

Manual

SYS Design Kit

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SYS DESIGN KIT PROGRAM. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFICATION THE RESULTS

Contents

Introduction to SYS Design Kit	4
Truss Design Module	12
Lattice Truss Design	14
Howe Truss Design	14
Arch Truss Design	15
Frame Design Module	25
Floor Design Module	35
Reference	

Introduction

Introduction

SYS Design Kit is a design software for steel structure. The software is able to design truss structure, frame structure and floor structure. There are five modules namely, Lattice Truss Design, Howe Truss Design, Arch Truss Design, Frame Design and Floor Design. All modules are written in the same environment and interactively interface with users.

The software uses windows platform. Users can easily control the program by the command buttons as shown.

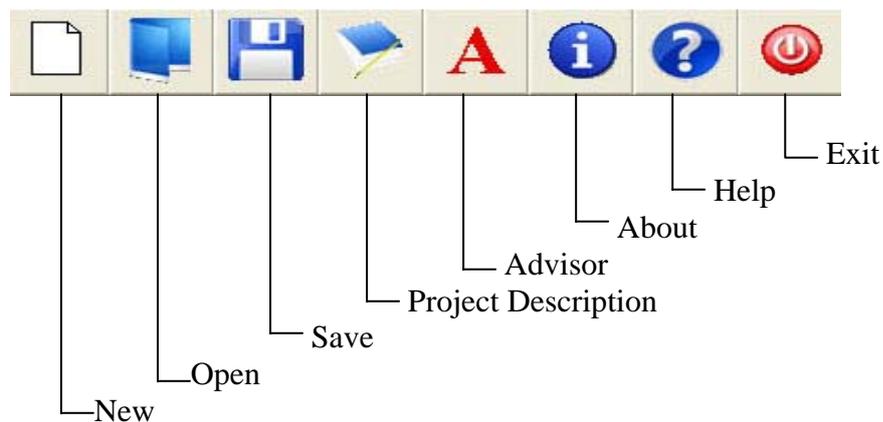


Figure 1. Control Buttons

Control Buttons

1. New File Button

To be used for creating the new file.

2. Open File Button

To be used for opening the existing file in the database.

3. Save File Button

To be used for saving the file onto hard disk or floppy disk or removable disk.

4. Project Description Button

To be used for showing user information about the project e.g. project name, job title, job no., engineer. The relevant information of Design task can be input as the reference.



Figure 2. Project Description

5. Advisor Button

To be used for helping user to create truss geometry: lattice truss, howe truss and howe truss.

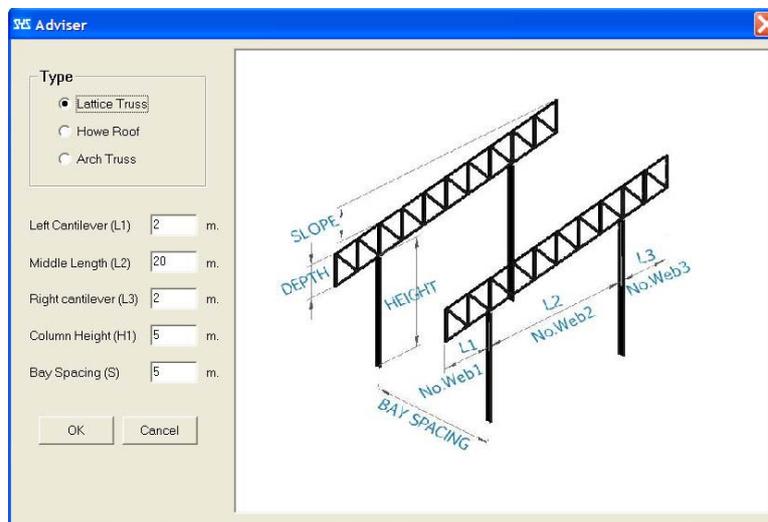


Figure 3. Advisor for Lattice Truss

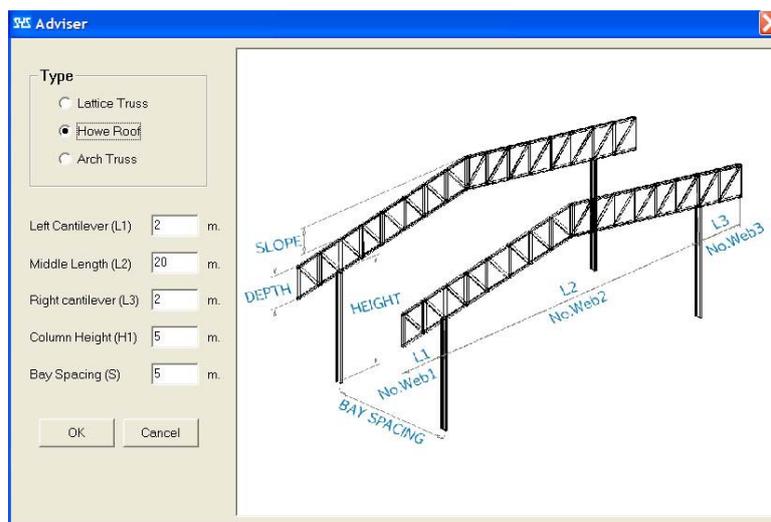


Figure 4. Advisor for Howe Truss

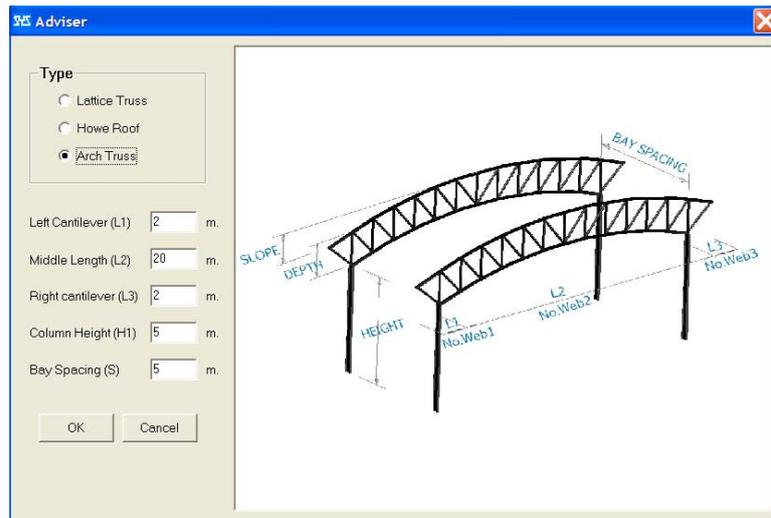


Figure 5. Adviser for Arch Truss

6. About Button

Containing program information such as, a version of the program is now being used.

7. Help Button

To be used for going to user's manual.

8. Exit Button

To be used for exiting the program.

System Requirement

CD-ROM drive (for installation)

Microsoft Windows 98 (original and Second Edition), Windows Millennium Edition (ME), Windows NT 4.0 (with Service Pack 5 for Y2K compliancy or Service Pack 6a), Windows 2000 (Service Packs 1 or 2), or Windows XP

Pentium, Pentium Pro, Pentium II, Pentium III, Pentium IV, Xeon or AMD Athlon, Athlon XP based personal computer

64 MB RAM minimum, 128 MB RAM recommended

Disk space varies depending on size of partition and installation

8-bit graphics adapter and display

Design Module 1: Steel Truss Design

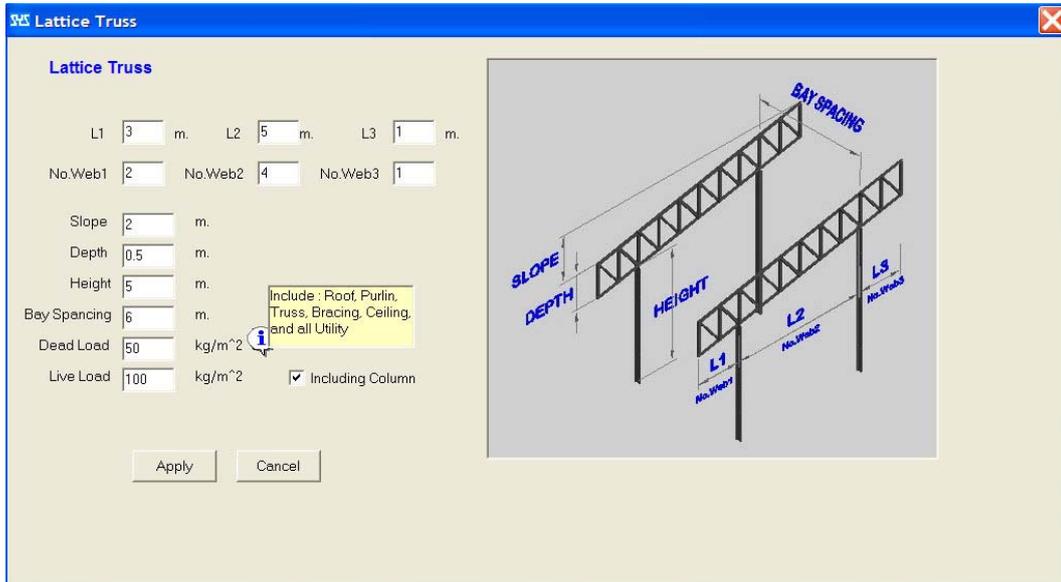


Figure 6. Lattice Truss Design Module

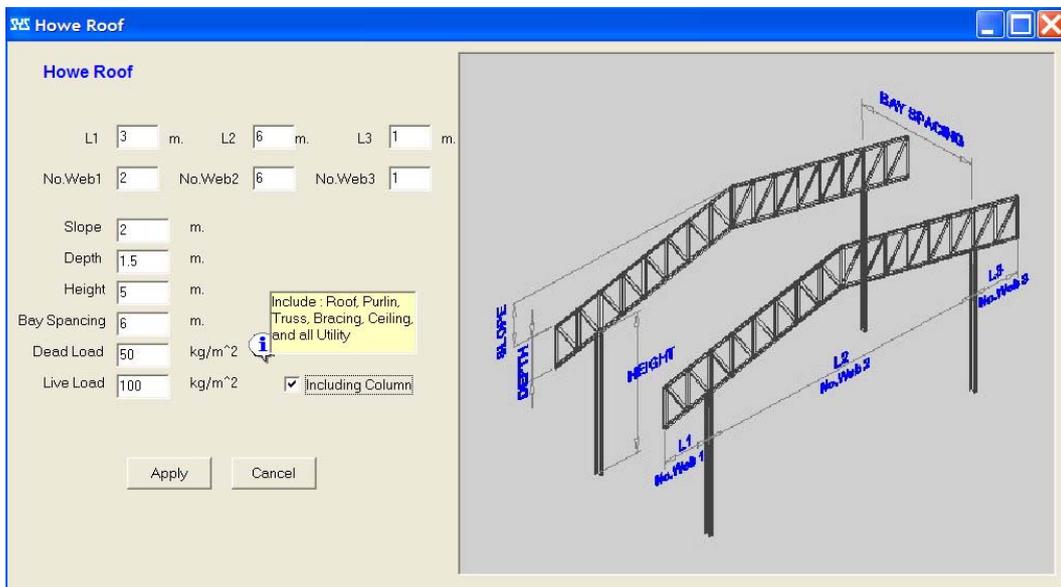


Figure 7. Howe Truss Design Module

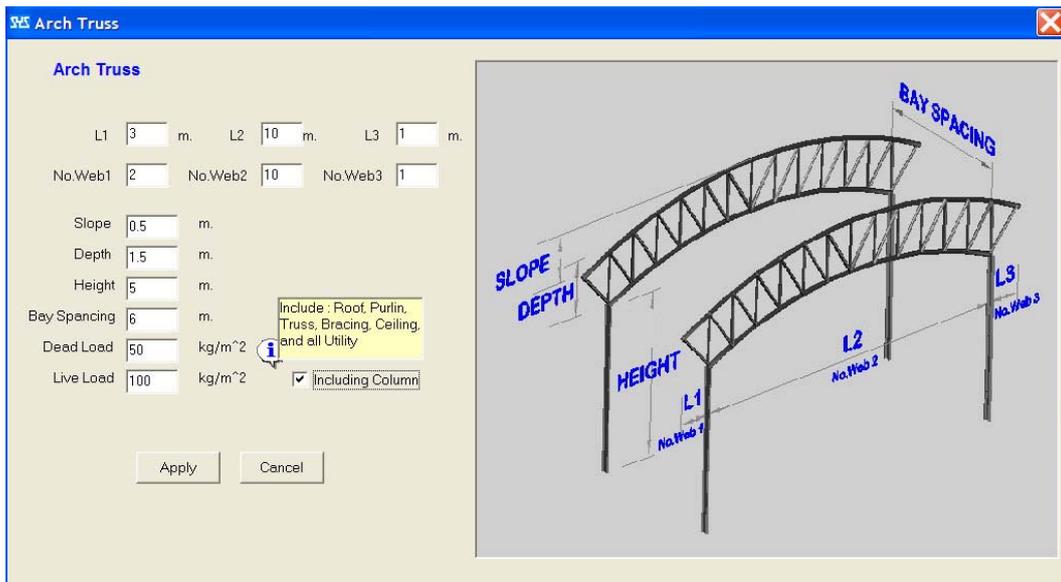


Figure 8. Arch Truss Design Module

Steel truss design module provides the design of lattice truss, howe truss and arch truss geometry as shown in Figure 6-8. All joints in model of truss are pinned joints. The dead load and live load are applied to truss member by point load at joint. All information required for truss design such as truss and column geometry, member section, dead load, live load and so on, shall be prepared and filled in the input box by users. The option on truss geometry advisor is provided for user as a first start of design, then user can modify the data for his/her designed variables.

Design Module 2: Steel Frame Design

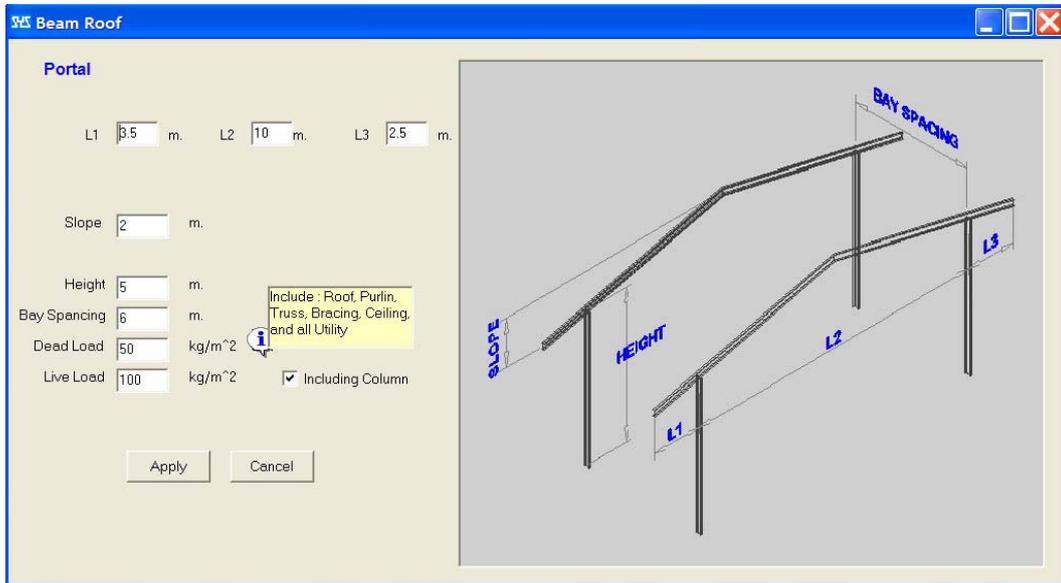


Figure 9. Steel Frame Design Module

Steel frame design module provides the design of roof frame structure as shown in Figure 9. All joints in model of roof frame are fixed joints. The dead load and live load are applied to frame member by uniform load along the member. All information required for frame design such as frame geometry, member section, dead load, live load and so on, shall be prepared and filled in the input box by users.

Design Module 3: Steel Floor Design

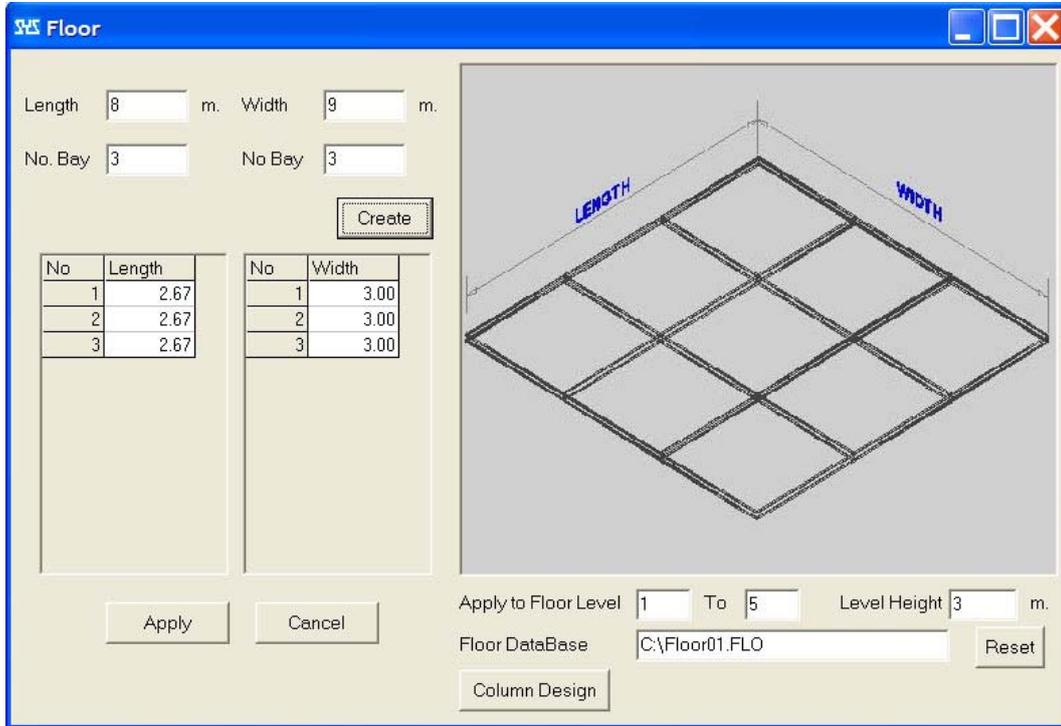


Figure 10. Steel Floor Design Module

Steel floor design module provides the design of floor structure as shown in Figure 10. All joints in model of floor frame are fixed joints. The dead load and live load are applied to floor member by uniform load along the member. All information required for frame design such as floor geometry, support location, member section, type of slab design, dead load, live load and so on, shall be prepared and filled in the input box by users.

Design Modules

Steel Truss Design

Steel truss design module provides the design of lattice truss, howe truss and arch truss geometry. All joints in model of truss are pinned joints. The dead load and live load are applied to truss member by point load at joint. All members are carrying compression and tension force from applied load. All information required for truss design such as truss and column geometry, member section, dead load, live load and so on, shall be prepared and filled in the input box by users.

Input Mode

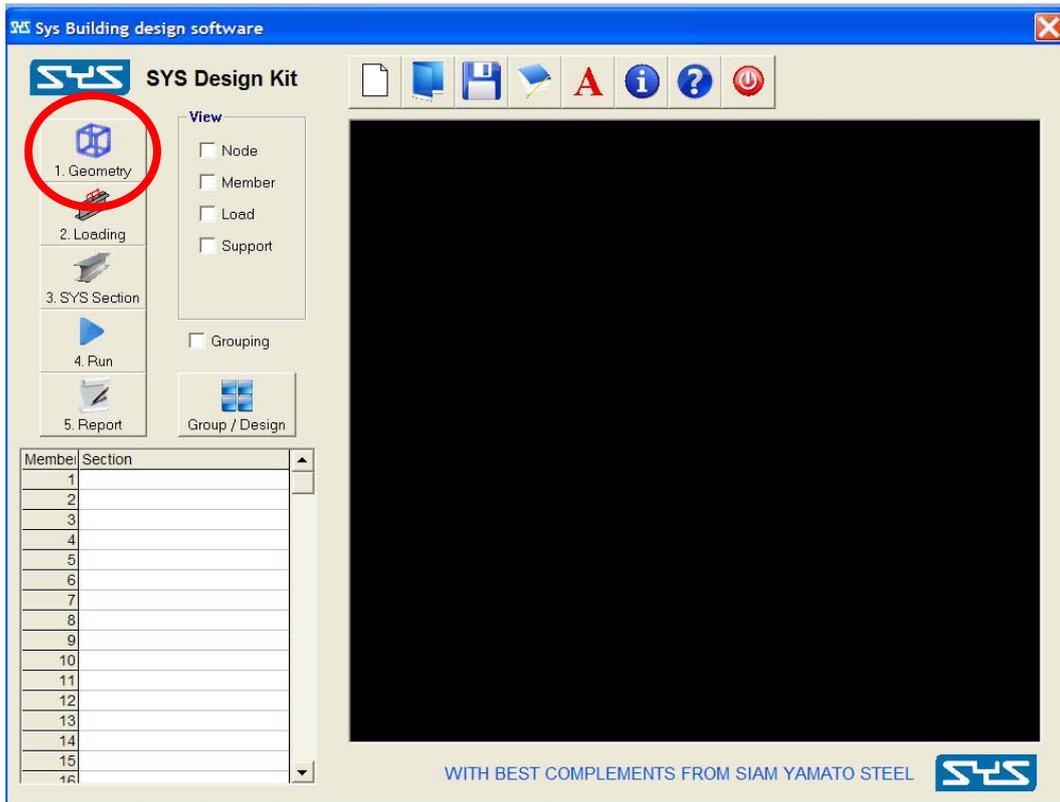


- 1) Steel truss design can be selected by clicking “Using Advisor” or “Go to Program” and then clicking “OK”

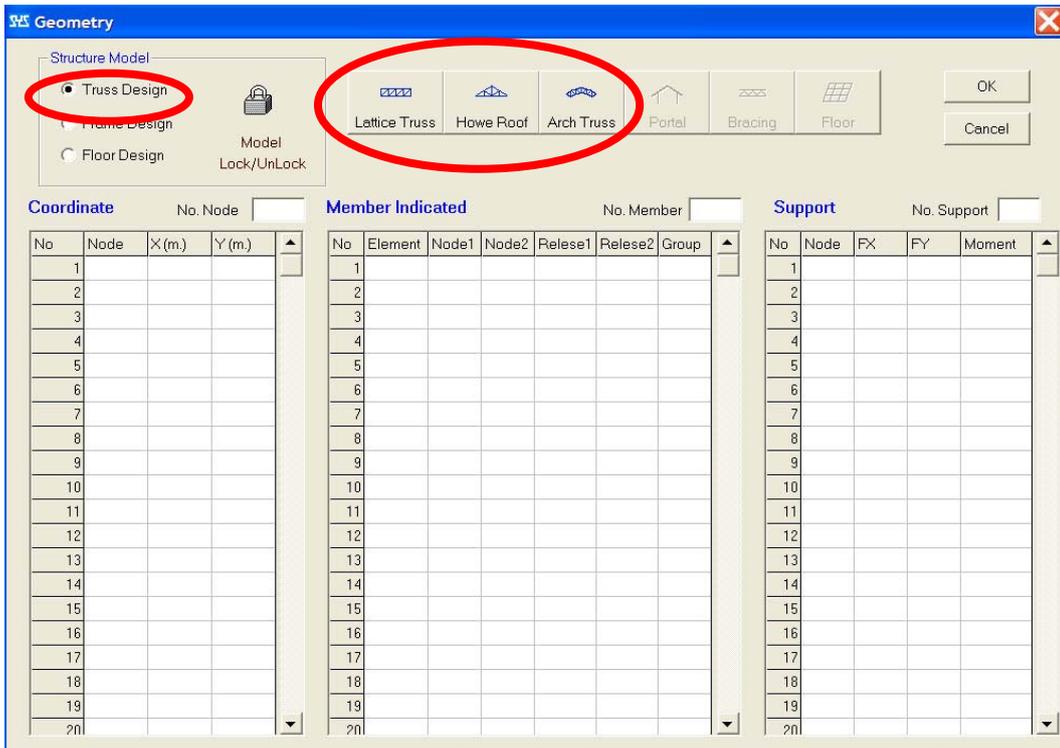
Using Advisor Button : Program will help user to crate geometry of truss, see explanation in Advisor Input Mode
 Go to ProgramButton : Go to normal main menu input program

- 2) Clicking “1.Geometry” button for input geometry of truss. In main menu page, there are “View” type for user to select display type of information in truss as described below

Node : Display node number of truss
 Member : Display member number of truss
 Load : Display load location, direction and magnitude
 Support : Display support location

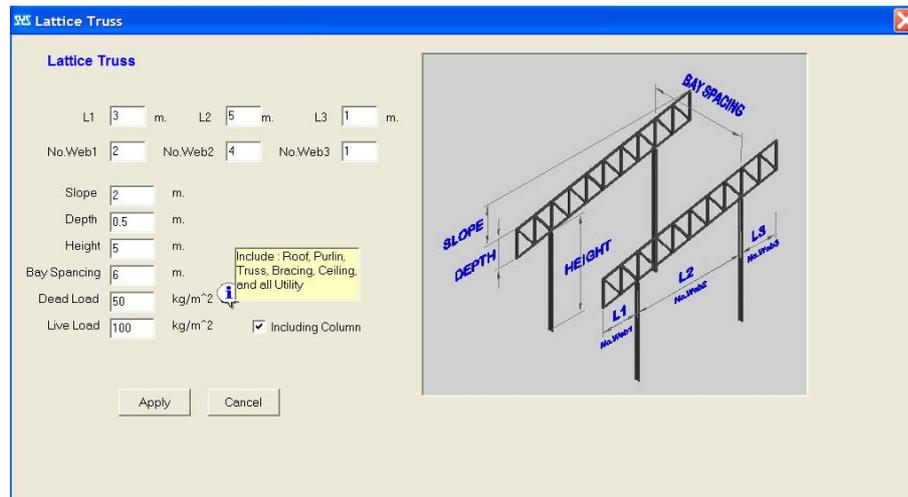


3) Clicking “Truss Design” and then select type of truss: “Lattice Truss”, “Howe Truss” and “Arch Truss”.



4) For Lattice Truss Design, variables are required to be filled as follows

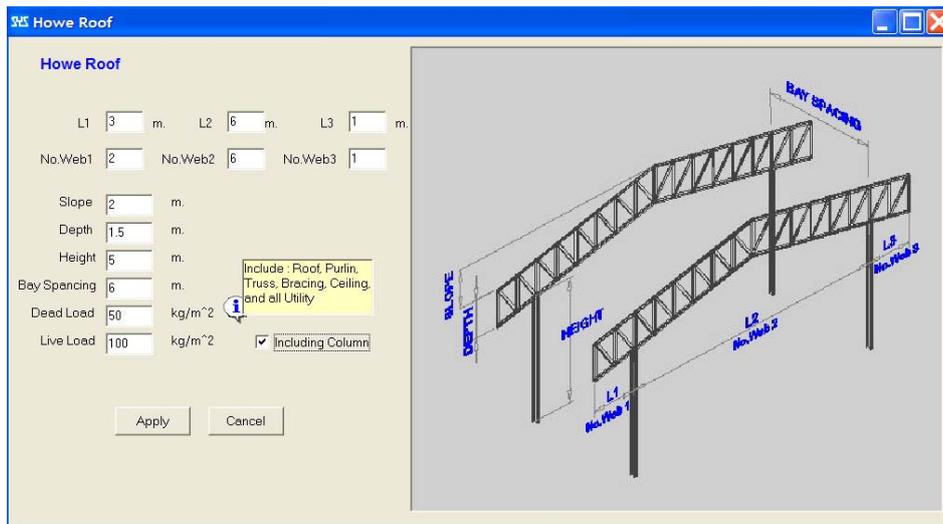
- L1 : Left cantiliver span length in meter
- No. of Web1 : Number of web in left cantiliver span
- L2 : Middle span length in meter
- No. of Web2 : Number of web in middle span
- L3 : Right cantiliver length in meter
- No. of Web3 : Number of web in right cantiliver span
- Slope : Different elevation (slope) between left and right truss in meter
- Depth : Depth of truss in meter
- Height (H1) : Height of left column in meter
- Bay Spacing (S) : Spacing between bay of column in meter
- Dead load : Dead load in kilogram per square meter including load of roof, purlin, truss, bracing, ceiling and all utility
- Live load : Live load in kilogram per square meter
- Including column : Including or not including column in design



5) For Howe Truss Design, variables are required to be filled as follows

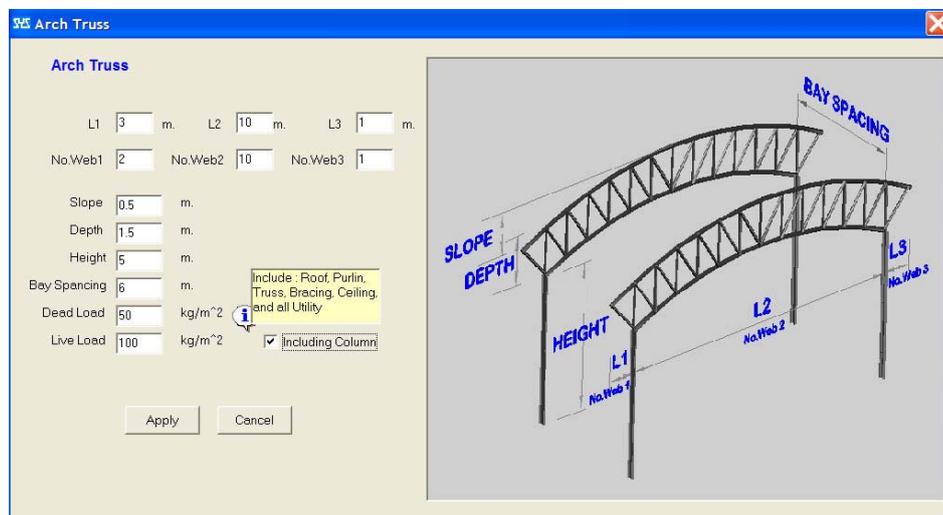
- L1 : Left cantiliver span length in meter
- No. of Web1 : Number of web in left cantiliver span
- L2 : Middle span length in meter
- No. of Web2 : Number of web in middle span
- L3 : Right cantiliver length in meter
- No. of Web3 : Number of web in right cantiliver span
- Slope : Different elevation (slope) between left and right column in meter
- Depth : Depth of truss in meter
- Height (H1) : Height of left column in meter
- Bay Spacing (S) : Spacing between bay of column in meter
- Dead load : Dead load in kilogram per square meter including load of roof, purlin, truss, bracing, ceiling and all utility
- Live load : Live load in kilogram per square meter

Including column : Including or not including column in design

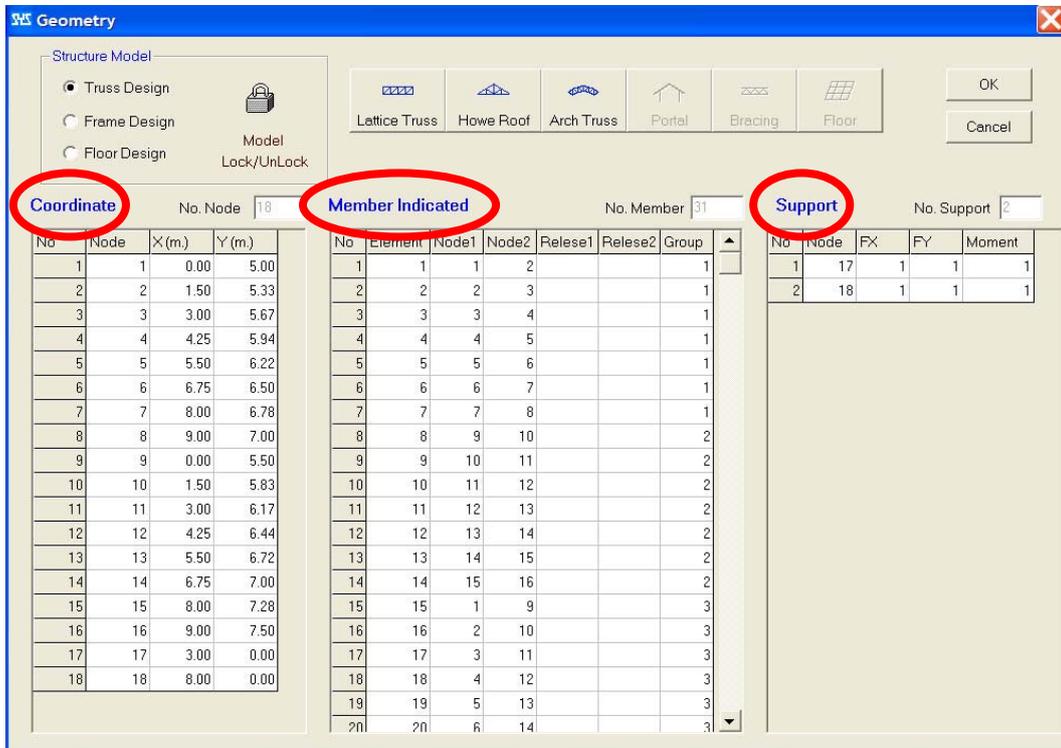


6) For Arch Truss Design, variables are required to be filled as follows

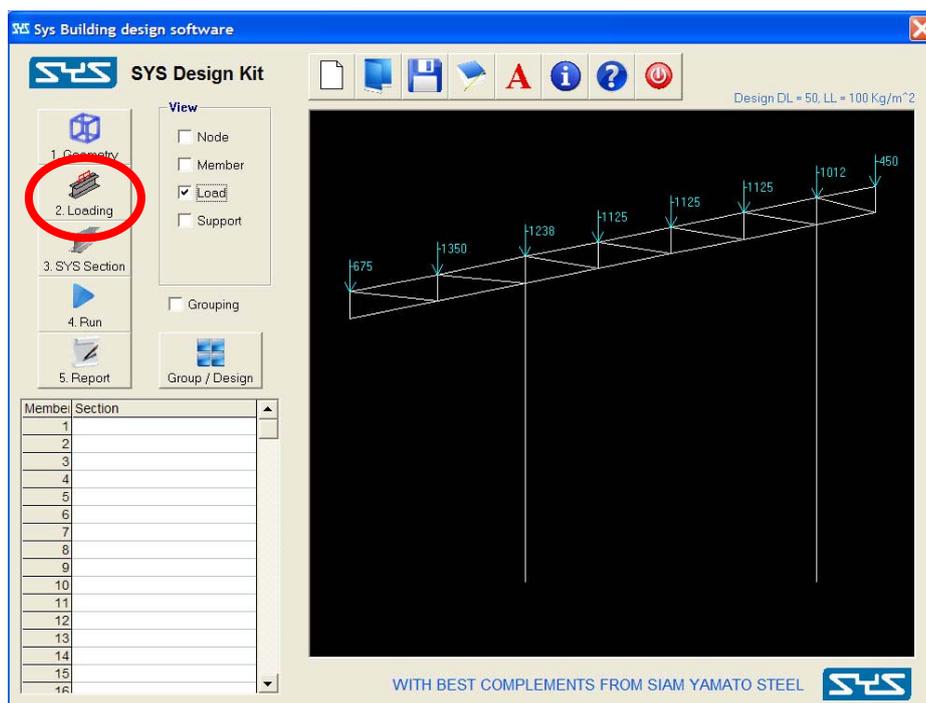
- L1 : Left cantiliver span length in meter
- No. of Web1 : Number of web in left cantiliver span
- L2 : Middle span length in meter
- No. of Web2 : Number of web in middle span
- L3 : Right cantiliver length in meter
- No. of Web3 : Number of web in right cantiliver span
- Slope : Different elevation (slope) between left and right column in meter
- Depth : Depth of truss in meter
- Height (H1) : Height of left column in meter
- Bay Spacing (S) : Spacing between bay of column in meter
- Dead load : Dead load in kilogram per square meter including load of roof, purlin, truss, bracing, ceiling and all utility
- Live load : Live load in kilogram per square meter
- Including column : Including or not including column in design

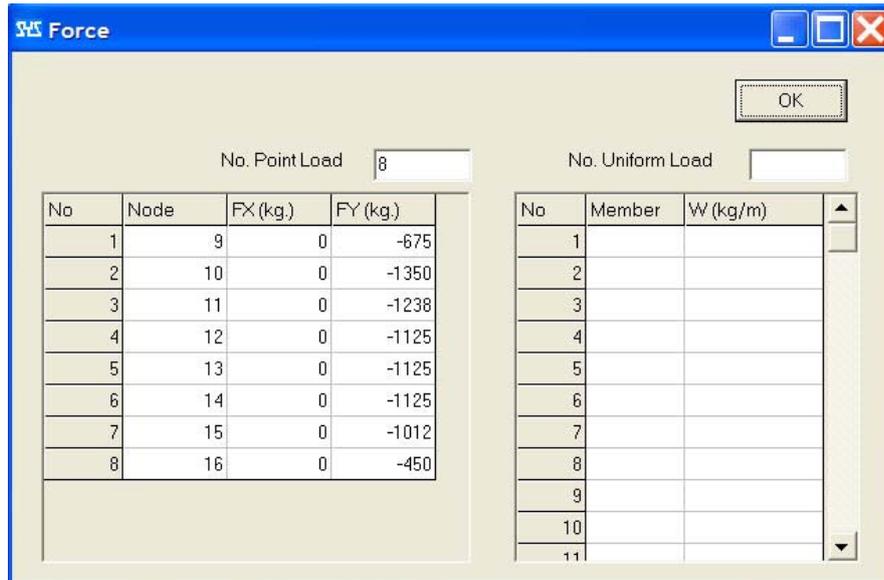


- 7) Then “Coordinate”, “Member indicated” and “Support” will be created in window and then clicking “OK” to go back to main menu. The user can edit any data of “Coordinate”, “Member indicated” and “Support” by double click and edit data in the window.



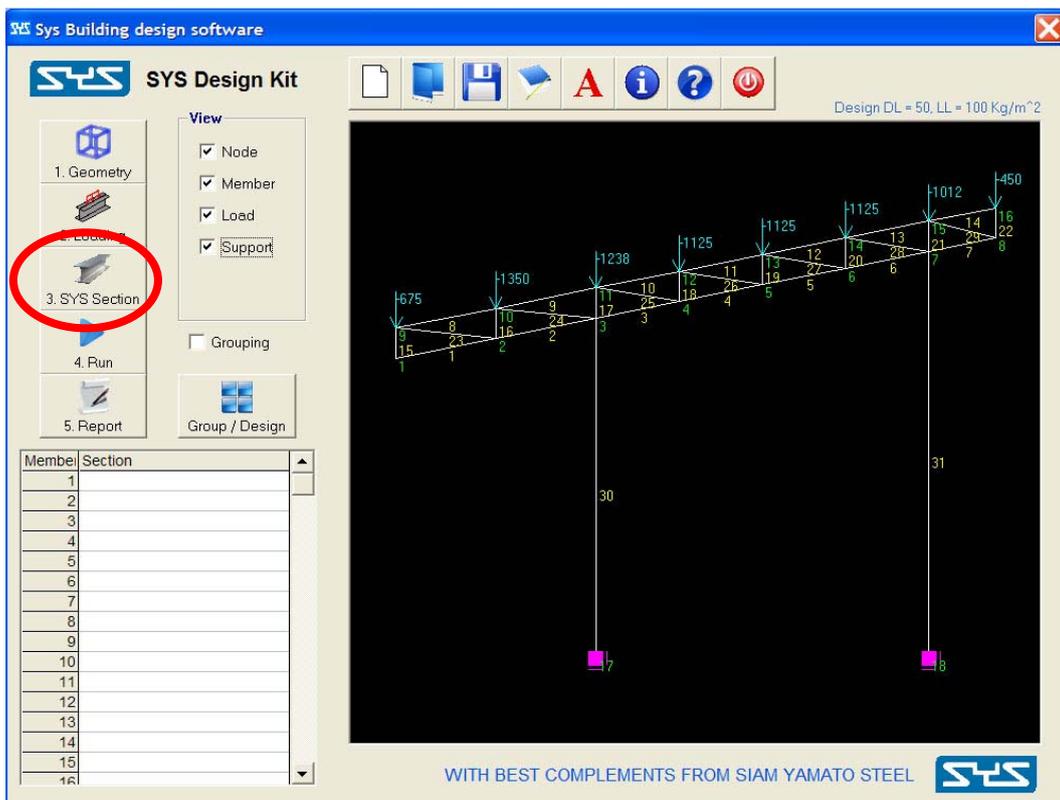
- 8) Clicking “2.Loading” button in main menu to go to edit load data and clicking “OK” to go back to main menu. Program will automatically generate point load from dead load and point load data in step 4 or 5 or 6. For other applied force such as wind load, seismic load, user can edit load data in window by clicking and editing load at the joint number.

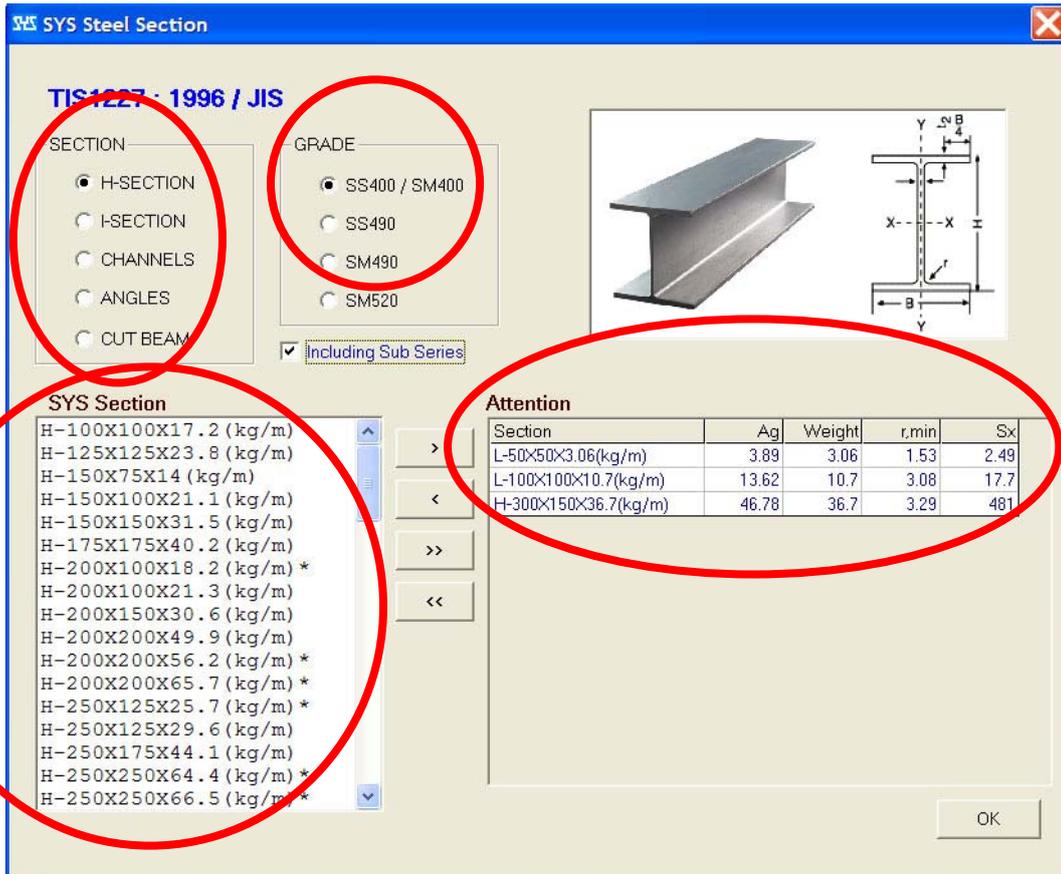




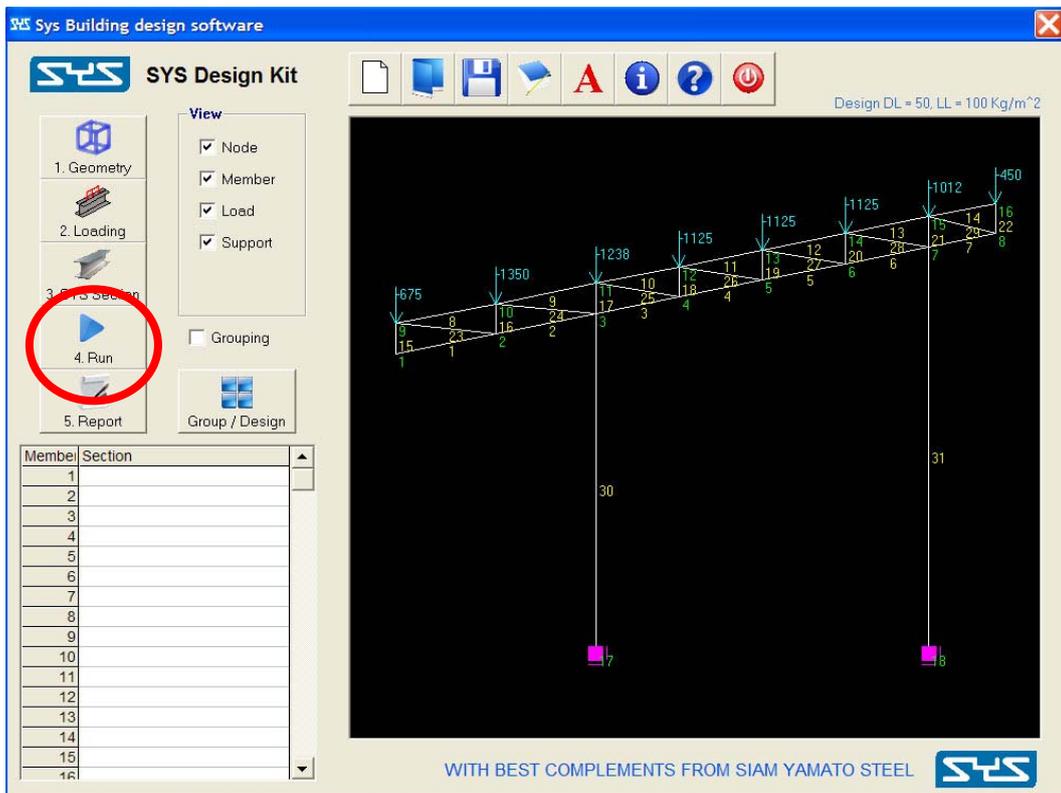
- 9) Select “SYS Section” button to go to input steel section. User can select “H-Section”, “I-Section”, “Channels”, “Angles” and “Cut beam” from left column to “Attention Table” in the right column for program to use this section to design. More steel section from SYS are provided in program by clicking “Including Sub Series”. The sub series sections are not normal sections in the market. Please ask SYS for more information.

Select “Steel Grade” of member by clicking buttons in front of type of steel grade.





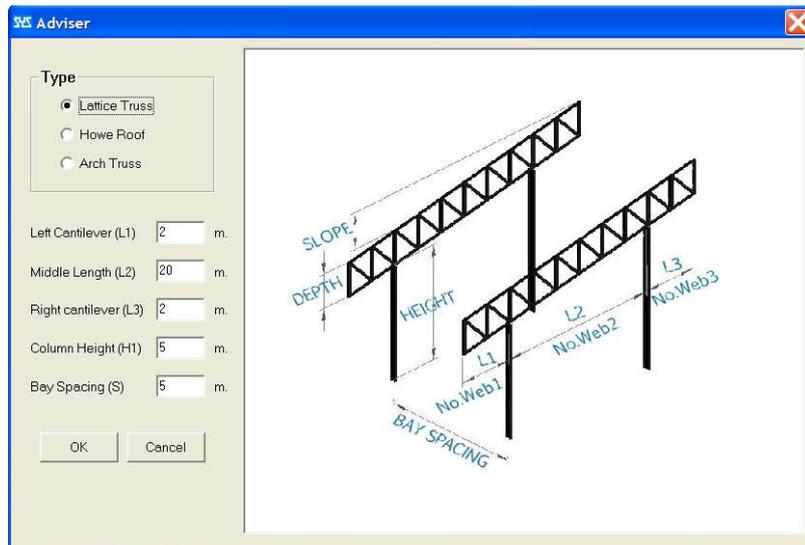
10) Run the program by clicking “4.Run” button.



Advisor Input Mode

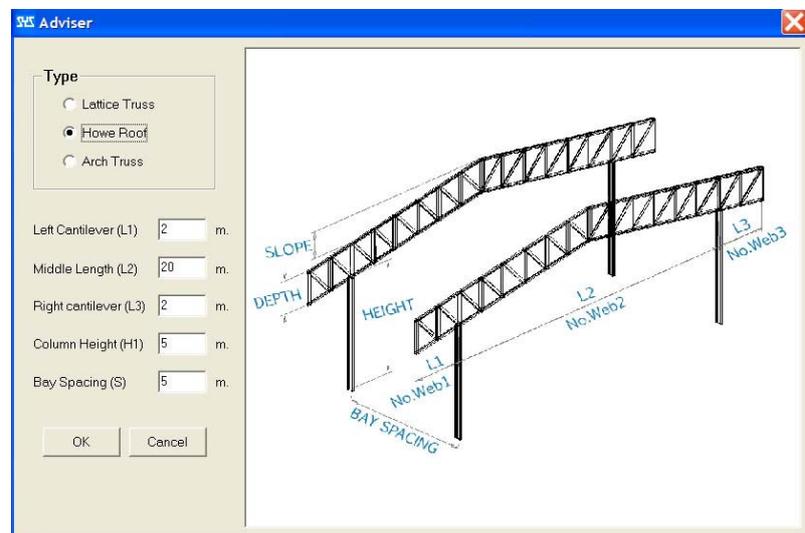
A) For lattice truss, user can use advisor input mode for helping to create truss geometry.

L1 : Left cantiliver span length in meter
 L2 : Middle span length in meter
 L3 : Right cantiliver length in meter
 Height (H1) : Height of left column in meter
 Bay Spacing (S) : Spacing between bay of column in meter



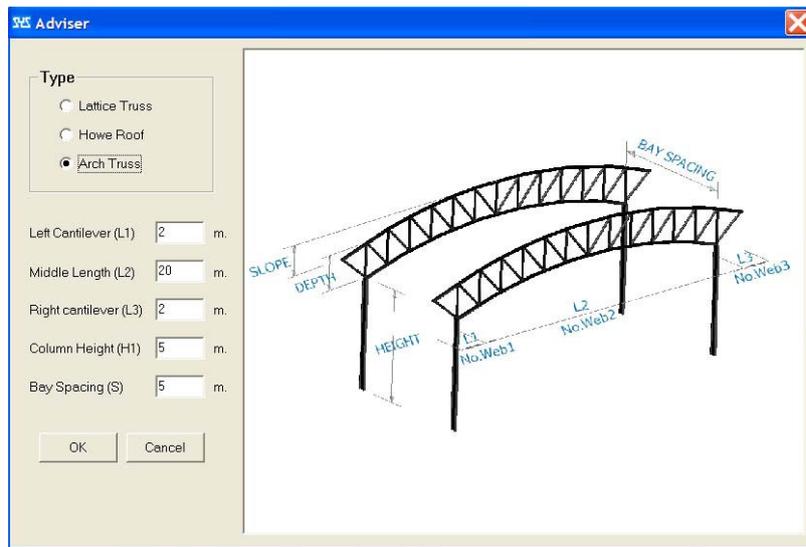
B) For howe truss, user can use advisor input mode for helping to create truss geometry.

L1 : Left cantiliver span length in meter
 L2 : Middle span length in meter
 L3 : Right cantiliver length in meter
 Height (H1) : Height of left column in meter
 Bay Spacing (S) : Spacing between bay of column in meter



C) For arch truss, user can use advisor input mode for helping to create truss geometry.

- L1 : Left cantiliver span length in meter
- L2 : Middle span length in meter
- L3 : Right cantiliver length in meter
- Height (H1) : Height of left column in meter
- Bay Spacing (S) : Spacing between bay of column in meter



D) After input advior mode, the user can edit data and input more data by procedure step 2-10.

FINITE ELEMENT ANALYSIS

Member	Node	Axial	Shear-Y	Shear-Z	Torsion	Moment-Y	Moment-Z
1	NODE(1)	7.0548E+00	-2.1228E+00	0.0000E+00	0.0000E+00	0.0000E+00	-1.6021E+00
	MIDSPAN					0.0000E+00	-2.8069E-02
2	NODE(2)	-7.0548E+00	2.1228E+00	0.0000E+00	0.0000E+00	0.0000E+00	-1.6582E+00
	MIDSPAN	2.1020E+03	-8.3925E+00	0.0000E+00	0.0000E+00	0.0000E+00	-4.8445E+00
3	NODE(3)	-2.1020E+03	8.3925E+00	0.0000E+00	0.0000E+00	0.0000E+00	-8.0635E+00
	MIDSPAN	8.2366E+03	1.2963E+01	0.0000E+00	0.0000E+00	0.0000E+00	-1.6095E+00
4	NODE(4)	-8.2366E+03	-1.2963E+01	0.0000E+00	0.0000E+00	0.0000E+00	5.3194E+00
	MIDSPAN	2.1191E+03	2.9268E+00	0.0000E+00	0.0000E+00	0.0000E+00	1.6224E+00
5	NODE(5)	-2.1191E+03	-2.9268E+00	0.0000E+00	0.0000E+00	0.0000E+00	2.1267E+00
	MIDSPAN	-1.1521E+03	1.0212E-02	0.0000E+00	0.0000E+00	0.0000E+00	-7.4634E-01
6	NODE(6)	1.1521E+03	-1.0212E-02	0.0000E+00	0.0000E+00	0.0000E+00	7.5288E-01
	MIDSPAN	-1.5432E+03	-4.2544E+00	0.0000E+00	0.0000E+00	0.0000E+00	-3.0293E+00
7	NODE(7)	1.5432E+03	4.2544E+00	0.0000E+00	0.0000E+00	0.0000E+00	3.0507E-01
	MIDSPAN	9.1310E+02	3.6941E+00	0.0000E+00	0.0000E+00	0.0000E+00	-2.4198E+00
8	NODE(8)	-9.1310E+02	-3.6941E+00	0.0000E+00	0.0000E+00	0.0000E+00	2.4854E+00
	MIDSPAN	-2.0660E+03	-9.4928E-01	0.0000E+00	0.0000E+00	0.0000E+00	-5.9419E-01
9	NODE(9)	-2.0660E+03	-9.4928E-01	0.0000E+00	0.0000E+00	0.0000E+00	1.2970E+00
	MIDSPAN					0.0000E+00	-8.2500E-01

ANALYSIS STATUS: Finish

Design Back

12) The design member sections are shown member-section table.

Sys Building design software

SYS Design Kit

Design DL = 50. LL = 100 Kg/m²

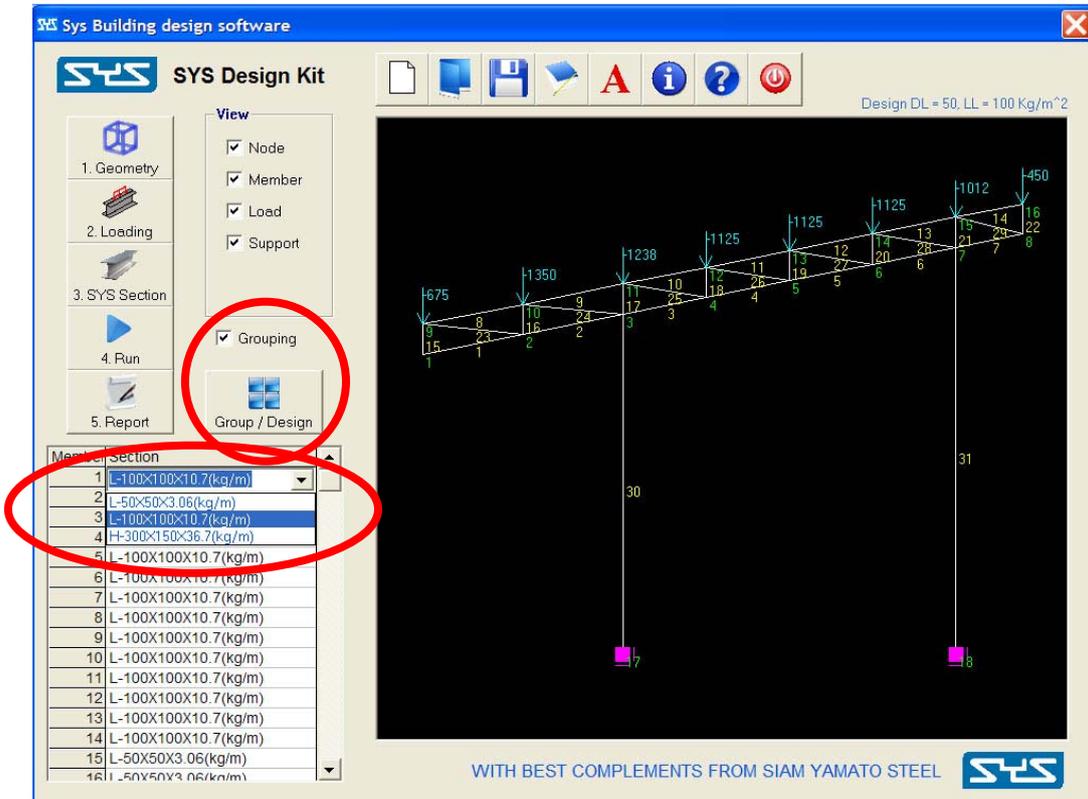
View: Node, Member, Load, Support, Grouping, Group / Design

1. Geometry, 2. Loading, 3. SYS Section, 4. Run, 5. Report

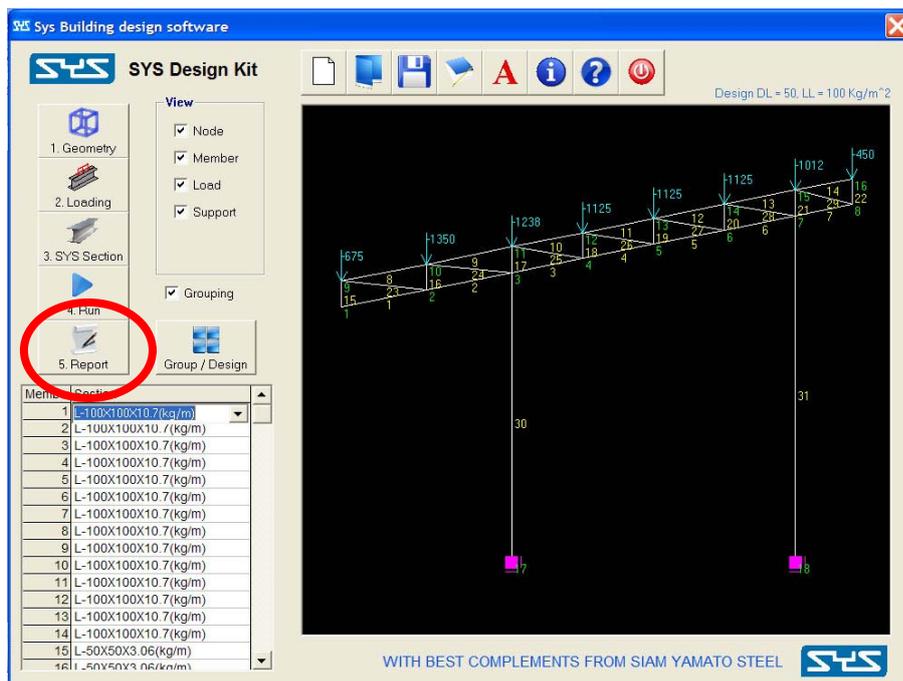
Member	Section
1	L-50X50X3.06(kg/m)
2	L-50X50X3.06(kg/m)
3	L-100X100X10.7(kg/m)
4	L-50X50X3.06(kg/m)
5	L-50X50X3.06(kg/m)
6	L-50X50X3.06(kg/m)
7	L-50X50X3.06(kg/m)
8	L-50X50X3.06(kg/m)
9	L-100X100X10.7(kg/m)
10	L-50X50X3.06(kg/m)
11	L-50X50X3.06(kg/m)
12	L-50X50X3.06(kg/m)
13	L-50X50X3.06(kg/m)
14	L-50X50X3.06(kg/m)
15	L-50X50X3.06(kg/m)
16	L-50X50X3.06(kg/m)

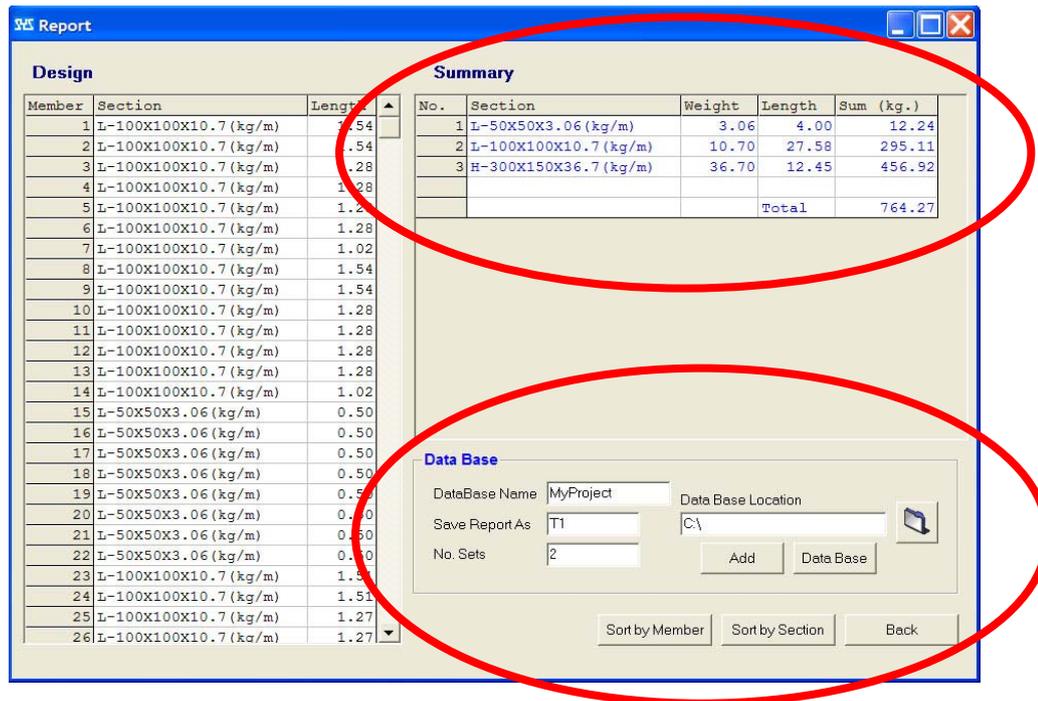
WITH BEST COMPLEMENTS FROM SIAM YAMATO STEEL

- 13) In practice, the top chord member, bottom chord member and diagonal member along truss length should be the same section for each member type. Group for each type of member can selecting by checking in front of “Grouping” and clicking “Group/Design” button. User can edit member section for each member by double click at member section and change member section in the list.



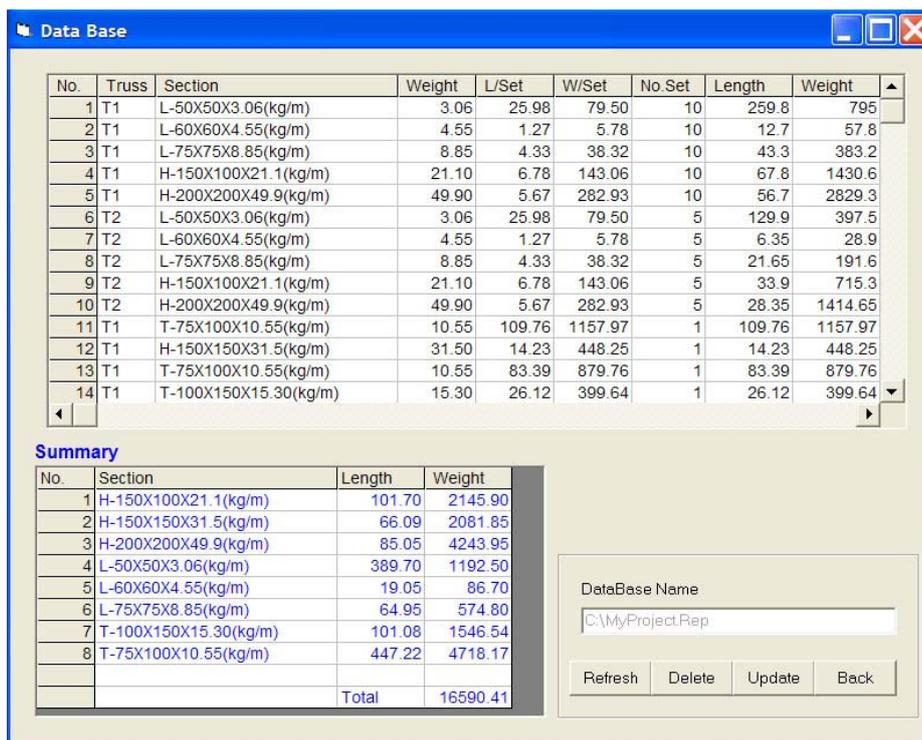
- 14) Clicking “5.Report” to see member design section, summary of steel section, summary of steel weight and save data to database.





- “Sort by Member” : Rearrange the design member table in left table by member number
- “Sort by Section” : Rearrange the design member table in left table by section of member
- “Data Base Location” : Select the location of data base to be saved
- “Data Base Name” : Name of data base to be saved
- “Save Report As” : Name of truss output to be saved
- “No. Sets” : Number set of truss in project

15) After clicking “Data base” button, the summary of all truss section in database will show.



Steel Frame Design

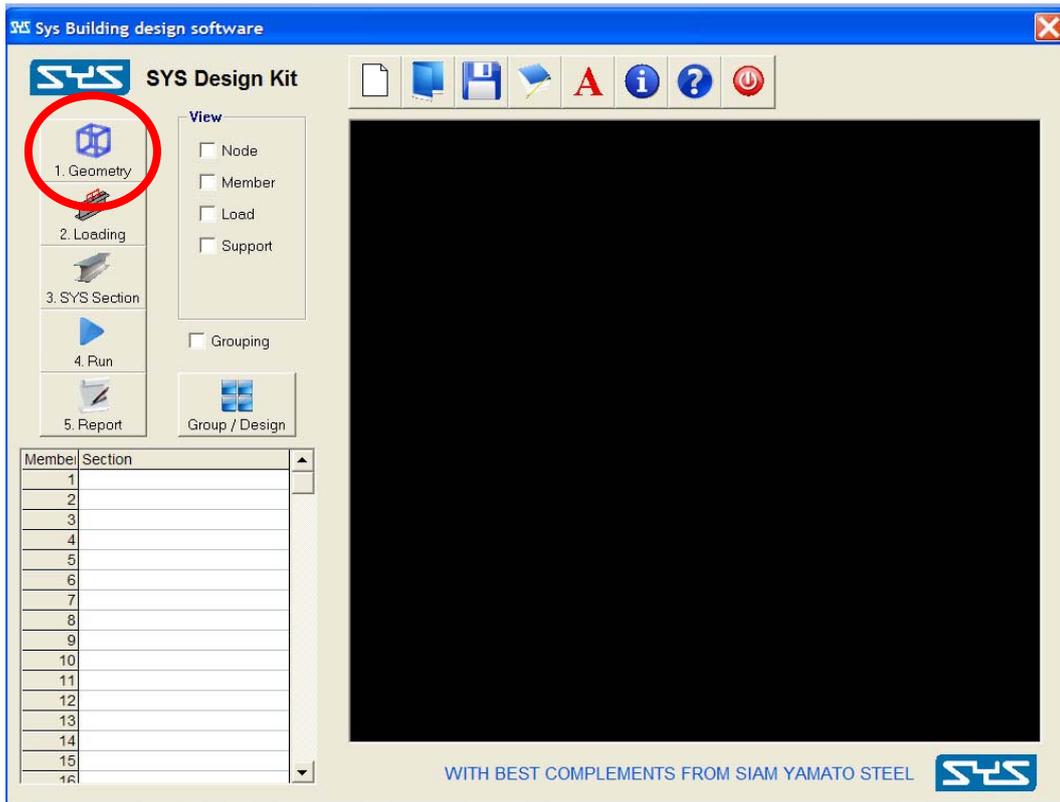
Steel frame design module provides the design of roof frame structure. All joints in model of frame are fixed joints. The dead load and live load are applied to frame member by uniform load along member. All members are carrying axial force, shear force and moment force from applied load. All information required for design such as roof frame and column geometry, member section, dead load, live load and so on, shall be prepared and filled in the input box by users.

Input Mode

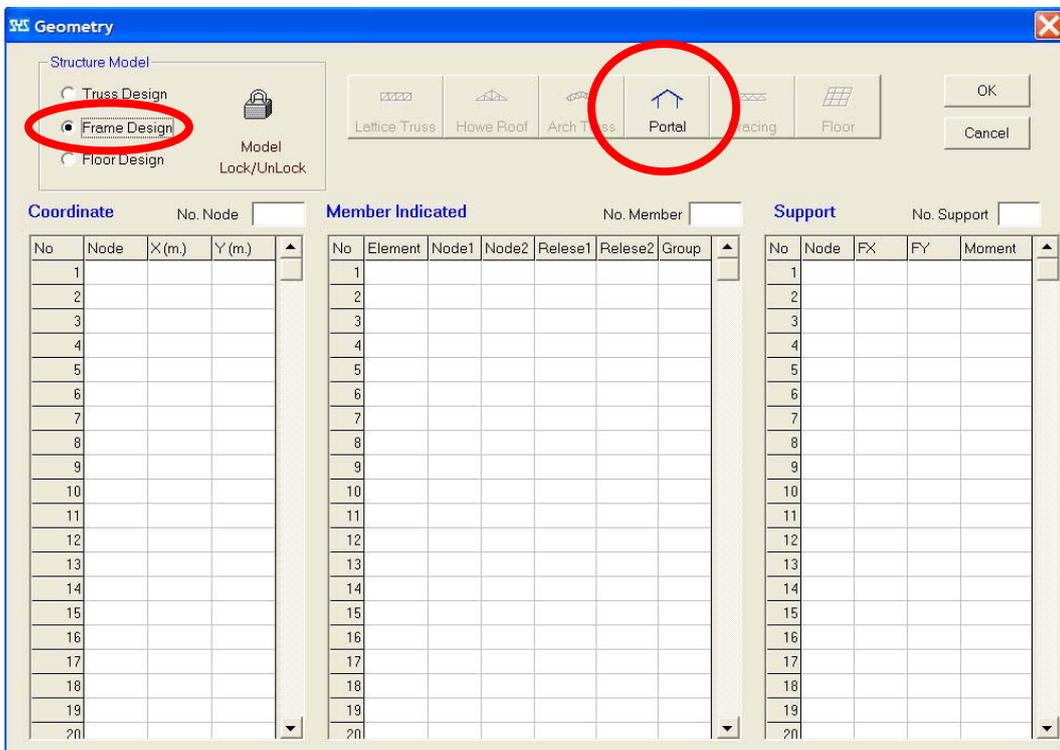


- 1) Steel frame design can be selected by clicking “Go to Program” and then clicking “OK”
- 2) Clicking “1.Geometry” button for input geometry of frame. In main menu page, there are “View” type for user to select display type of information in truss as described below

Node	:	Display node number of frame
Member	:	Display member number of frame
Load	:	Display load location, direction and magnitude
Support	:	Display support location

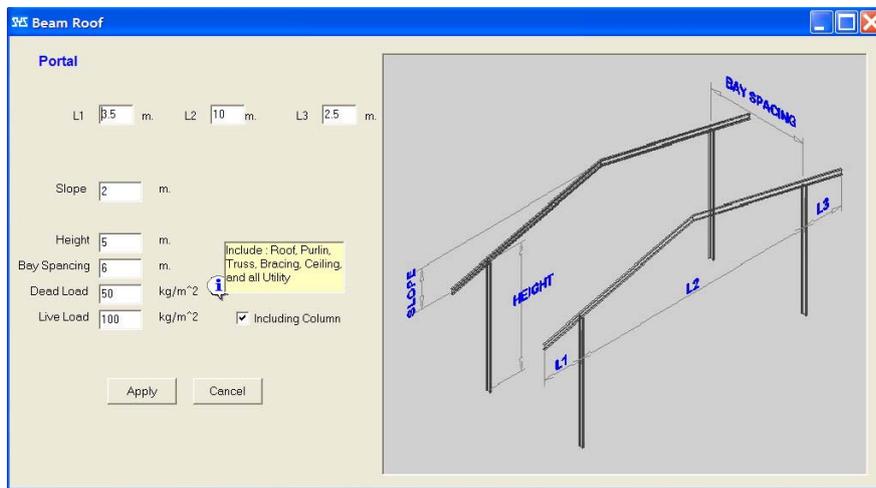


3) Clicking “Frame Design” and then select “Portal”.

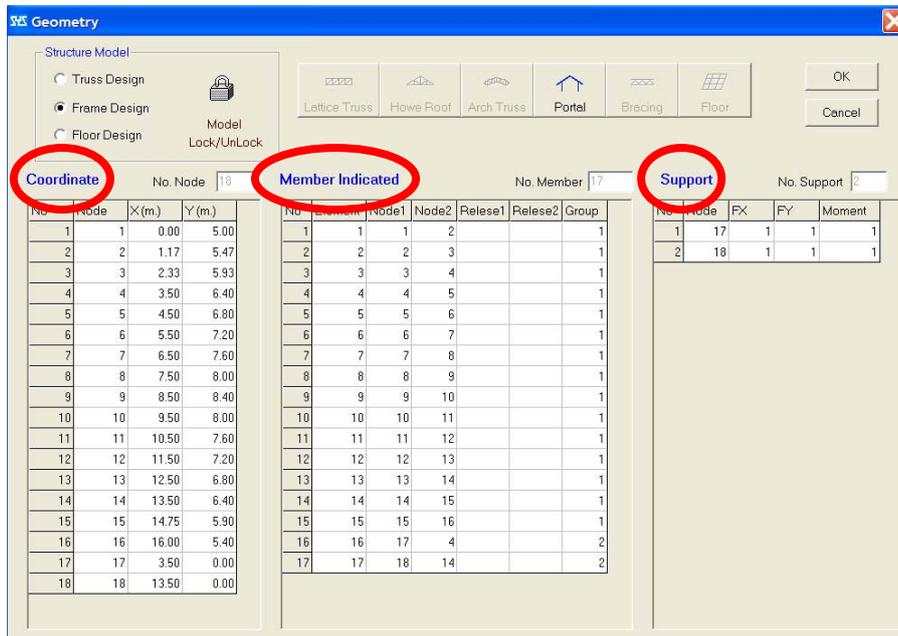


4) For portal frame design, variables are required to be filled as follows

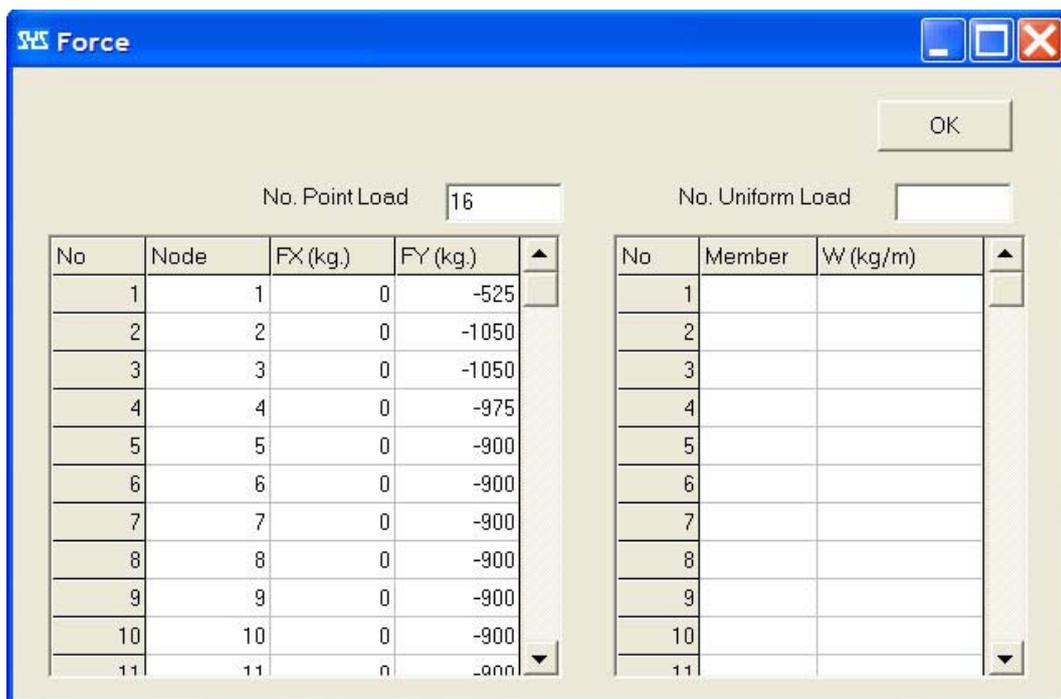
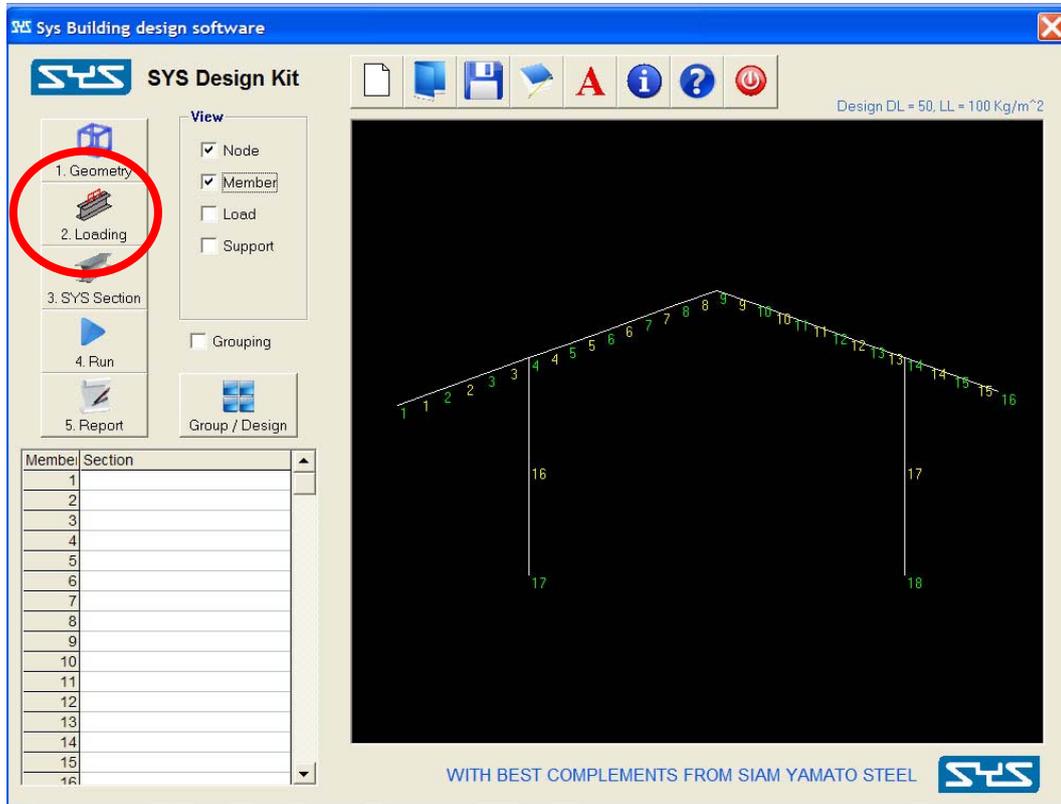
- L1 : Left cantiliver span length in meter
- L2 : Middle span length in meter
- L3 : Right cantiliver length in meter
- Slope : Different elevation (slope) between top of roof and top of column in meter
- Height (H1) : Height of column in meter
- Bay Spacing (S) : Spacing between bay of column in meter
- Dead load : Dead load in kilogram per square meter including load of roof, purlin, truss, bracing, ceiling and all utility
- Live load : Live load in kilogram per square meter
- Including column : Including or not including column in design



5) Then “Coordinate”, “Member indicated” and “Support” will be created in window and then clicking “OK” to go back to main menu. The user can edit any data of “Coordinate”, “Member indicated” and “Support” by double click and edit data in the window.

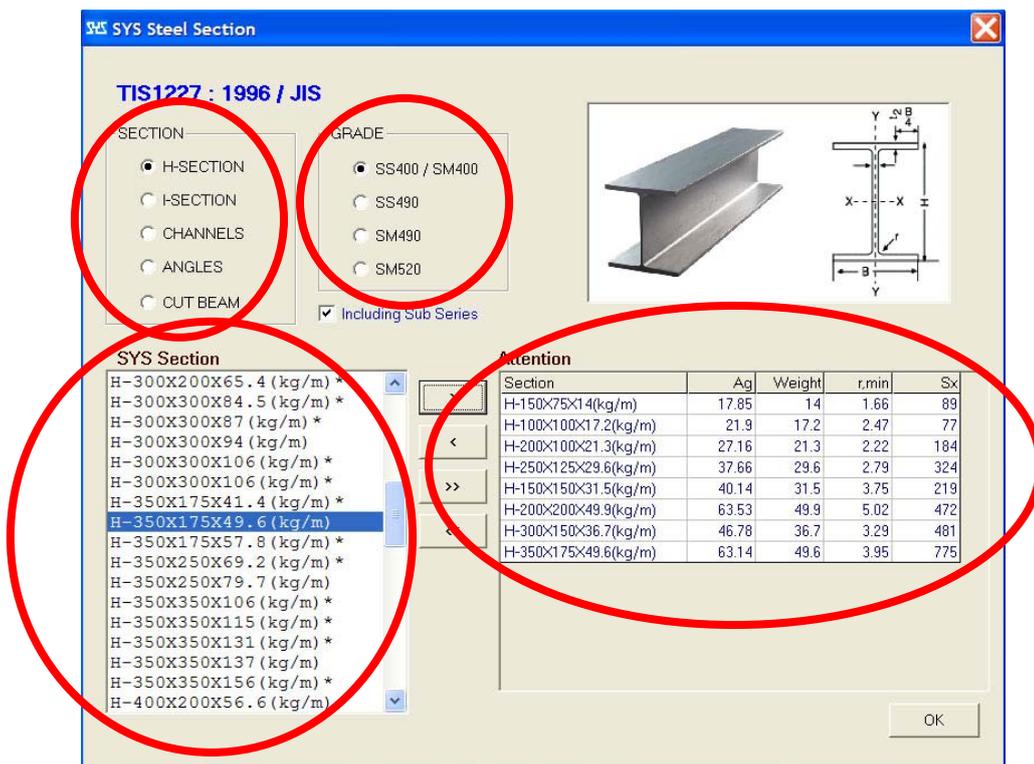
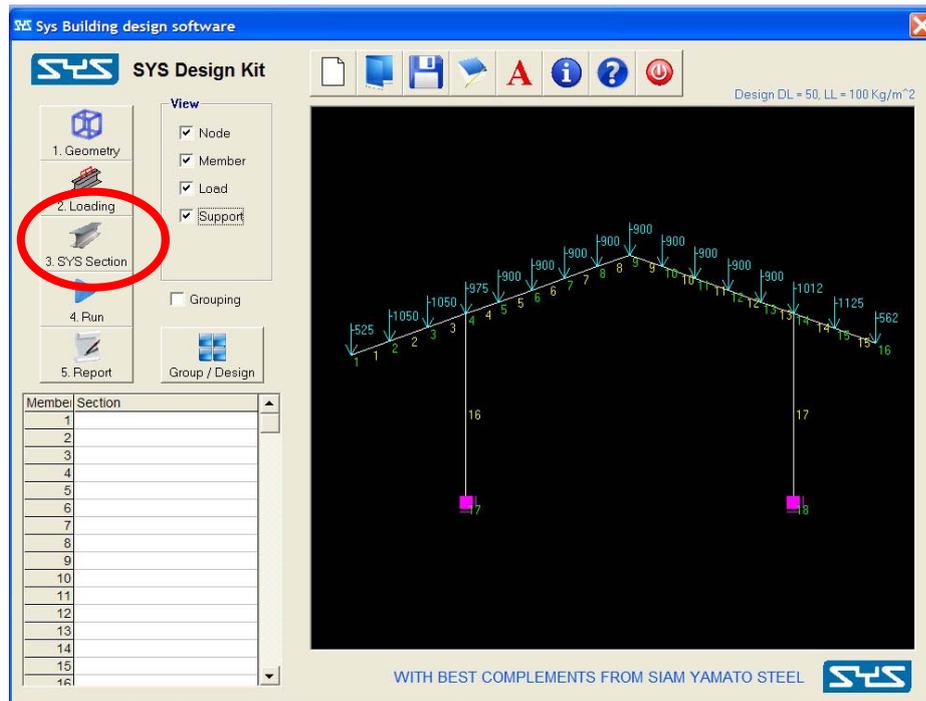


- 6) Clicking “2.Loading” button in main menu to go to edit load data and clicking “OK” to go back to main menu. Program will automatically generate point load from dead load and point load data in step 4. For other applied force such as wind load, seismic load, user can edit load data in window by clicking and editing load at the joint number.

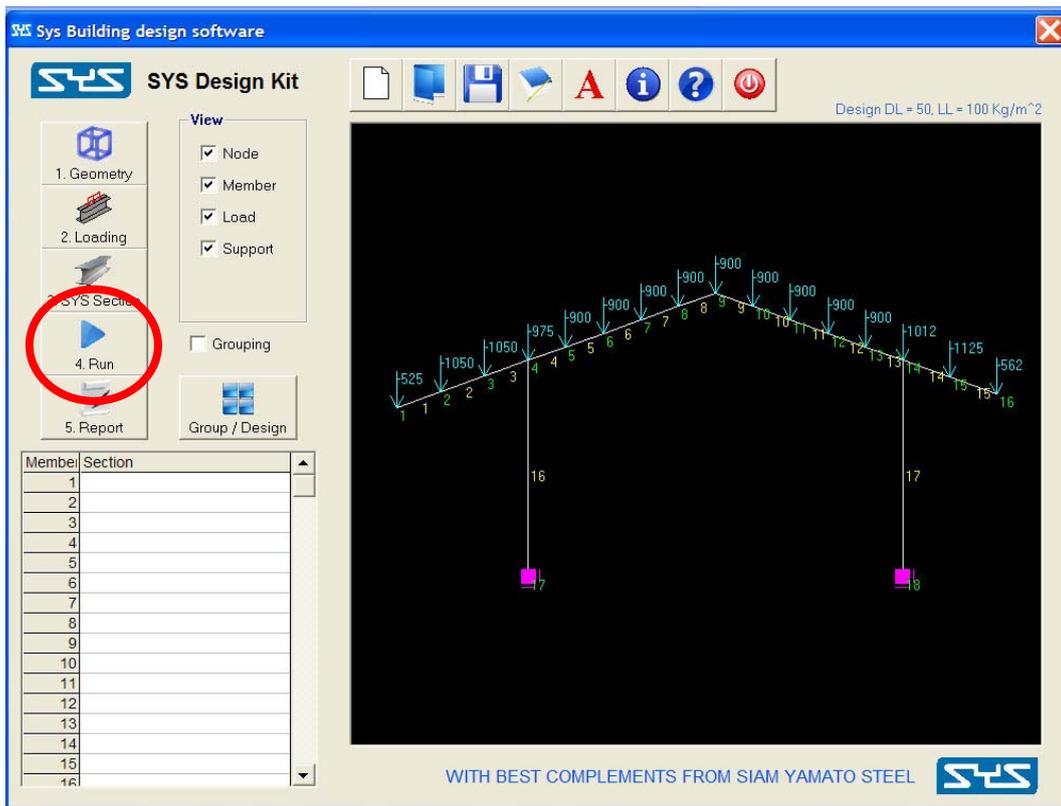


- 7) Select “SYS Section” button to go to input steel section. User can select “H-Section”, “I-Section”, “Channels”, “Angles” and “Cut beam” from left column to “Attention Table” in the right column for program to use this section to design. More steel section from SYS are provided in program by clicking “Including Sub Series”. The sub series sections are not normal sections in the market. Please ask SYS for more information.

Select “Steel Grade” of member by clicking buttons in front of type of steel grade.



8) Run the program by clicking “4.Run” button.



FINITE ELEMENT ANALYSIS

Member	Node	Axial	Shear-Y	Shear-Z	Torsion	Moment-Y	Moment-Z
1	NODE(1)	-1.9570E+02	4.8716E+02	0.0000E+00	0.0000E+00	0.0000E+00	-9.0949E-12
	MIDSPAN					0.0000E+00	3.0712E+02
2	NODE(2)	1.9570E+02	-4.8716E+02	0.0000E+00	0.0000E+00	0.0000E+00	6.1425E+02
	MIDSPAN	-5.8059E+02	1.4641E+03	0.0000E+00	0.0000E+00	0.0000E+00	-6.1425E+02
3	NODE(3)	5.8059E+02	-1.4641E+03	0.0000E+00	0.0000E+00	0.0000E+00	2.4412E+03
	MIDSPAN	-9.7849E+02	2.4358E+03	0.0000E+00	0.0000E+00	0.0000E+00	-2.4412E+03
4	NODE(4)	9.7849E+02	-2.4358E+03	0.0000E+00	0.0000E+00	0.0000E+00	5.5125E+03
	MIDSPAN	2.2018E+03	-3.7085E+03	0.0000E+00	0.0000E+00	0.0000E+00	-7.5620E+03
5	NODE(5)	-2.2018E+03	3.7085E+03	0.0000E+00	0.0000E+00	0.0000E+00	3.5679E+03
	MIDSPAN	1.8675E+03	-2.8729E+03	0.0000E+00	0.0000E+00	0.0000E+00	-3.5679E+03
6	NODE(6)	-1.8675E+03	2.8729E+03	0.0000E+00	0.0000E+00	0.0000E+00	4.7371E+02
	MIDSPAN	1.5333E+03	-2.0372E+03	0.0000E+00	0.0000E+00	0.0000E+00	-4.7371E+02
7	NODE(7)	-1.5333E+03	2.0372E+03	0.0000E+00	0.0000E+00	0.0000E+00	-1.7204E+03
	MIDSPAN	1.1990E+03	-1.2016E+03	0.0000E+00	0.0000E+00	0.0000E+00	1.7204E+03
8	NODE(8)	-1.1990E+03	1.2016E+03	0.0000E+00	0.0000E+00	0.0000E+00	-3.0146E+03
	MIDSPAN	8.6476E+02	-3.6597E+02	0.0000E+00	0.0000E+00	0.0000E+00	3.0146E+03

ANALYSIS STATUS: Finish

Design Back

10) The design member sections are shown member-section table.

Sys Building design software

SYS Design Kit

Design DL = 50. LL = 100 Kg/m²

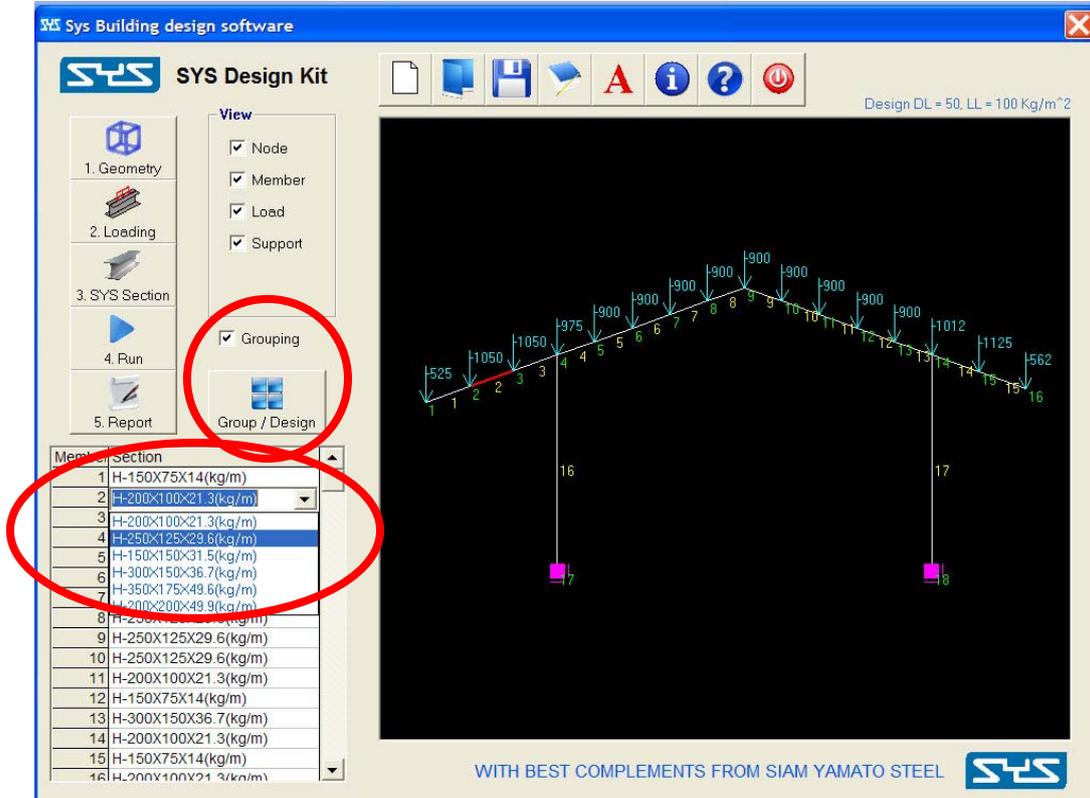
View: Node, Member, Load, Support, Grouping, Group / Design

1. Geometry, 2. Loading, 3. SYS Section, 4. Run, 5. Results

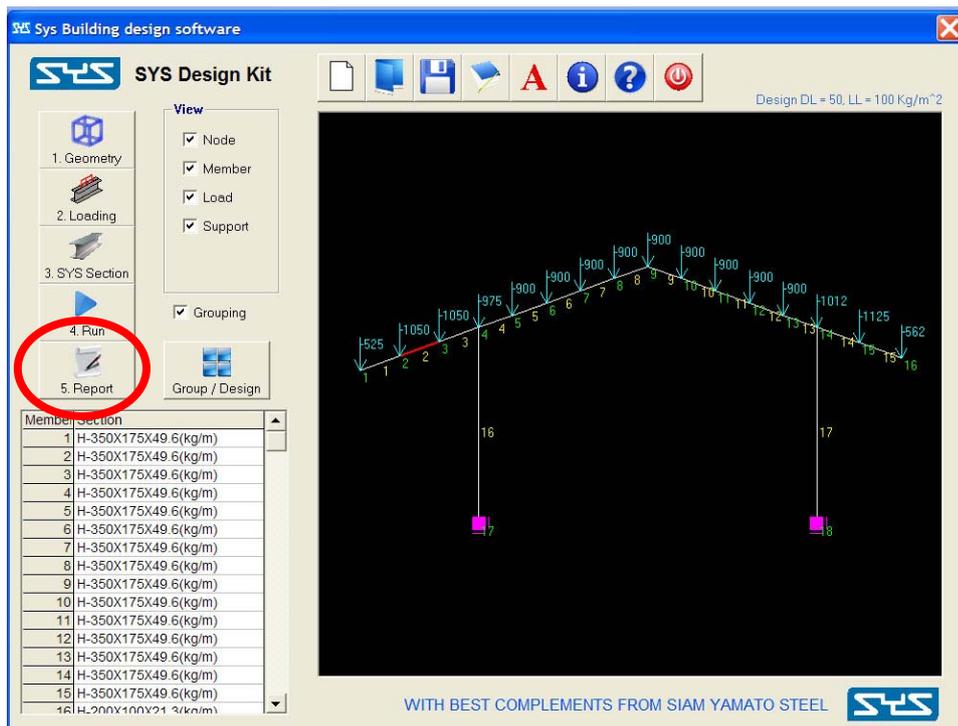
Member	Section
1	H-150X75X14(kg/m)
2	H-200X100X21.3(kg/m)
3	H-300X150X36.7(kg/m)
4	H-350X175X49.6(kg/m)
5	H-250X125X29.6(kg/m)
6	H-200X100X21.3(kg/m)
7	H-250X125X29.6(kg/m)
8	H-250X125X29.6(kg/m)
9	H-250X125X29.6(kg/m)
10	H-250X125X29.6(kg/m)
11	H-200X100X21.3(kg/m)
12	H-150X75X14(kg/m)
13	H-300X150X36.7(kg/m)
14	H-200X100X21.3(kg/m)
15	H-150X75X14(kg/m)
16	H-200X100X21.3(kg/m)

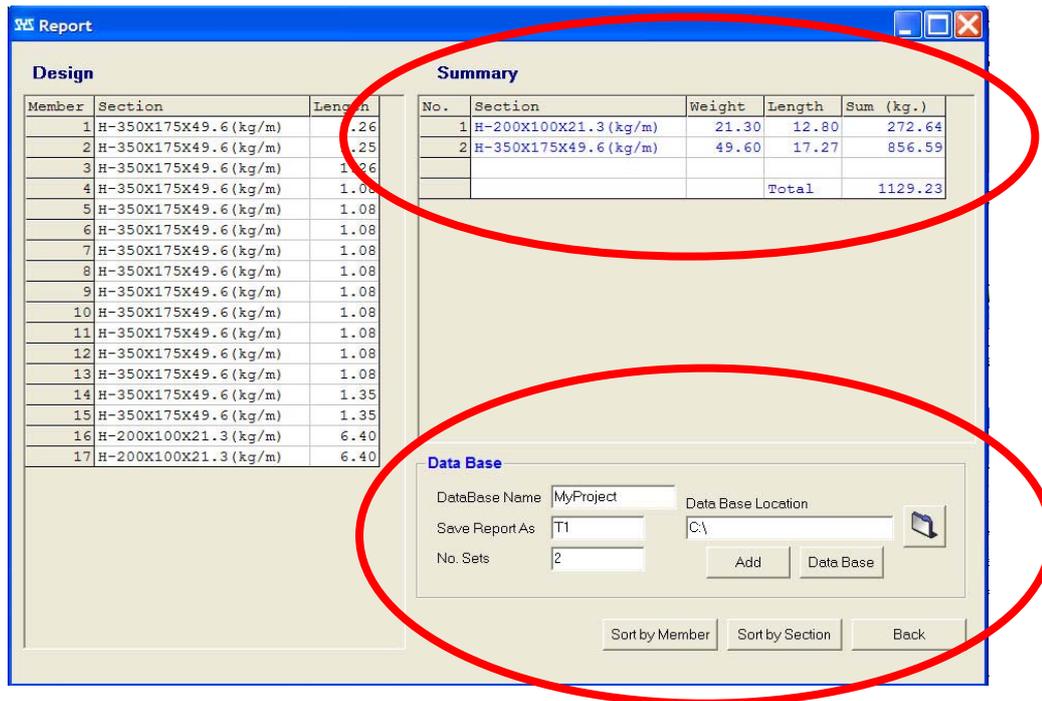
WITH BEST COMPLEMENTS FROM SIAM YAMATO STEEL

- 11) In practice, the beam section along frame length should be the same section for each member. Group for each type of member can selecting by checking in front of “Grouping” and clicking “Group/Design” button. User can edit member section for each member by double click at member section and change member section in the list.



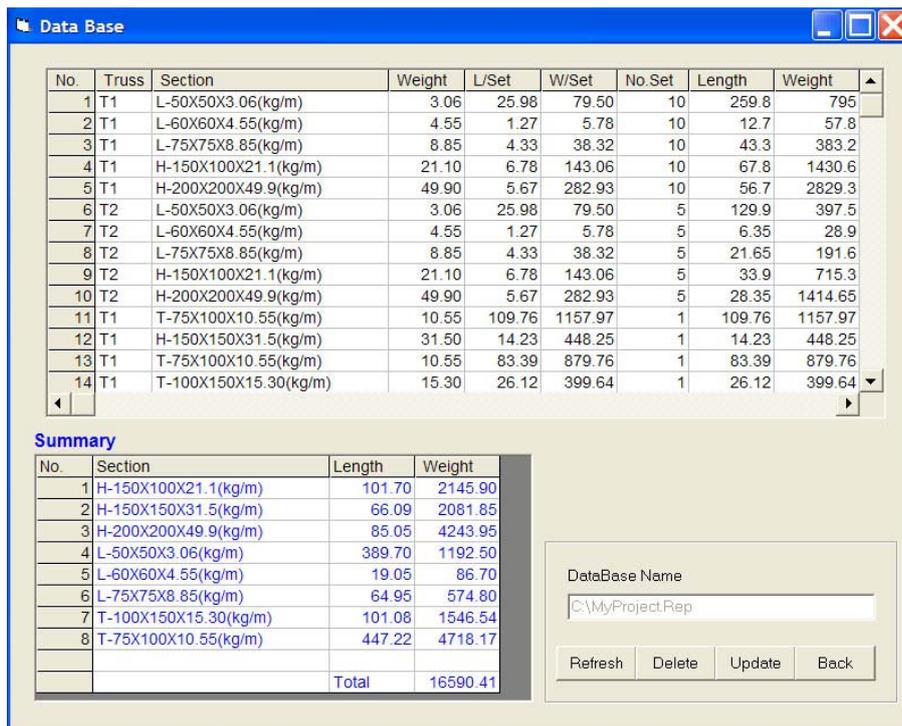
- 12) Clicking “5.Report” to see member design section, summary of steel section, summary of steel weight and save data to database.





- “Sort by Member” : Rearrange the design member table in left table by member number
- “Sort by Section” : Rearrange the design member table in left table by section of member
- “Data Base Location” : Select the location of data base to be saved
- “Data Base Name” : Name of data base to be saved
- “Save Report As” : Name of frame output to be saved
- “No. Sets” : Number set of frame in project

13) After clicking “Data Base” button, the summary of all frame section in database will show.



Steel Floor Design

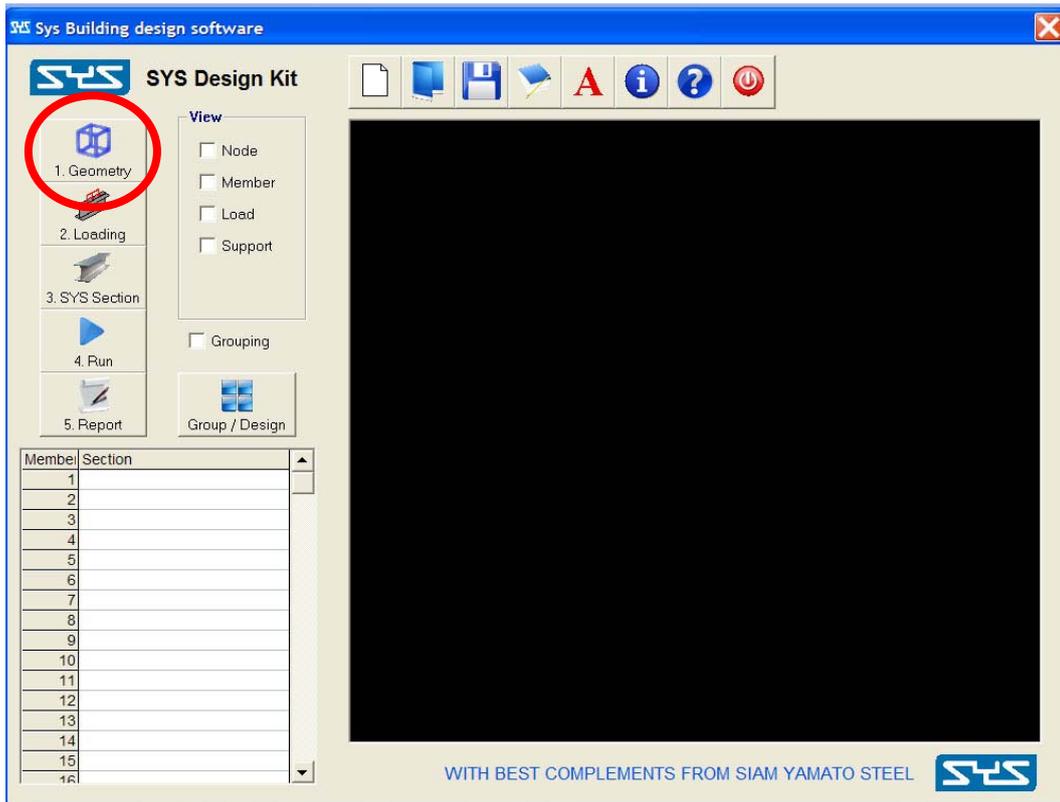
Steel floor design module provides the design of floor frame structure. All joints in model of floor are fixed joints. The dead load and live load are applied to frame member by uniform load along member. All members are carrying axial force, shear force and moment force from applied load. All information required for design such as floor geometry, support location, member section, type of slab design, dead load, live load and so on, shall be prepared and filled in the input box by users.

Input Mode

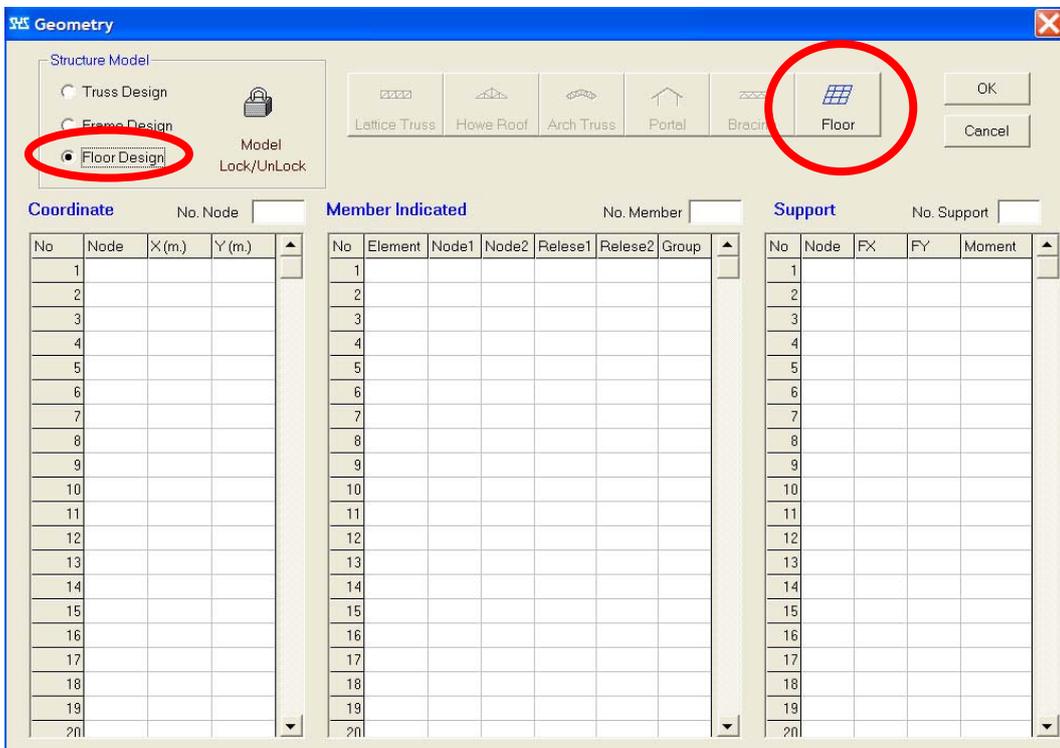


- 1) Steel floor design can be selected by clicking “Go to Program” and then clicking “OK”
- 2) Clicking “1.Geometry” button for input geometry of floor frame. In main menu page, there are “View” type for user to select display type of information in truss as described below

Node	:	Display node number of floor frame
Member	:	Display member number of floor frame
Load	:	Display load location, direction and magnitude
Support	:	Display support location

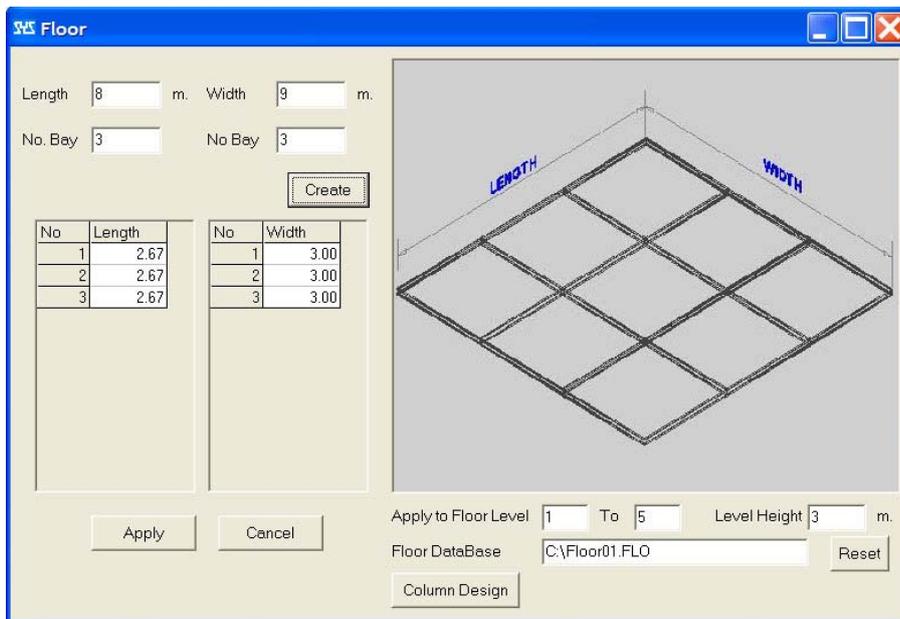


3) Clicking “Floor Design” and then select “Floor”.

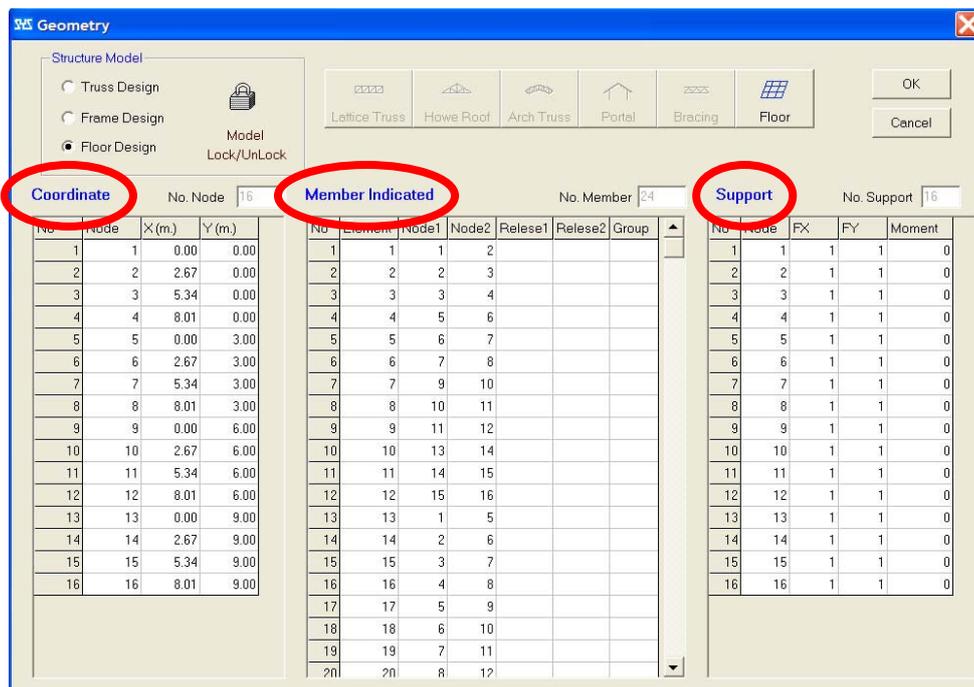


4) For floor design, variables are required to be filled as follows

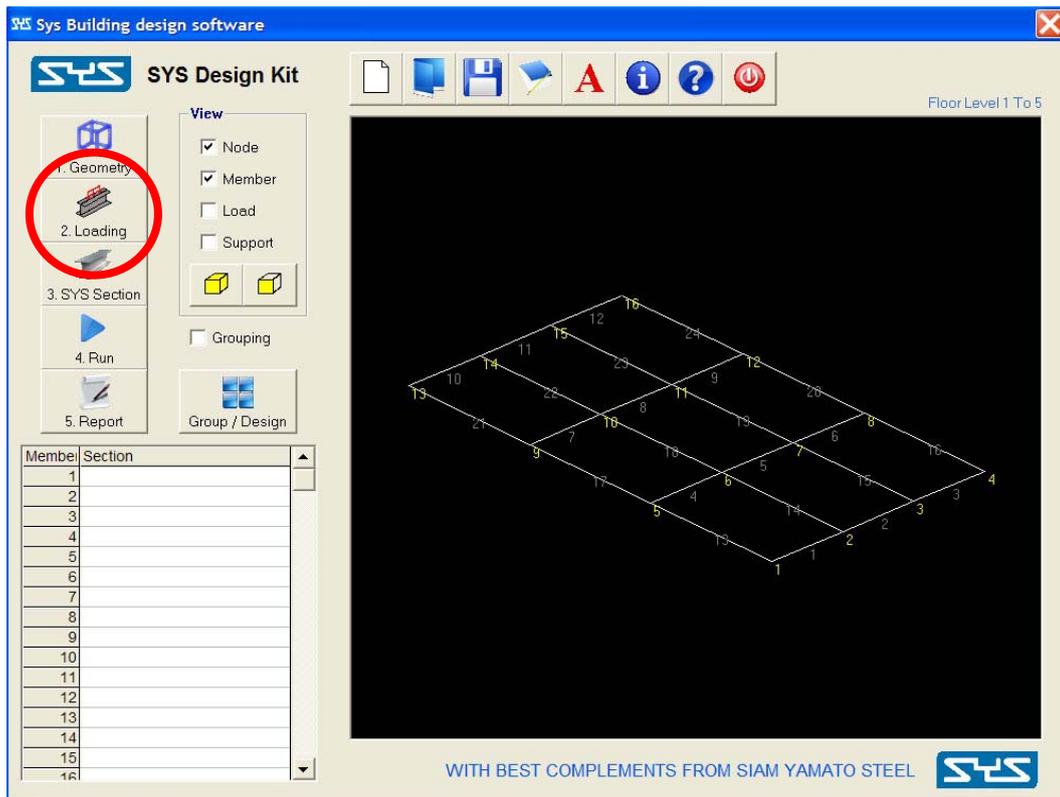
- Length : Length of floor in meter
- No. Bay : Number of bay in length direction
- Width : Width of floor in meter
- No. Bay : Number of bay in width direction
- Apply to Floor Level : Number of floor structure
- Level Height : Height of floor level



5) Then “Coordinate”, “Member indicated” and “Support” will be created in window and then clicking “OK” to go back to main menu. The user can edit any data of “Coordinate”, “Member indicated” and “Support” by double click and edit data in the window.



6) Clicking “2.Loading” button in main menu to go to input load data.



7) Input floor dead load, live load, number of floor load, number of wall load, wall load, type of slab, direction of slab load in input table and click “Create Load” to create uniform load to beam frame. Variables are required to be filled as follows

- No. Floor Load : Number of floor load
- Node 1 : First node number of floor load (shown in blue font)
- Node 2 : Second node number of floor load (shown in blue font)
- Node 3 : Third node number of floor load (shown in blue font)
- Node 4 : Fourth node number of floor load (shown in blue font)
- DL : Dead load in kilogram per square meter
- LL : Live load in kilogram per square meter
- No.way : Number of slab type (1 = one way, 2 = two way)
- Direction : Direction of one way slab (0= X-direction, 90= Y-direction)
- No. Wall Load : Number of wall load
- Member : Member number to applied wall load (shown in brown font)
- Weight : Weight of wall in kilogram per square meter
- Height : Height of wall in meter

Floor Loading

No. Floor Load: 5 No. Wall Load: 4

No	Node1	Node2	Node3	Node4	DL	LL	No.Way	Direction
2	14	15	10	11	240	200	2	0
3	15	16	11	12	300	220	2	0
4	9	10	5	6	240	200	1	90
5	5	6	1	2	222	300	1	0

No	Member	Weight	Height
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			
13			
14			
15			
16			
17			
18			
19			
20			
21			
22			
23			
24			

Unit
D,LL,L = Kg/m²
Weight = Kg/m.

Member Load

Member	Load
1	
2	
3	
4	
5	
6	
7	
8	
9	
10	
11	
12	
13	
14	
15	
16	
17	
18	
19	
20	
21	
22	
23	
24	

Floor Loading

No. Floor Load: 5 No. Wall Load: 2

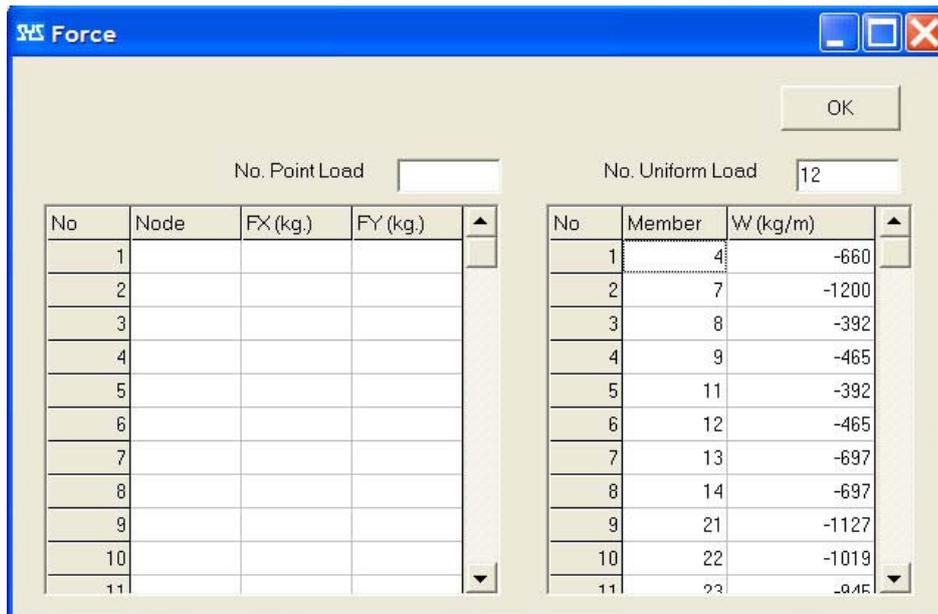
No	Node1	Node2	Node3	Node4	DL	LL	No.Way	Direction
2	14	15	10	11	240	200	2	0
3	15	16	11	12	300	222	2	0
4	9	10	5	6	240	200	1	90
5	5	6	1	2	222	300	1	0

No	Member	Weight	Height
1	21	180	3
2	7	180	3

Unit
D,LL,L = Kg/m²
Weight = Kg/m.

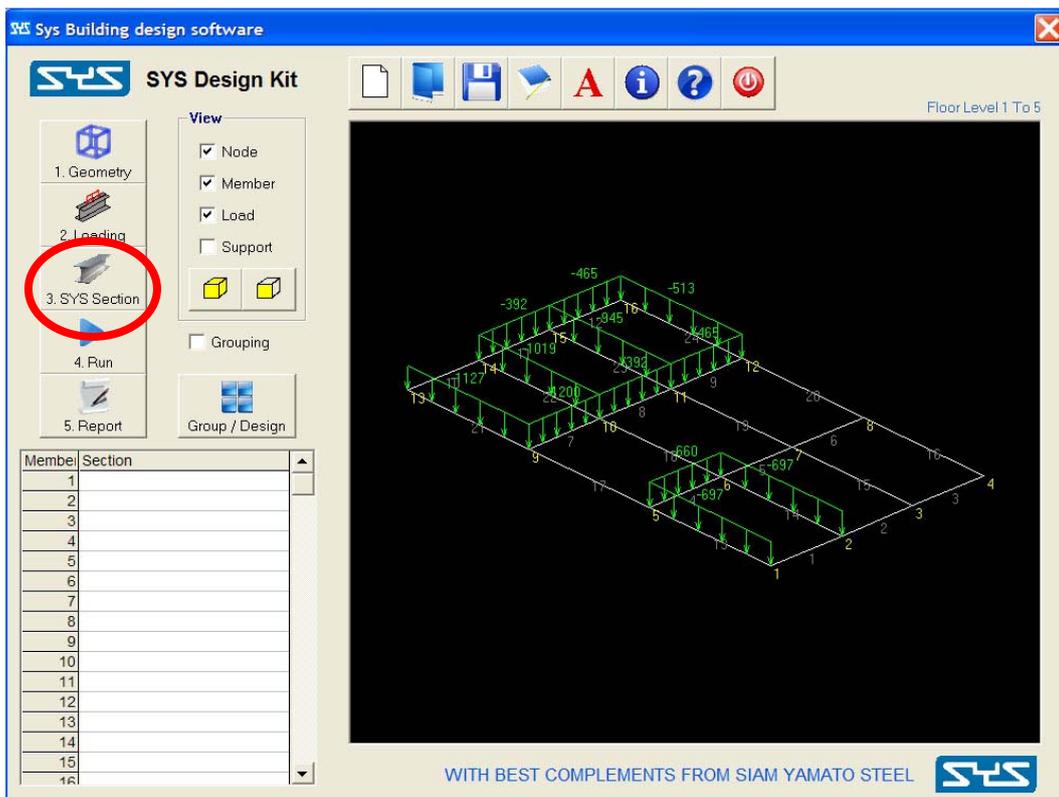
Member Load

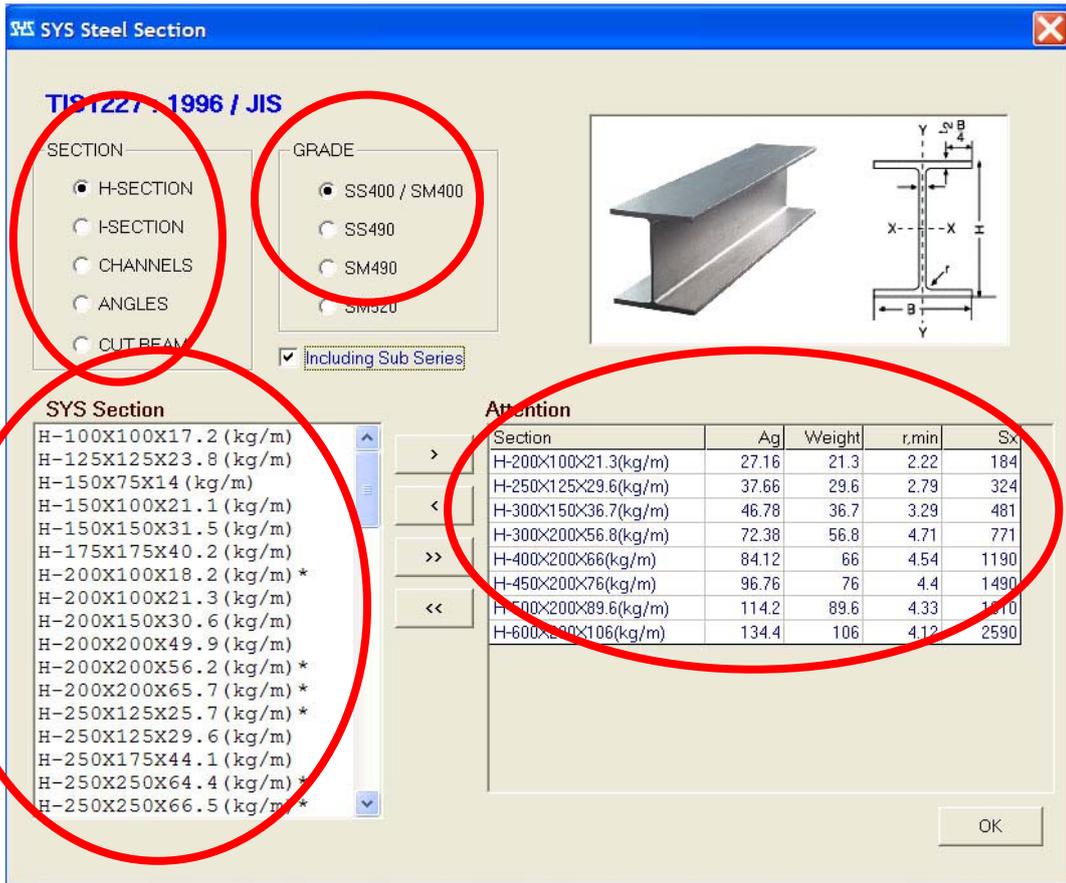
Member	Load
1	0
2	
3	
4	-660
5	
6	
7	-1200
8	-392
9	-465
10	0
11	-392
12	-465
13	-697
14	-697
15	
16	
17	0
18	0
19	
20	
21	-1127
22	-1019
23	-945
24	-513



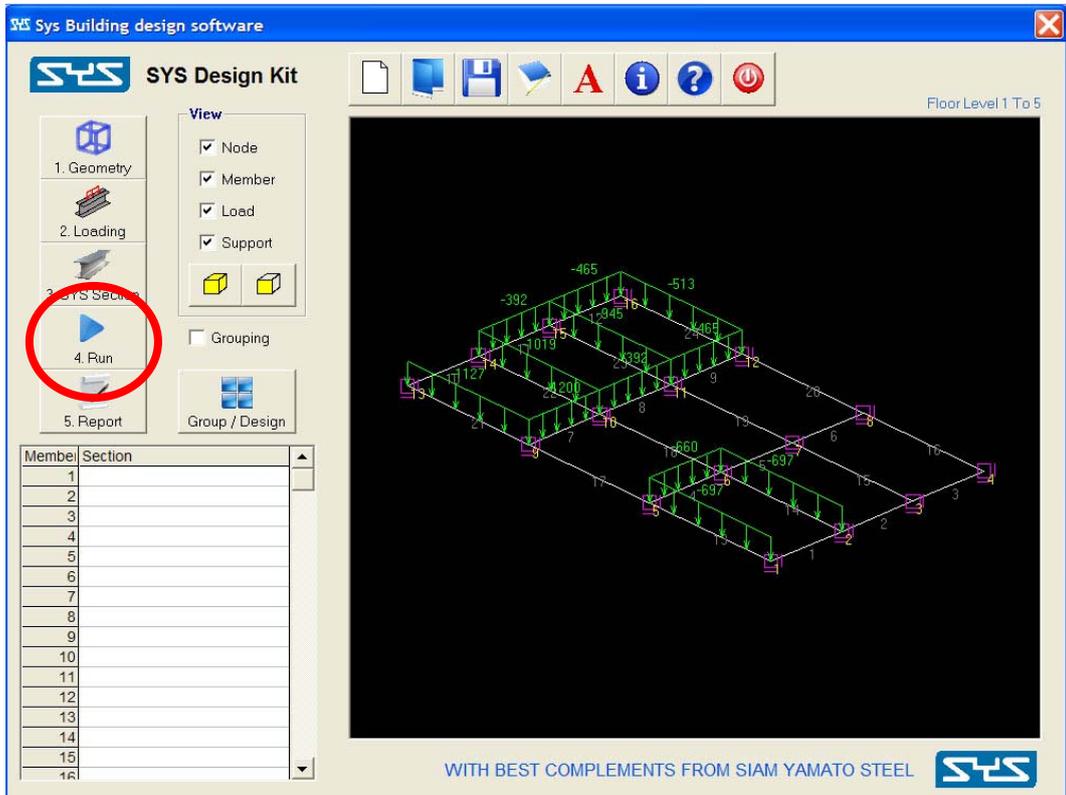
8) Select “SYS Section” button to go to input steel section. User can select “H-Section”, “I-Section”, “Channels”, “Angles” and “Cut beam” from left column to “Attention Table” in the right column for program to use this section to design. More steel section from SYS are provided in program by clicking “Including Sub Series”. The sub series sections are not normal sections in the market. Please ask SYS for more information.

Select “Steel Grade” of member by clicking buttons in front of type of steel grade.





9) Run the program by clicking “4.Run” button.



FINITE ELEMENT ANALYSIS

Member	Node	Axial	Shear-Y	Shear-Z	Torsion	Moment-Y	Moment-Z
1	NODE(1)	0.0000E+00	0.0000E+00	-2.2067E-05	1.4871E-06	6.2721E-05	0.0000E+00
	MIDSPAN					-3.3261E-05	0.0000E+00
	NODE(2)	0.0000E+00	0.0000E+00	2.2067E-05	-1.4871E-06	-3.8013E-06	0.0000E+00
2	NODE(2)	0.0000E+00	0.0000E+00	1.4237E-05	1.0659E-04	-3.6112E-05	0.0000E+00
	MIDSPAN					1.7106E-05	0.0000E+00
	NODE(3)	0.0000E+00	0.0000E+00	-1.4237E-05	-1.0659E-04	-1.9007E-06	0.0000E+00
3	NODE(3)	0.0000E+00	0.0000E+00	-2.8474E-06	5.9485E-06	1.3305E-05	0.0000E+00
	MIDSPAN					-9.5033E-06	0.0000E+00
	NODE(4)	0.0000E+00	0.0000E+00	2.8474E-06	-5.9485E-06	-5.7020E-06	0.0000E+00
4	NODE(5)	0.0000E+00	0.0000E+00	7.6362E+02	-2.9742E-06	-1.7546E-05	0.0000E+00
	MIDSPAN					-4.3130E+02	0.0000E+00
	NODE(6)	0.0000E+00	0.0000E+00	9.9858E+02	2.9742E-06	3.1367E+02	0.0000E+00
5	NODE(6)	0.0000E+00	0.0000E+00	1.4685E+02	-6.9220E-05	-3.1367E+02	0.0000E+00
	MIDSPAN					1.1763E+02	0.0000E+00
	NODE(7)	0.0000E+00	0.0000E+00	-1.4685E+02	6.9220E-05	-7.8418E+01	0.0000E+00
6	NODE(7)	0.0000E+00	0.0000E+00	-2.9370E+01	-1.1897E-05	7.8418E+01	0.0000E+00
	MIDSPAN					-3.9209E+01	0.0000E+00
	NODE(8)	0.0000E+00	0.0000E+00	2.9370E+01	1.1897E-05	-3.2994E-05	0.0000E+00
7	NODE(9)	0.0000E+00	0.0000E+00	1.3568E+03	1.0410E-05	-1.5921E-04	0.0000E+00
	MIDSPAN					-7.4194E+02	0.0000E+00
	NODE(10)	0.0000E+00	0.0000E+00	1.8472E+03	-1.0410E-05	6.5479E+02	0.0000E+00
8	NODE(10)	0.0000E+00	0.0000E+00	6.8686E+02	2.6327E-05	-6.5479E+02	0.0000E+00

ANALYSIS STATUS: Finish

Design Back

11) The design member sections are shown member-section table.

SYS Building design software

SYS Design Kit

View: Node, Member, Load, Support

1. Geometry, 2. Loading, 3. SYS Section, 4. Run, 5. Report

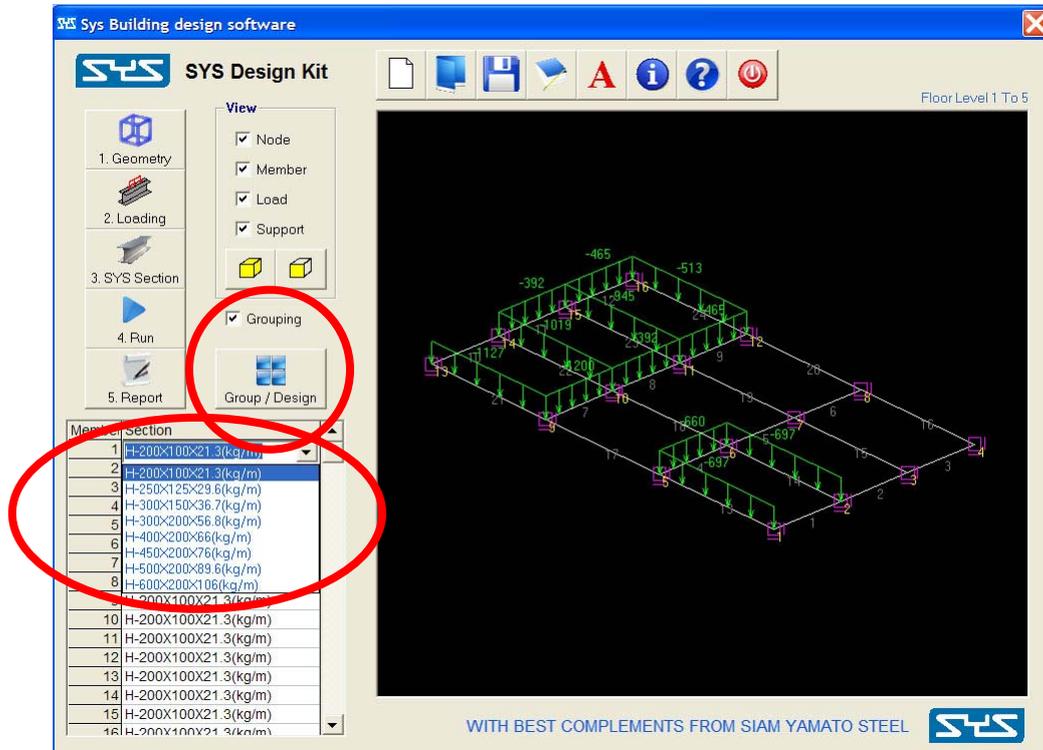
Group / Design

Member	Section
1	H-200X100X21.3(kg/m)
2	H-200X100X21.3(kg/m)
3	H-200X100X21.3(kg/m)
4	H-200X100X21.3(kg/m)
5	H-200X100X21.3(kg/m)
6	H-200X100X21.3(kg/m)
7	H-200X100X21.3(kg/m)
8	H-200X100X21.3(kg/m)
9	H-200X100X21.3(kg/m)
10	H-200X100X21.3(kg/m)
11	H-200X100X21.3(kg/m)
12	H-200X100X21.3(kg/m)
13	H-200X100X21.3(kg/m)
14	H-200X100X21.3(kg/m)
15	H-200X100X21.3(kg/m)
16	H-200X100X21.3(kg/m)

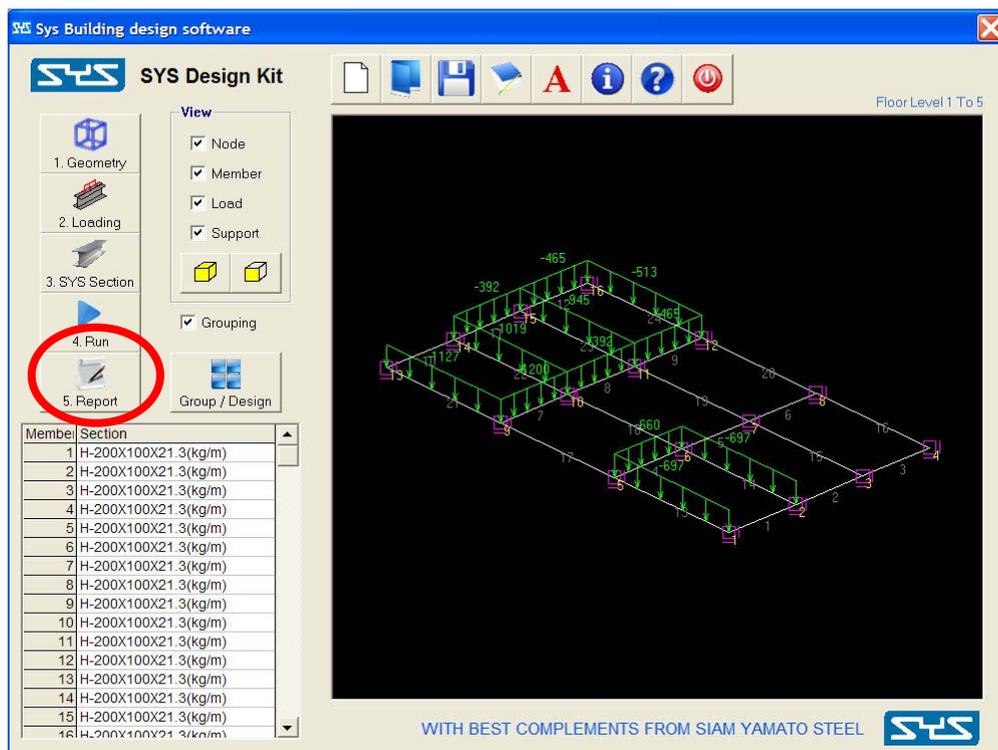
Floor Level 1 To 5

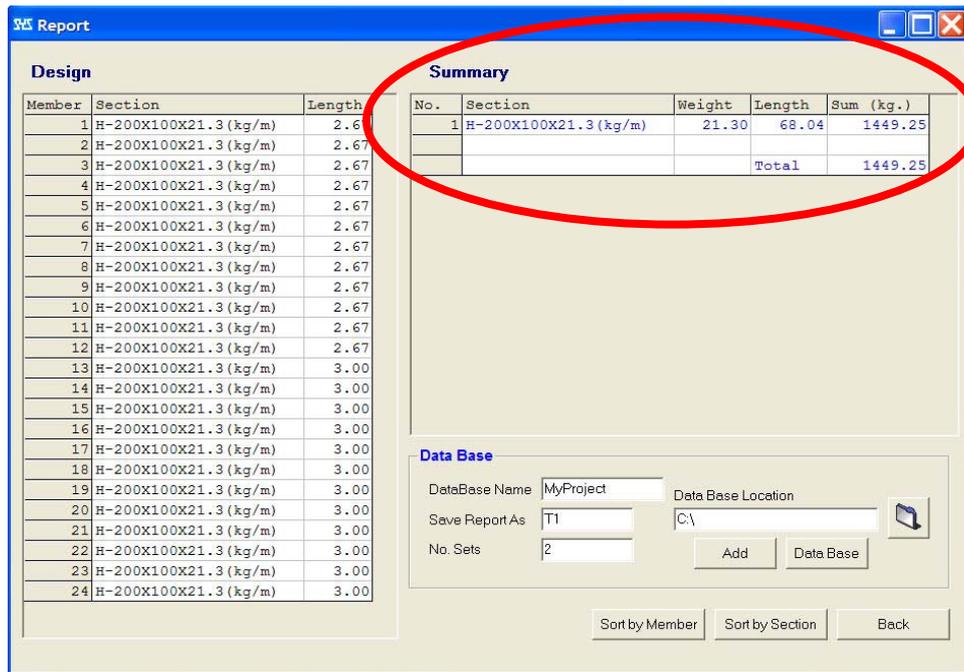
WITH BEST COMPLEMENTS FROM SIAM YAMATO STEEL

12) In practice, the beam section along frame length should be the same section for each member. Group for each type of member can selecting by checking in front of “Grouping” and clicking “Group/Design” button. User can edit member section for each member by double click at member section and change member section in the list.



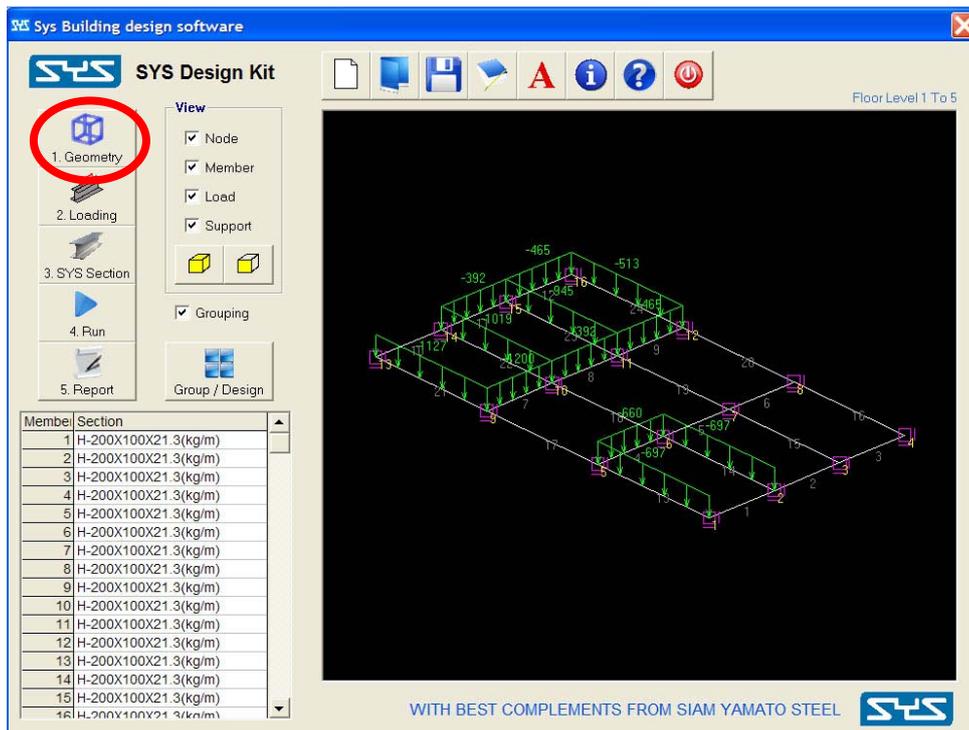
13) Clicking “5.Report” to see member design section, summary of steel section, summary of steel weight and save data to database.





- “Sort by Member” : Rearrange the design member table in left table by member number
- “Sort by Section” : Rearrange the design member table in left table by section of member
- “Data Base Location” : Select the location of data base to be saved
- “Data Base Name” : Name of data base to be saved
- “Save Report As” : Name of floor output to be saved
- “No. Sets” : Number set of floor in project

14) For column design, user has to go back to Floor input by click “1.Geometry”, click “Floor” and click “Column Design” in Floor input.



Geometry

Structure Model

Truss Design
 Frame Design
 Floor Design

Model Lock/UnLock

Lattice Truss
 Howe Roof
 Arch Truss
 Portal
 Bracing
 Floor

OK
Cancel

Coordinate No. Node 16

No	Node	X(m)	Y(m)
1	1	0.00	0.00
2	2	2.67	0.00
3	3	5.34	0.00
4	4	8.01	0.00
5	5	0.00	3.00
6	6	2.67	3.00
7	7	5.34	3.00
8	8	8.01	3.00
9	9	0.00	6.00
10	10	2.67	6.00
11	11	5.34	6.00
12	12	8.01	6.00
13	13	0.00	9.00
14	14	2.67	9.00
15	15	5.34	9.00
16	16	8.01	9.00

Member Indicated No. Member 24

No	Element	Node1	Node2	Release1	Release2	Group
1	1	1	2			
2	2	2	3			
3	3	3	4			
4	4	4	5			
5	5	5	6			
6	6	6	7			
7	7	7	8			
8	8	8	9			
9	9	9	10			
10	10	10	11			
11	11	11	12			
12	12	12	13			
13	13	13	14			
14	14	14	15			
15	15	15	16			
16	16	16	1			
17	17	17	2			
18	18	18	3			
19	19	19	4			
20	20	20	5			

Support No. Support 16

No	Node	FX	FY	Moment
1	1	1	1	0
2	2	1	1	0
3	3	1	1	0
4	4	1	1	0
5	5	1	1	0
6	6	1	1	0
7	7	1	1	0
8	8	1	1	0
9	9	1	1	0
10	10	1	1	0
11	11	1	1	0
12	12	1	1	0
13	13	1	1	0
14	14	1	1	0
15	15	1	1	0
16	16	1	1	0

Floor

Length 8 m. Width 9 m.

No. Bay 3 No. Bay 3

Create

No	Length
1	2.67
2	2.67
3	2.67

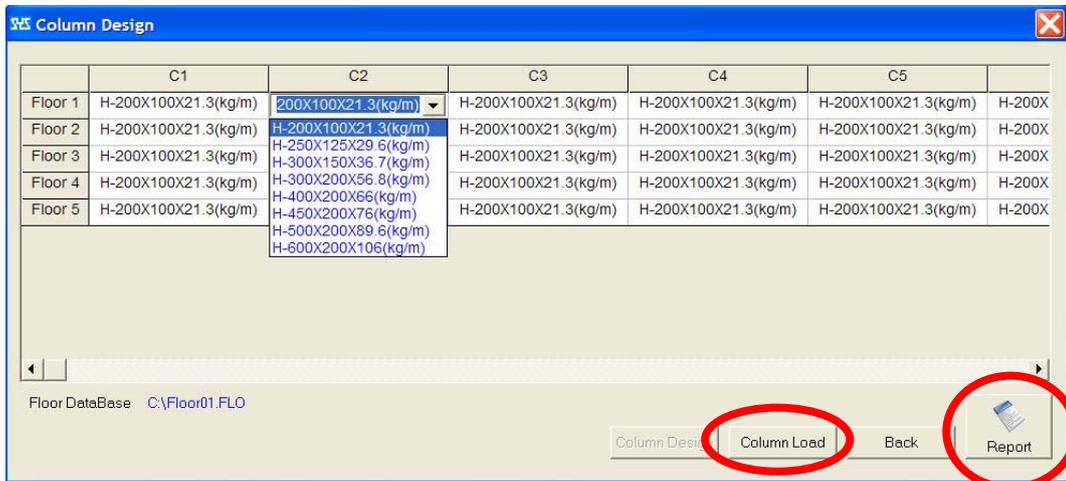
No	Width
1	3.00
2	3.00
3	3.00

Apply to Floor Level 1 To 5 Level Height 3 m.

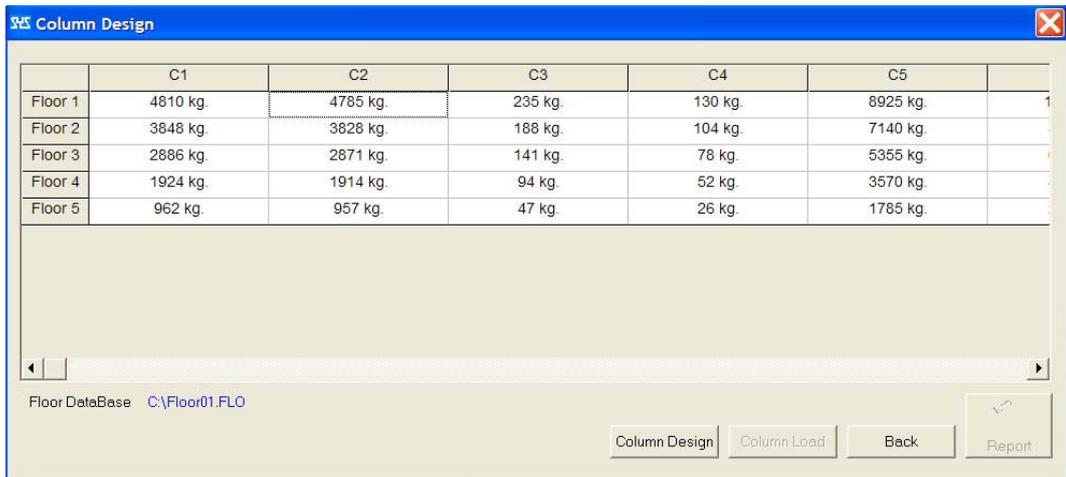
Floor DataBase C:\Floor01.FLO Reset

Column Design

15) The column design members will show in table. The vertical data in table shows the column number. The horizontal data in table shows the floor number of column. The user can edit member property of each member by click and edit data in table.



16) User can see column load by click “Column Load” button. Otherwise, click “Report” to see member design section, summary of column section, summary of steel weight and save data to database.



- “Sort by Member” : Rearrange the design member table in left table by member number
- “Sort by Section” : Rearrange the design member table in left table by section of member
- “Data Base Location” : Select the location of data base to be saved
- “Data Base Name” : Name of data base to be saved
- “Save Report As” : Name of floor column output to be saved
- “No. Sets” : Number set of floor column in project

Design

Member	Section	Length
1-C01	H-200X100X21.3 (kg/m)	3.00
2-C02	H-200X100X21.3 (kg/m)	3.00
3-C03	H-200X100X21.3 (kg/m)	3.00
4-C04	H-200X100X21.3 (kg/m)	3.00
5-C05	H-200X100X21.3 (kg/m)	3.00
6-C06	H-200X100X21.3 (kg/m)	3.00
7-C07	H-200X100X21.3 (kg/m)	3.00
8-C08	H-200X100X21.3 (kg/m)	3.00
9-C09	H-250X125X29.6 (kg/m)	3.00
10-C10	H-250X125X29.6 (kg/m)	3.00
11-C11	H-200X100X21.3 (kg/m)	3.00
12-C12	H-200X100X21.3 (kg/m)	3.00
13-C13	H-200X100X21.3 (kg/m)	3.00
14-C14	H-200X100X21.3 (kg/m)	3.00
15-C15	H-200X100X21.3 (kg/m)	3.00
16-C16	H-200X100X21.3 (kg/m)	3.00
17-C01	H-200X100X21.3 (kg/m)	3.00
18-C02	H-200X100X21.3 (kg/m)	3.00
19-C03	H-200X100X21.3 (kg/m)	3.00
20-C04	H-200X100X21.3 (kg/m)	3.00
21-C05	H-200X100X21.3 (kg/m)	3.00
22-C06	H-200X100X21.3 (kg/m)	3.00
23-C07	H-200X100X21.3 (kg/m)	3.00
24-C08	H-200X100X21.3 (kg/m)	3.00
25-C09	H-200X100X21.3 (kg/m)	3.00
26-C10	H-250X125X29.6 (kg/m)	3.00

Summary

No.	Section	Weight	Length	Sum (kg.)
1	H-200X100X21.3 (kg/m)	21.30	231.00	4920.30
2	H-250X125X29.6 (kg/m)	29.60	9.00	266.40
Total				5186.70

Data Base

DataBase Name: MyProject Data Base Location: C:\

Save Report As: T1

No. Sets: 2

Buttons: Add, Data Base, Sort by Member, Sort by Section, Back

17) After clicking “Data Base” button, the summary of all floor member in database will show.

Data Base

No.	Truss	Section	Weight	L/Set	W/Set	No. Set	Length	Weight
1	T1	L-50X50X3.06(kg/m)	3.06	25.98	79.50	10	259.8	795
2	T1	L-60X60X4.55(kg/m)	4.55	1.27	5.78	10	12.7	57.8
3	T1	L-75X75X8.85(kg/m)	8.85	4.33	38.32	10	43.3	383.2
4	T1	H-150X100X21.1(kg/m)	21.10	6.78	143.06	10	67.8	1430.6
5	T1	H-200X200X49.9(kg/m)	49.90	5.67	282.93	10	56.7	2829.3
6	T2	L-50X50X3.06(kg/m)	3.06	25.98	79.50	5	129.9	397.5
7	T2	L-60X60X4.55(kg/m)	4.55	1.27	5.78	5	6.35	28.9
8	T2	L-75X75X8.85(kg/m)	8.85	4.33	38.32	5	21.65	191.6
9	T2	H-150X100X21.1(kg/m)	21.10	6.78	143.06	5	33.9	715.3
10	T2	H-200X200X49.9(kg/m)	49.90	5.67	282.93	5	28.35	1414.65
11	T1	T-75X100X10.55(kg/m)	10.55	109.76	1157.97	1	109.76	1157.97
12	T1	H-150X150X31.5(kg/m)	31.50	14.23	448.25	1	14.23	448.25
13	T1	T-75X100X10.55(kg/m)	10.55	83.39	879.76	1	83.39	879.76
14	T1	T-100X150X15.30(kg/m)	15.30	26.12	399.64	1	26.12	399.64

Summary

No.	Section	Length	Weight
1	H-150X100X21.1(kg/m)	101.70	2145.90
2	H-150X150X31.5(kg/m)	66.09	2081.85
3	H-200X200X49.9(kg/m)	85.05	4243.95
4	L-50X50X3.06(kg/m)	389.70	1192.50
5	L-60X60X4.55(kg/m)	19.05	86.70
6	L-75X75X8.85(kg/m)	64.95	574.80
7	T-100X150X15.30(kg/m)	101.08	1546.54
8	T-75X100X10.55(kg/m)	447.22	4718.17
Total			16590.41

Data Base

DataBase Name: C:\MyProject.Rep

Buttons: Refresh, Delete, Update, Back