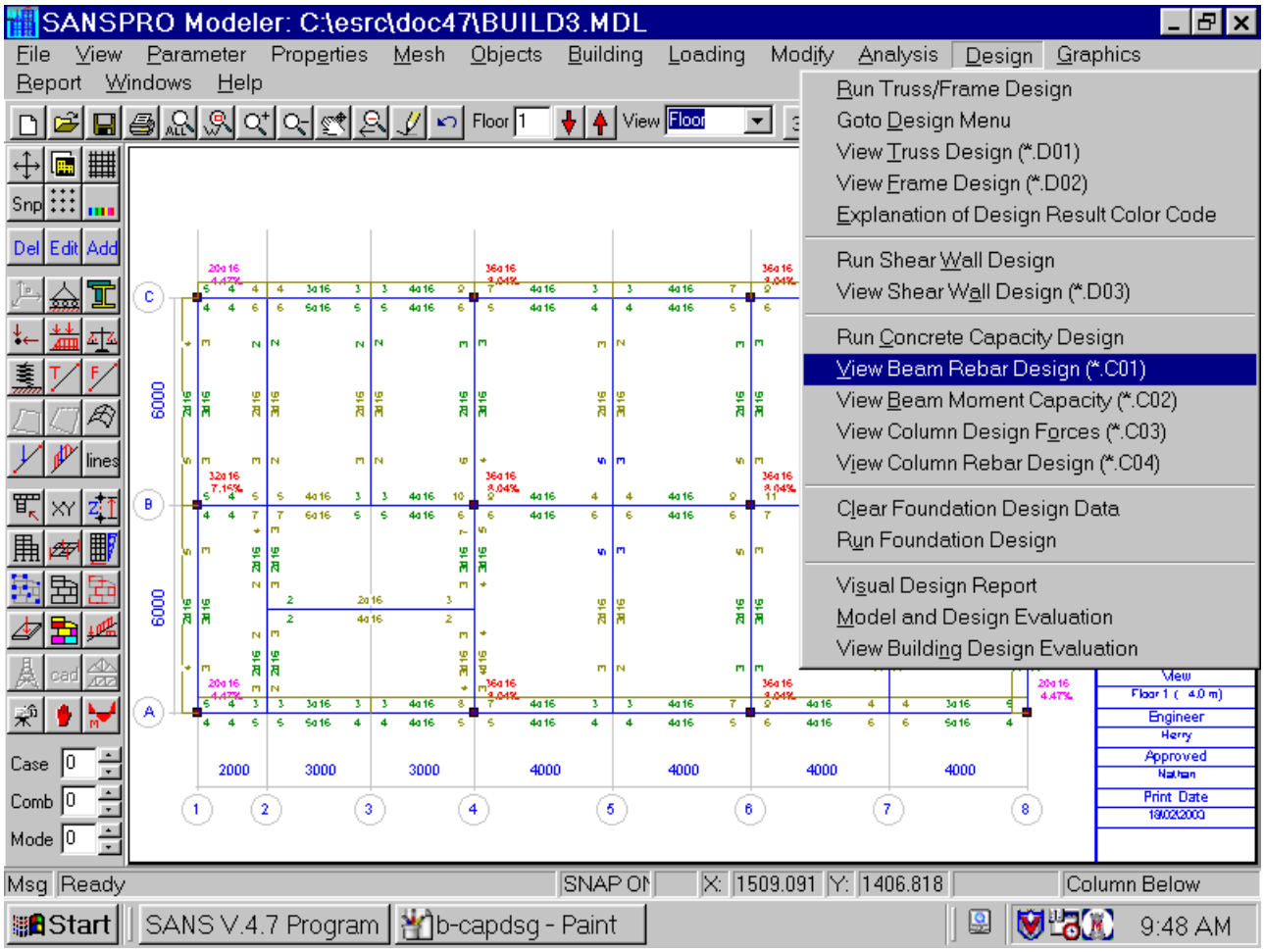

SANSPRO Modeler: C:\esrc\doc47\BUILD3.MDL
File View Parameter Properties Mesh Objects Building Loading Modify Analysis Design Graphics
Report Windows Help
SANS V.4.7 Program
Concrete Capacity Design : Column
OK
SANSPRO Version 4.7
Floor View, Above
Support
Column Element
Beam Element
Floor Slab
Column Below Floor
Main Rebar
Loading Comb = ALL
Licensee
Ir. Nathan Maduluh, M.Sc.
Bandung
Project
Noname
Filename
build3.mdl
View
Floor 3 (11.0 m)
Engineer
Herry
Approved
Nathan
Print Date
18/02/2000
Msg Ready
Column Below
Start
SANS V.4.7 Program
b-cap7 - Paint
9:47 AM

4. Visual Design Report

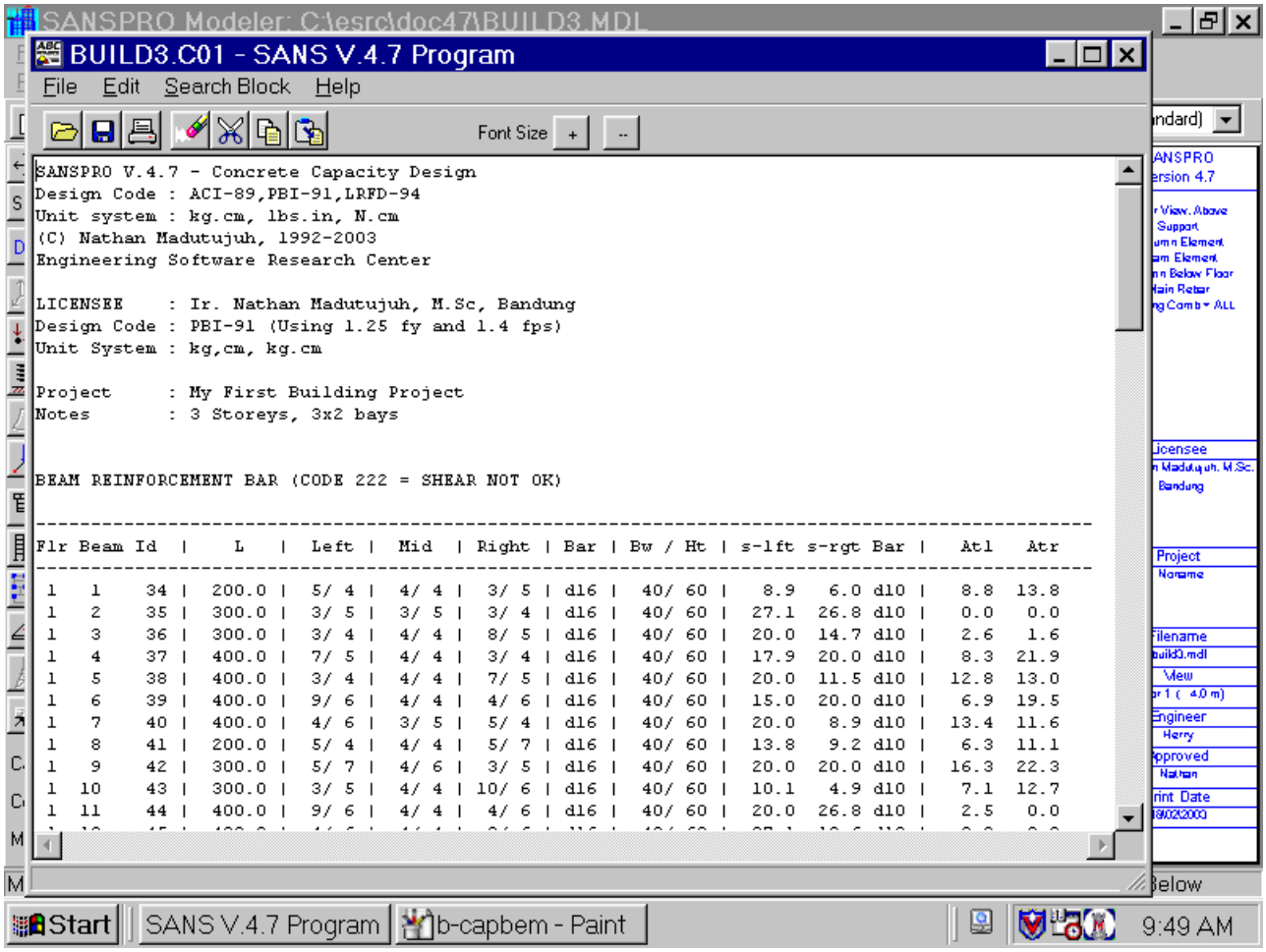
- To see Capacity Design Results, **Right-Click** and Select **View Design Results** as usual.
- Move up/down floor to see other floor
- Design Capacity only give a set of design result stored at Load Combination 1



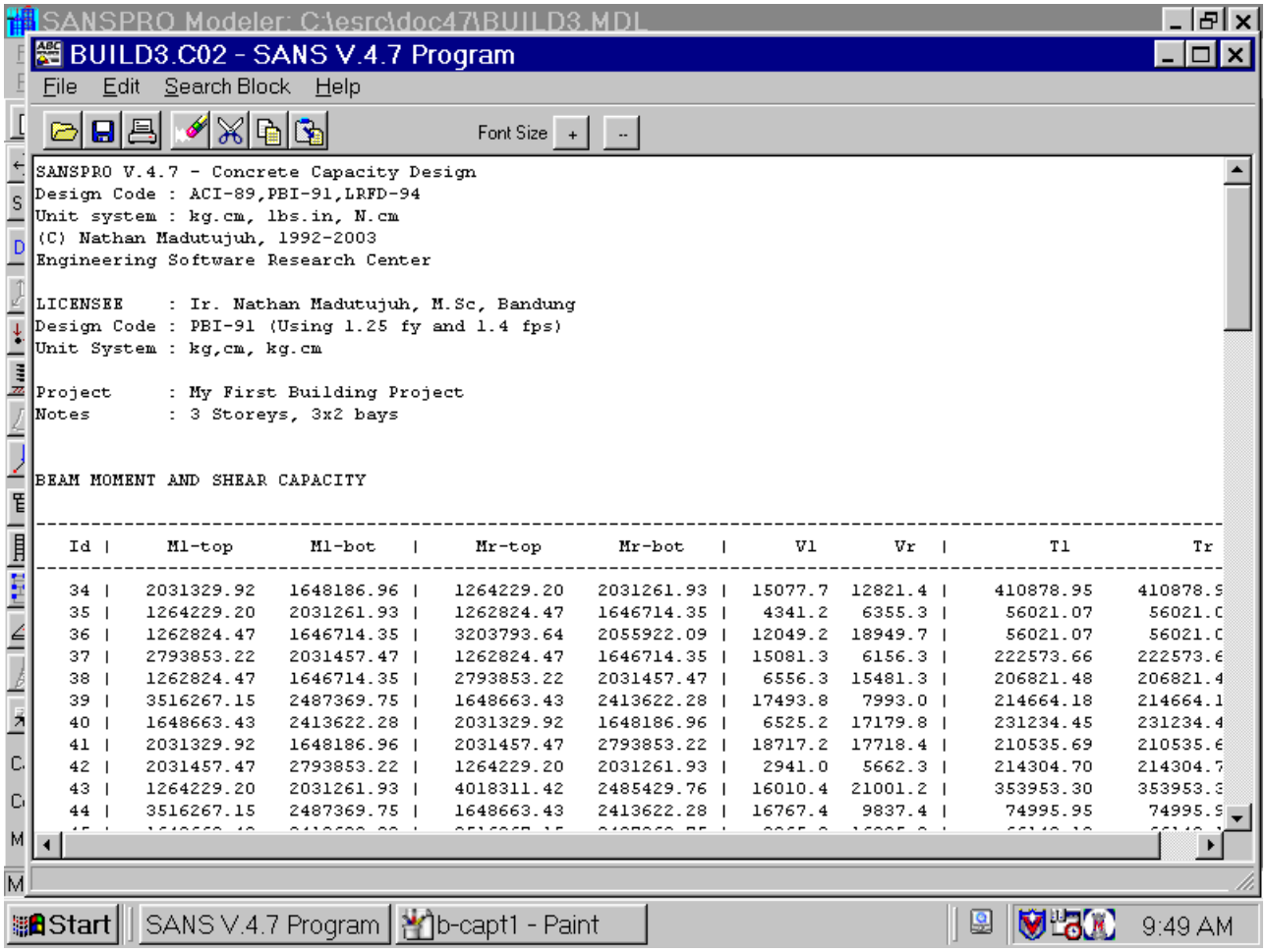
- To see Capacity Design Text Output:



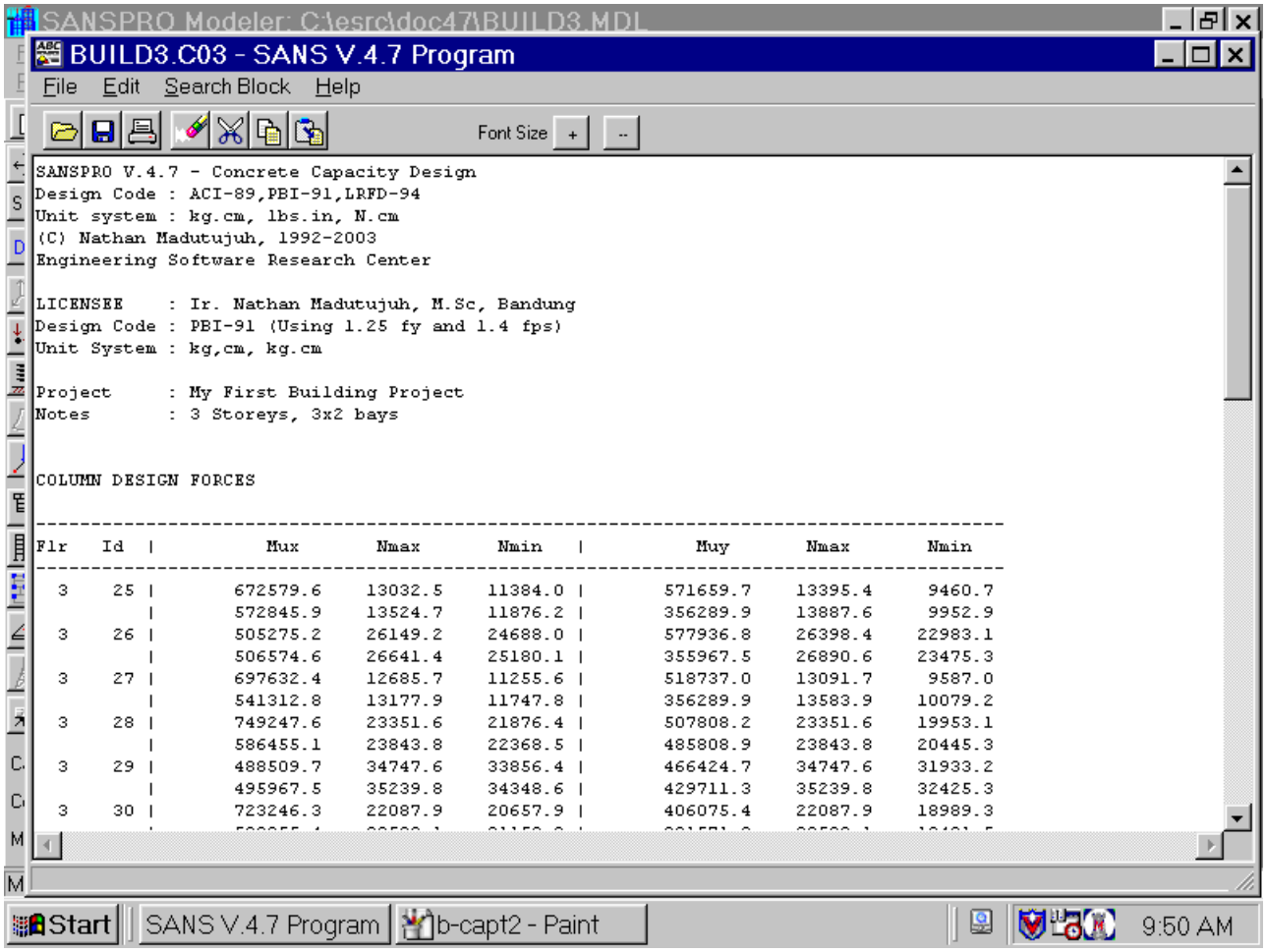
5. Capacity Design Report: Beam Rebar



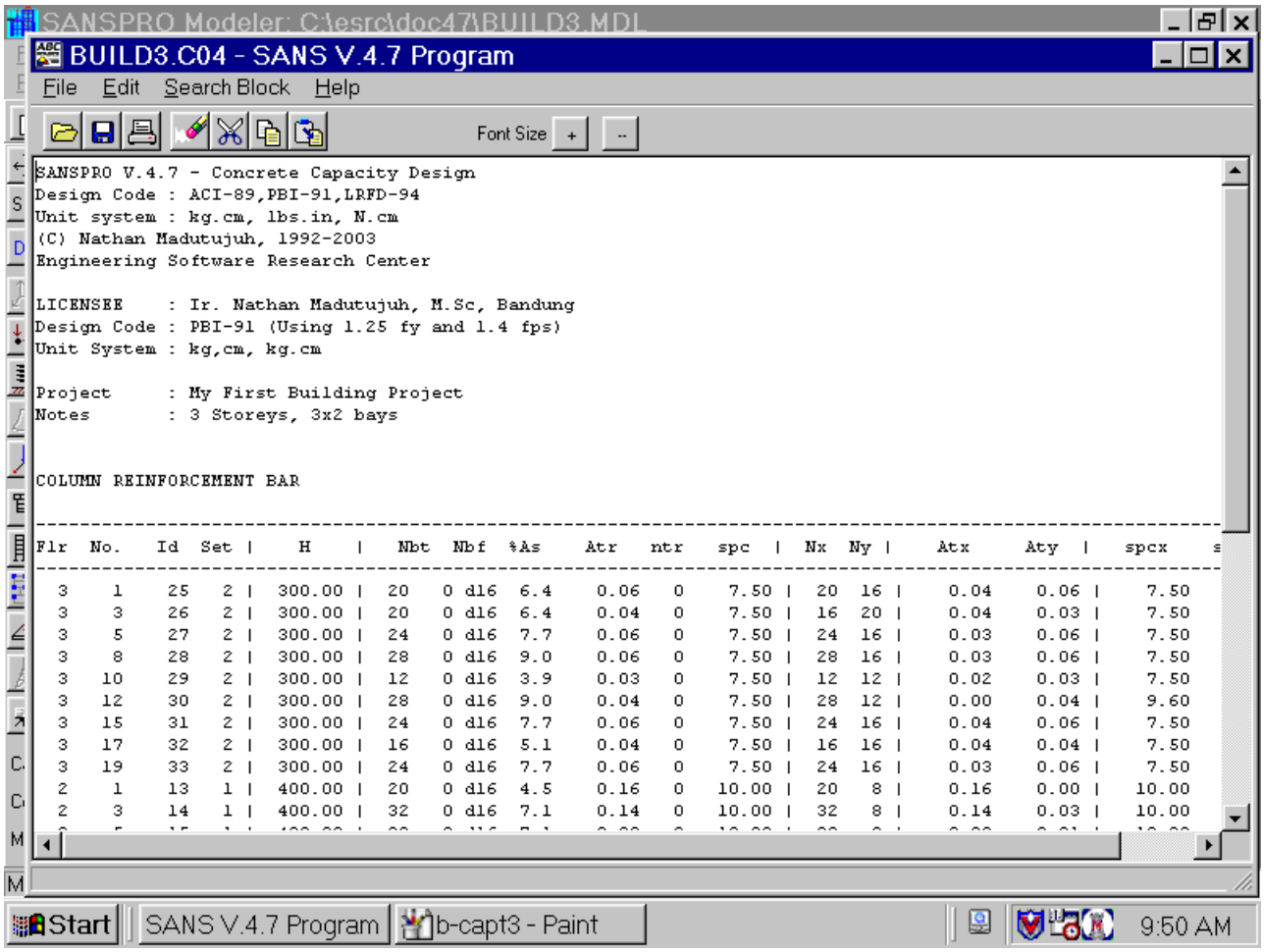
6. Capacity Design Report: Beam Moment Capacity



7. Capacity Design Report: Column Design Forces



8. Capacity Design Report: Column Rebar



Flr	No.	Id	Set	H	Nbt	Nbf	%As	Atr	ntr	spc	Nx	Ny	Atx	Aty	spcx
3	1	25	2	300.00	20	0 dl6	6.4	0.06	0	7.50	20	16	0.04	0.06	7.50
3	3	26	2	300.00	20	0 dl6	6.4	0.04	0	7.50	16	20	0.04	0.03	7.50
3	5	27	2	300.00	24	0 dl6	7.7	0.06	0	7.50	24	16	0.03	0.06	7.50
3	8	28	2	300.00	28	0 dl6	9.0	0.06	0	7.50	28	16	0.03	0.06	7.50
3	10	29	2	300.00	12	0 dl6	3.9	0.03	0	7.50	12	12	0.02	0.03	7.50
3	12	30	2	300.00	28	0 dl6	9.0	0.04	0	7.50	28	12	0.00	0.04	9.60
3	15	31	2	300.00	24	0 dl6	7.7	0.06	0	7.50	24	16	0.04	0.06	7.50
3	17	32	2	300.00	16	0 dl6	5.1	0.04	0	7.50	16	16	0.04	0.04	7.50
3	19	33	2	300.00	24	0 dl6	7.7	0.06	0	7.50	24	16	0.03	0.06	7.50
2	1	13	1	400.00	20	0 dl6	4.5	0.16	0	10.00	20	8	0.16	0.00	10.00
2	3	14	1	400.00	32	0 dl6	7.1	0.14	0	10.00	32	8	0.14	0.03	10.00