

Computer Analysis of Structures for Architectural Technology Students

Dr. Pattabhi Sitaram, Baker College of Flint

Dr. Sitaram has held leadership positions in both academia and industry. He is currently Professor, Program Director, and Chair of Engineering at Baker College Flint, Michigan, USA. He manages three ABET accredited programs- Mechanical, Civil, and Electrical engineering. He has 13 years full-time teaching experience and 15 years industry experience. He has taught extensively which includes more than 30 undergraduate, graduate, and lab courses in Civil and Mechanical Engineering in the areas of engineering mechanics, solid mechanics, structural engineering, structural design, vehicle crashworthiness, and plates & shells.

He worked in the automotive industry, primarily at GM, as crashworthiness product and simulation methods development engineer using nonlinear finite element analysis. He was the crashworthiness lead, GoFast lead, and subject matter expert (SME) for the material characterization of nonmetals such as plastics, foam, and rubber. He provided the early design for the front structure of few vehicles that passed the high speed internal test requirements, and also obtained Best Pick ratings (the results were shown on "Dateline" NBC TV program) in the tests conducted by IIHS (Insurance Institute of Highway and Safety) and 5 star ratings based on the federal tests by NHTSA (National Highway Traffic Safety Administration). Based on his recommendation for improving vehicle performance for crashworthiness, for the first time in the history of North America thick Dual Phase steels of high energy absorbing components were welded to each other, and not a single weld failed in the high speed full vehicle frontal impact tests. At GM, along with another engineer, he co-designed several component tests to improve upper interior head impact protection (FMVSS201u). It was the first time in the history of GM such components were designed. He was also one of the crashworthiness leads in the Fuel Efficient Light Vehicles programs that led to significant mass savings through use of advance materials, thorough understanding of crash load paths, and optimization.

Mr. Athanas Mutani Manyama, Baker College

Mr. Athanas Manyama is an Associate Professor at Baker College Flint, Michigan, USA and has been a full-time faculty member at Baker for 27 years. He was the Program Coordinator for the Architectural/Construction Technology (ACT) and Architectural Technology (AT) Programs for the Baker College System up to 2019. During that time, he also was the Chairperson for the Advisory Board for the programs. He worked with Baker College and the national American Institute of Architecture Students (AIAS) to organize, register and advise the American Institute of Architecture Students - Baker College Flint (AIAS-BCF) from 2005 to 2019. He was an adjunct faculty member at the same institution from 1990 – 1993. He has extensive experience in teaching various courses in areas such as architectural design, building codes and construction, structural analysis, structural design, building cost estimating, land surveying, CADD applications, and statistics.

He is a graduate of the Master of Science (M.Sc.) and Bachelor of Science (B.Sc.) in Architectural and Building Engineering from Technical University of Budapest, Hungary.

His institutional and community services include supervising and judging the Science Olympiad competitions for local schools, participating as advisory board member at local schools, member of the JEF Grant Committee, and presenter at the Career Exploration and STEMM programs.

Computer Analysis of Structures for Architectural Technology Students

ABSTRACT

Architectural technology students at our institution, Baker College Flint, Michigan, USA, opt for one of the two career pathways- a two-year Associate of Applied Science (AAS) degree or a four-year Bachelor of Science degree (BS). The AAS degree gives graduating students the opportunity to join workforce, or the BS program (usually at the junior level) at our school or elsewhere. Our students in both AAS and BS programs take the Structural Analysis I course. This course mainly involves hand calculations for the analysis of structures such as trusses, beams, frames, and columns. The BS program students additionally take the Structural Design I and II courses. The Structural Design I course deals with the analysis and design of reinforced concrete and masonry structures, and the Structural Design II course involves the analysis and design of steel and wood structures. In all these courses, the analysis and design of structural components are primarily based on hand solutions. Hand based solutions are extremely important as they provide good theoretical foundation and analytical skills. However, they become tedious and cumbersome when structures are made of many members or when calculations have to be repeated during the design iterations. Under such circumstances, the use of commercial structural analysis software greatly simplifies calculations, gives freedom to change loads and support conditions, and visualize deformations, internal forces, and stresses. This paper discusses the computer structural analysis learning modules that were developed and utilized in the three courses in our architectural technology program.

Introduction

The finite element analysis (FEA) is a powerful simulation method used for many applications in the industry. Commercial FEA software can perform structural analysis of both discrete and continuous structures. Several educators [1] – [4] have utilized the FEA software in the curricula of architectural engineering and architecture programs for structural and nonstructural applications. However, we have not come across any publication that discusses the application of structural analysis software in the architectural technology program curriculum. The current paper describes in detail the use of the commercial structural analysis software, SkyCiv [5], in three courses in the architecture technology program.

Students in the technology programs prefer hands-on activities and computer-based learning methods as opposed to calculus-based methods as in the engineering programs. In today's times, it is of paramount importance to incorporate computer-learning techniques across courses in the architecture technology curriculum for modeling, analysis, and design. Additionally, because our programs have many nontraditional students who work during the day and attend classes in the evenings, the computer-learning techniques also enhance student engagement. The analysis using SkyCiv complements the hand solutions discussed in the course lectures

Description of SkyCiv Software

SkyCiv is a cloud-based structural analysis and design software that can be used for FEA. It has both free trial and priced versions, is fully online, does not need any installation, and can be accessed from anywhere. The free trial version has limitation on the maximum number of elements and simulation menu options. Currently, students can sign up for the individual Student Account through two options- (a) Paying USD 14.00 per month with no contract or (b) Purchasing a year's access for USD 99.00. Alternatively, they can use the university's Educational License Account at USD 25.00 per year. SkyCiv software provides resources such as tutorials and featured projects. One of the main benefits of the software is that it also gives detailed reporting of the calculations, similar to hand calculations, thus enabling students to verify their hand solutions. Additionally, many reinforced concrete, steel, and wood structural design codes are fully integrated in the software and hence can be used in structural design courses as well.

Courses Selected for SkyCiv Application

ACT 201 Structural Analysis: This course concerns with the material and structural geometric properties of basic framing materials (wood, steel, concrete and masonry), force and loading systems, two-dimensional equilibrium conditions of structures, ideal supports and reactions, tensile and compressive axial stresses and strain, shear and bending stresses, and buckling. Students will use the studied concepts to design and select appropriate structural elements. Some of the student learning outcomes include (a) determination of member forces in plane trusses by joint equilibrium method or section method, (b) construction of shear and moment diagrams in beams through shear and moment equations.

The current approach of solving problems using hand solution methods lack the replication of environmental conditions that change when structures and components undergo deformation under various loading systems. In addition, solving complex problems involve significant amount of effort and time. On the other hand, applying SkyCiv will provide accurate and fast results, coupled with visualization features for postprocessing the results. Another advantage is the display of the effects of deformation on the structure, which can be used to optimize material selection and sizing of structural members.

AT 321 Structural Design I: This course involves the fundamental concepts/principles of mechanics and strength of materials in dealing with the state of rest of bodies under the action of forces. It applies the equilibrium conditions to the analyze concrete structures formed by connected members, including reinforced beams, columns, floors, walls, and footings. The design process is studied in depth utilizing AISC and ACI Standards/Codes. One of the student learning outcomes is the design of reinforced concrete beams and slabs by the working stress method.

AT 322 Structural Design II: This course continues the study of material strength, basic design, and calculations of lumber and steel structural systems. It includes the strength, stiffness, and stability of various materials, stresses caused by bending moments, shear forces, vertical and horizontal loadings, and dimensioning load supporting structural members under those

influences. The design process utilizes AISC and AF and PA standards. One of the student learning outcomes is the design of a simply supported steel beams for flexural and shear stresses, vertical deflections, and buckling.

Structural Analysis

Students performed structural analysis through three phases:

Phase 1: inspection method for identifying key components of structural behavior such as internal forces, equilibrium, and deflected shape. It helps students develop a physical feel for the structural behavior and anticipate the type of results to expect.

Phase 2: detailed hand calculations using the analytical methods discussed in the lectures, and verification of results

Phase 3: computer analysis using the learning modules and SkyCiv software

Three structures- plane truss, simple beam, and rigid frame were analyzed. The learning modules are described in the paper. For the sake of brevity, only learning module 1 has GUI instructions. The other two modules have the step_by_step command mode of instructions.

Learning Module 1: Plane truss analysis

The analysis involves the computation of support reactions and member forces in an overhanging truss [6] shown in figure 1, roller supported at A and pin supported at B.

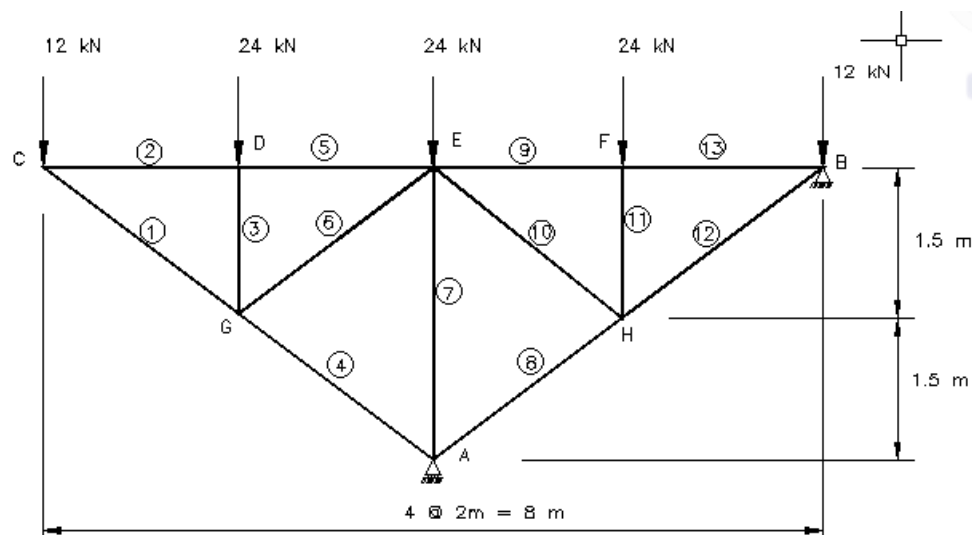


Figure 1: Overhanging truss

Phase 1 requires the identification of any zero-force members and the nature of member forces (tension or compression). By inspection, there are no zero-force members in this truss. Based on the deformation of the truss the top horizontal members (chord) are in tension, the vertical and

outer inclined members are in compression, and the inner inclined members are in tension because the vertical members are in compression. The horizontal reaction component at the pin support B is zero as there are no applied loads in the horizontal direction. The sum of the vertical components of the support reactions must be equal to the total downward load of 96 kN. Detailed hand calculations are carried out in phase 2 by the method of joints and the method of sections as discussed in the lectures, and the equilibrium check is performed. The computer analysis is performed in phase 3 and here are the steps to solve the problem using Skyciv.

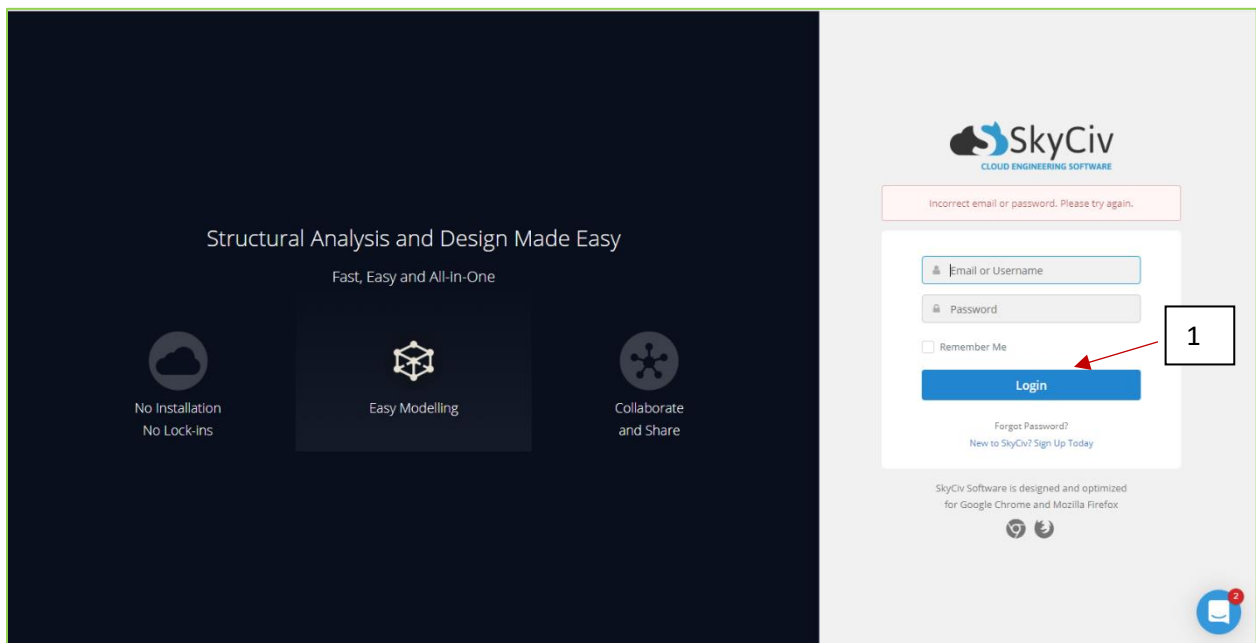
The analysis involves the computation of support reactions and member forces in an overhanging truss shown in figure 1 which is roller supported at A and pin supported at B. Phase 1 requires the identification of any zero-force members and the nature of member forces (tension or compression). By inspection, there are no zero-force members in this truss. The top horizontal members (chord) are in tension, the vertical and outer inclined members are in compression, and the inner inclined members are in tension because the vertical members are in compression. The horizontal reaction component at the pin support B is zero as there are no applied loads in the horizontal direction. The sum of the vertical components of the support reactions must be equal to the total downward load of 96 kN. Detailed hand calculations are carried out in phase 2 by the method of joints and the method of sections as discussed in the lectures, and the equilibrium check is performed. The computer analysis is performed in phase 3 and here are the steps to solve the problem using Skyciv.

- **SkyCiv implementation**

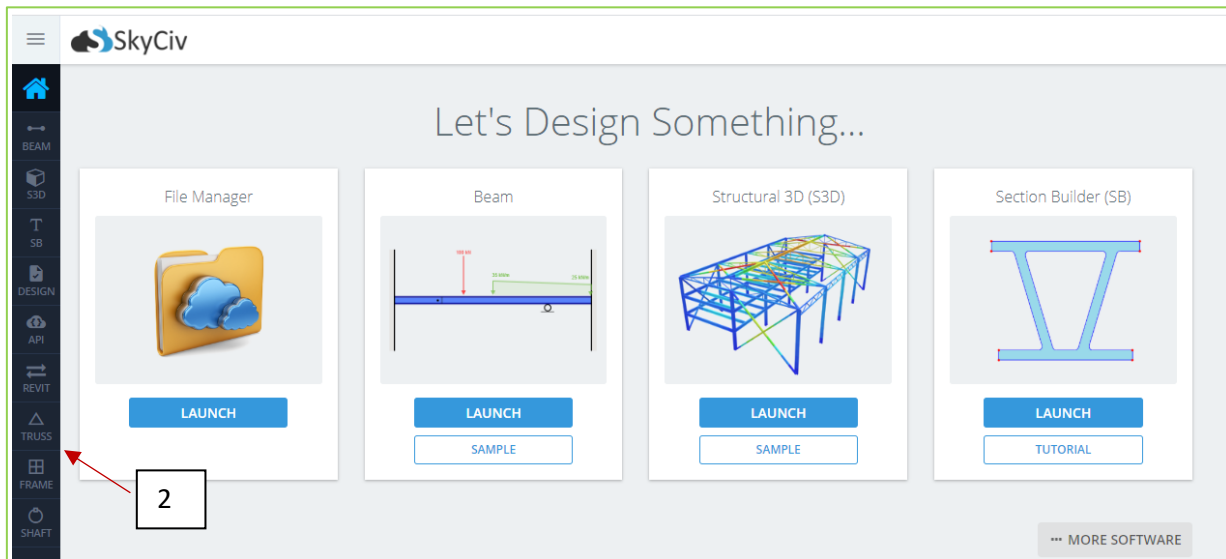
Create the truss

1. Launch the SkyCiv website and login to the dashboard.

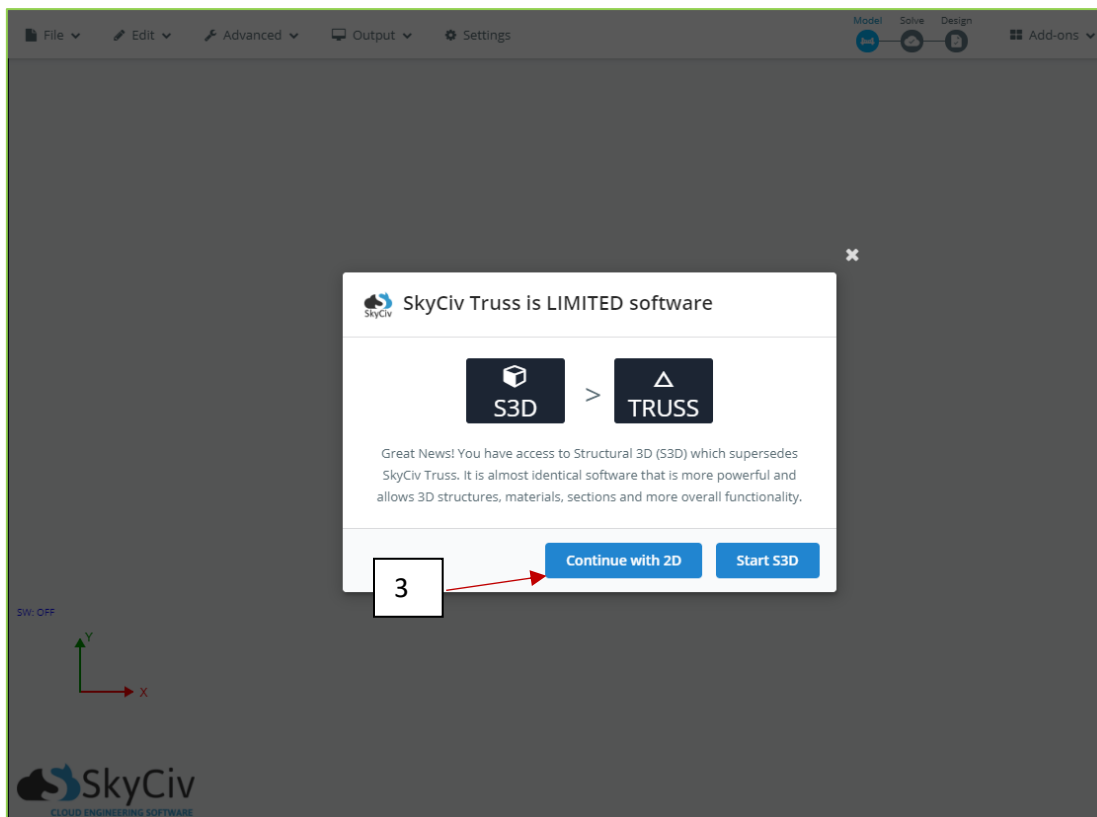
<https://skyciv.com/>



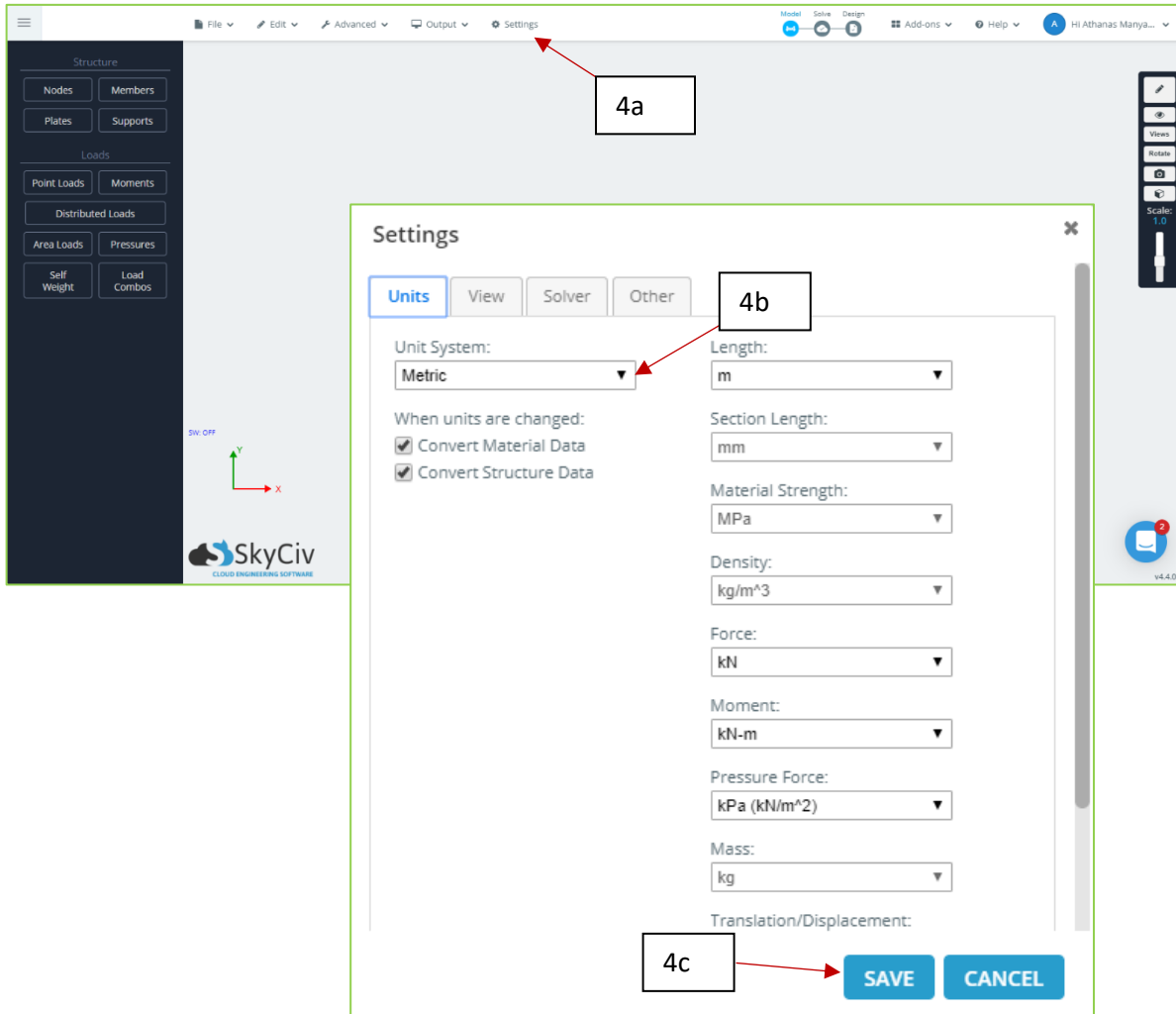
2. Scrow down on the left side of the screen to select “Truss”



3. Select “Continue with 2D”.



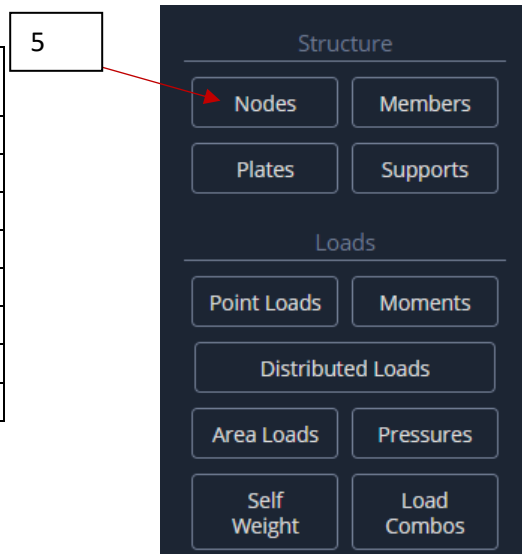
4. Specify the units to be used by clicking on “Settings” and save the model.



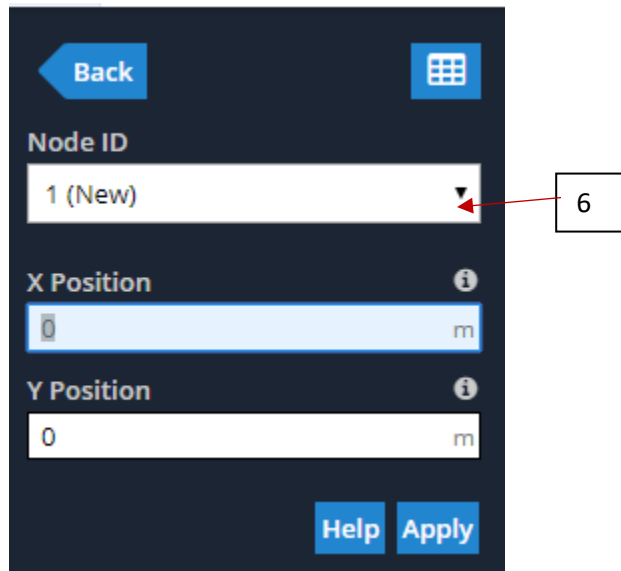
5. Use the table 1 to create nodes, members, and supports.

Nodes	X-Coordinates	Y-Coordinate
1	0	0
2	2	-1.5
3	2	0
4	4	-3
5	4	0
6	6	-1.5
7	6	0
8	8	0

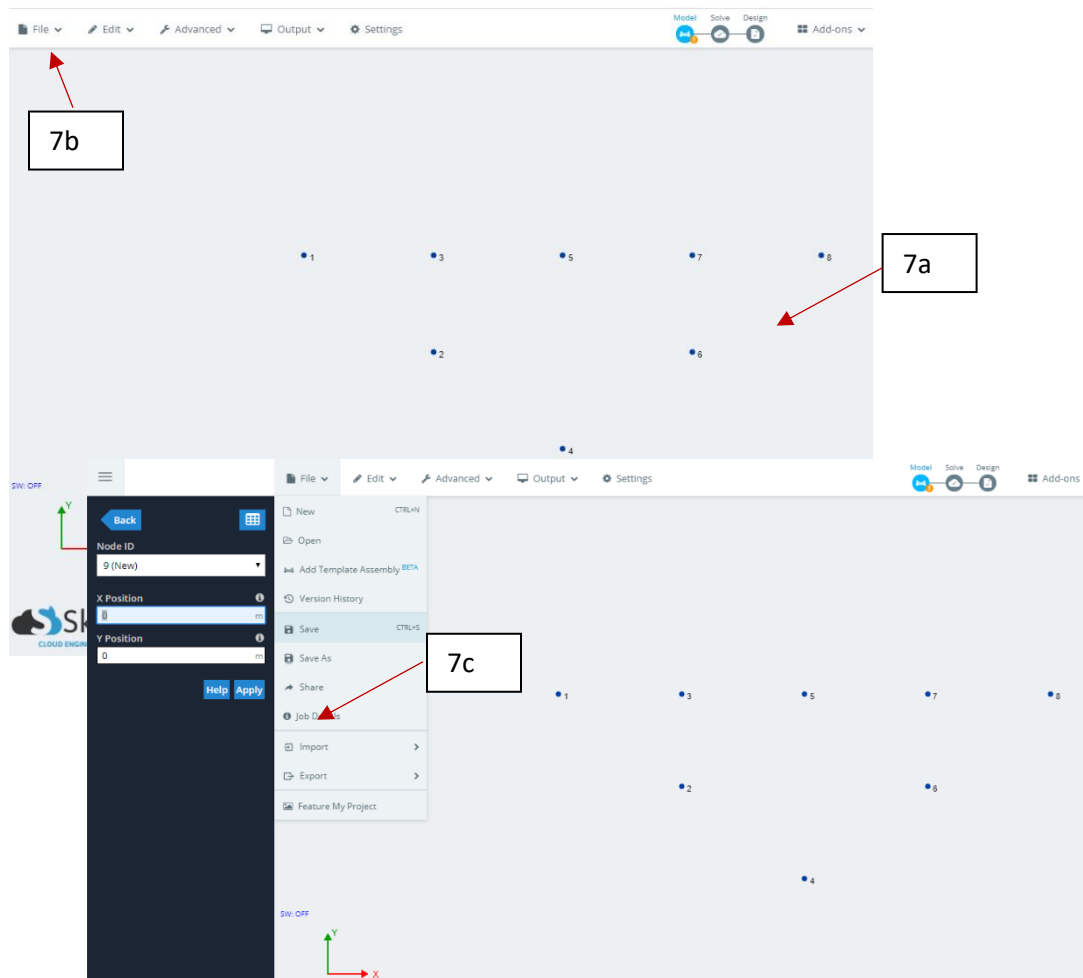
Table 1: Nodal coordinates



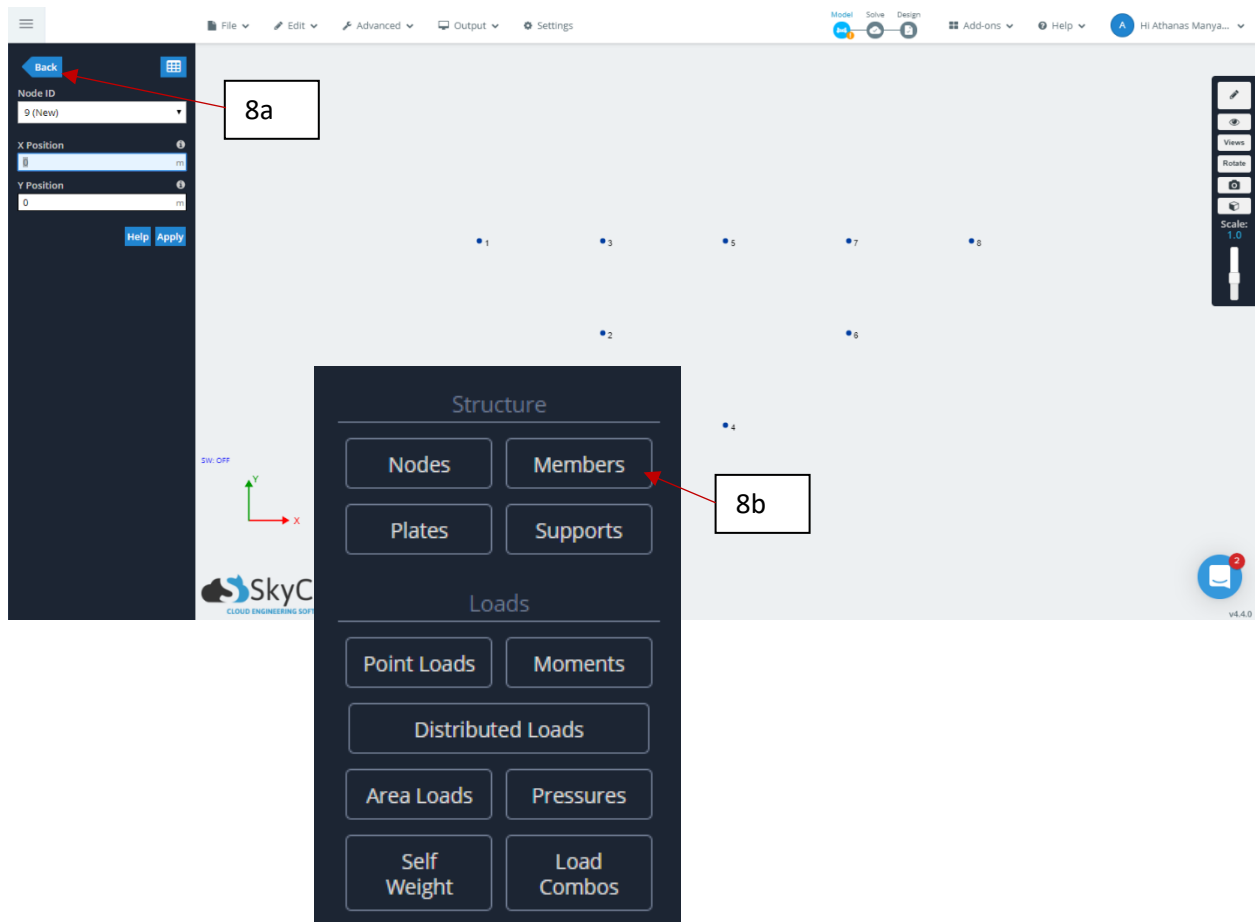
6. Add node 1 to start building model by specifying the x and y coordinates (see table above for coordinates). Click on “Apply”.



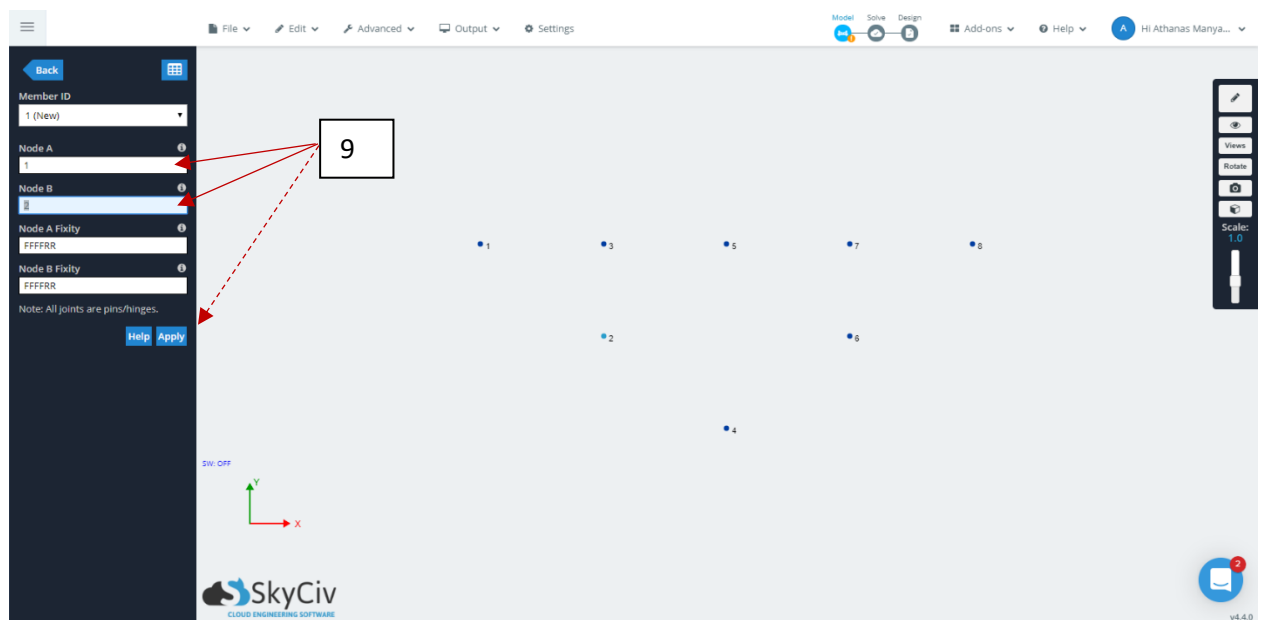
7. Repeat the process to add all 8 nodes; save the work by selecting “File” and “Save”.



8. Click on “back” and select “Members” to continue creating members between nodes.



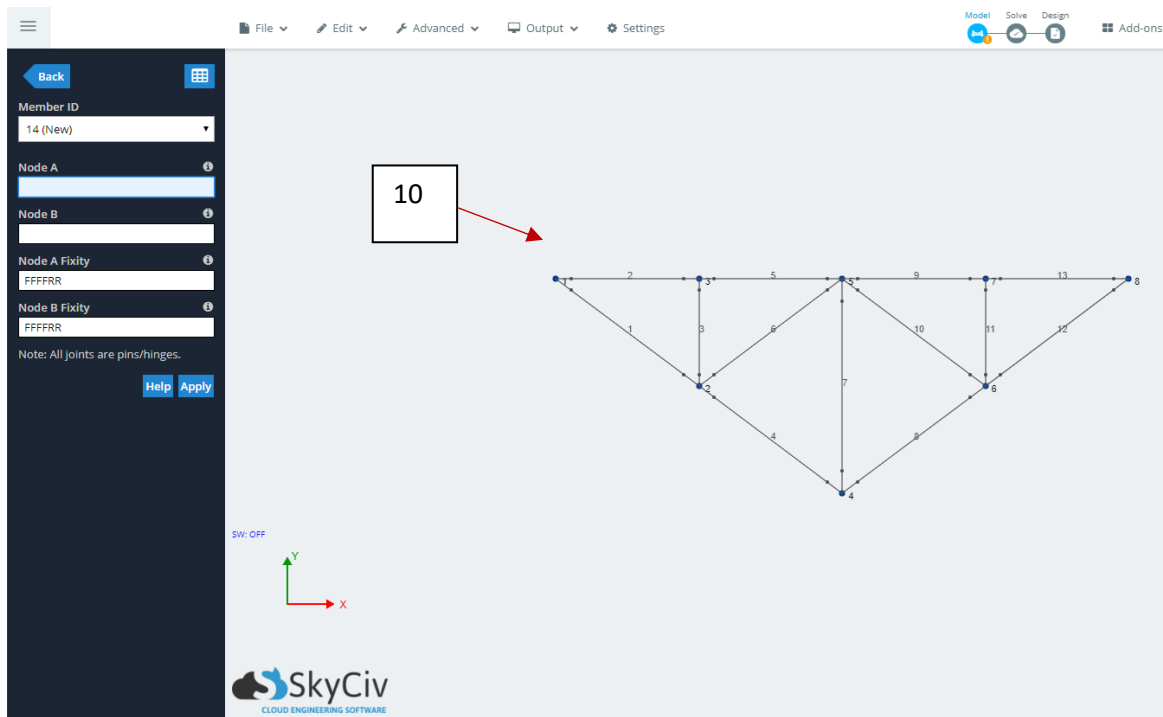
9. Create member #1 by specifying nodes 1 and 2. Click on “Apply”.



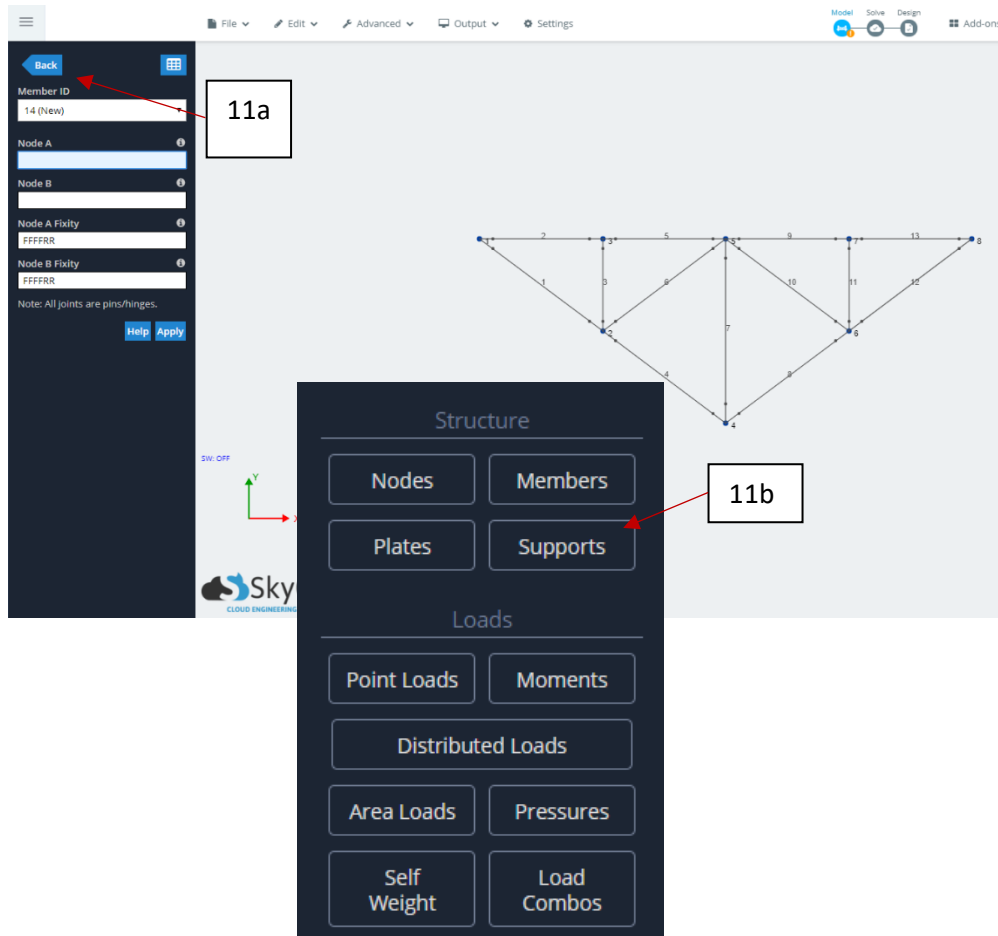
10. Repeat step # 9 to create all members using table 2

Nodes	Members
1 and 2	1
1 and 3	2
2 and 3	3
2 and 4	4
3 and 5	5
2 and 5	6
4 and 5	7
4 and 6	8
5 and 7	9
5 and 6	10
6 and 7	11
6 and 8	12
7 and 8	13

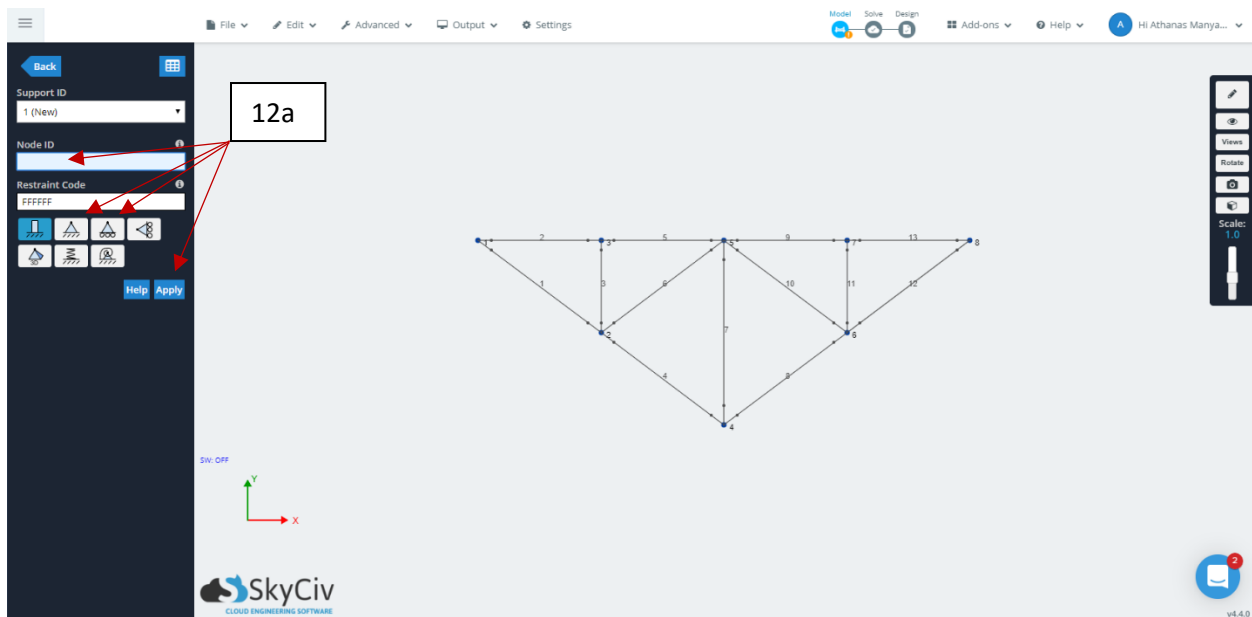
Table 2: Member connectivity nodes

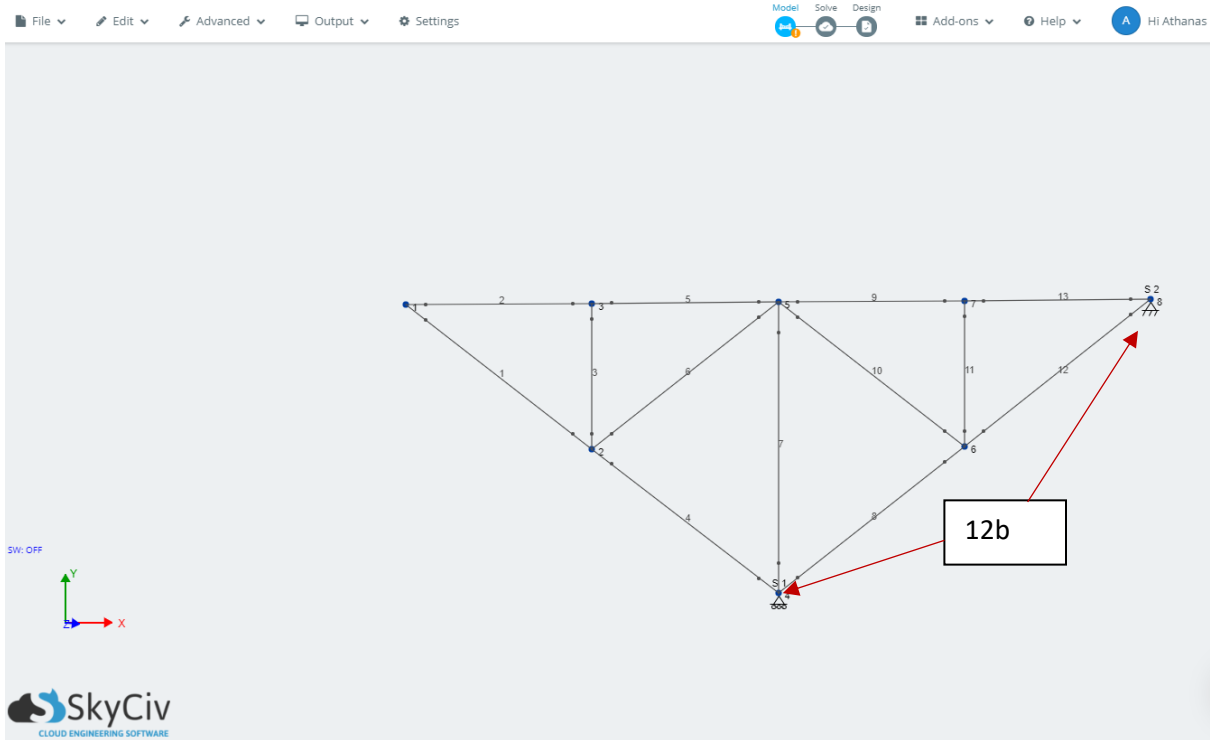


11. Click on “back” and select “Supports” in the pop up window.

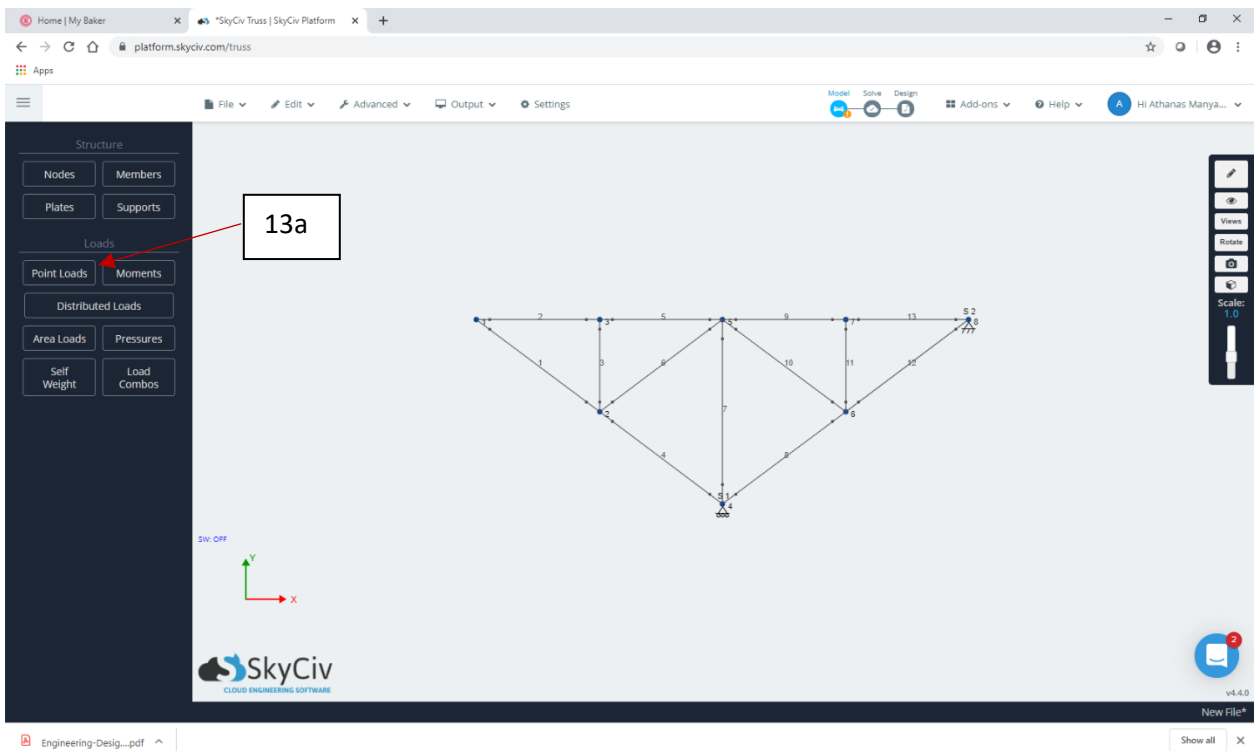


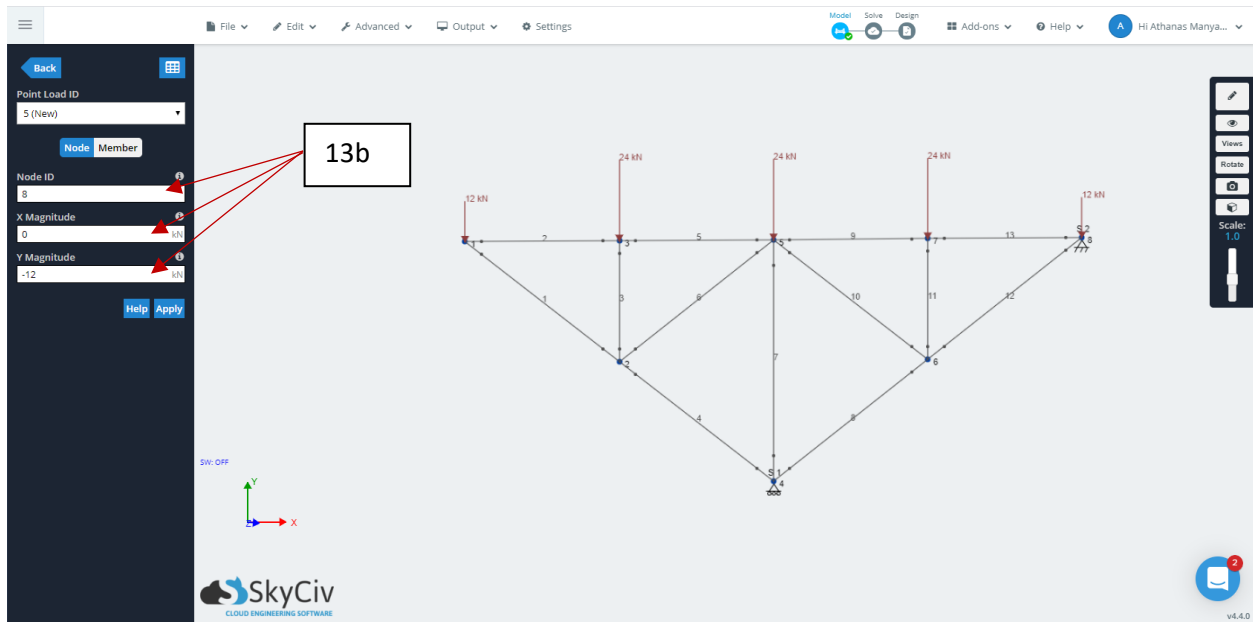
12. Add roller support at node 4 and pin support at node 8.



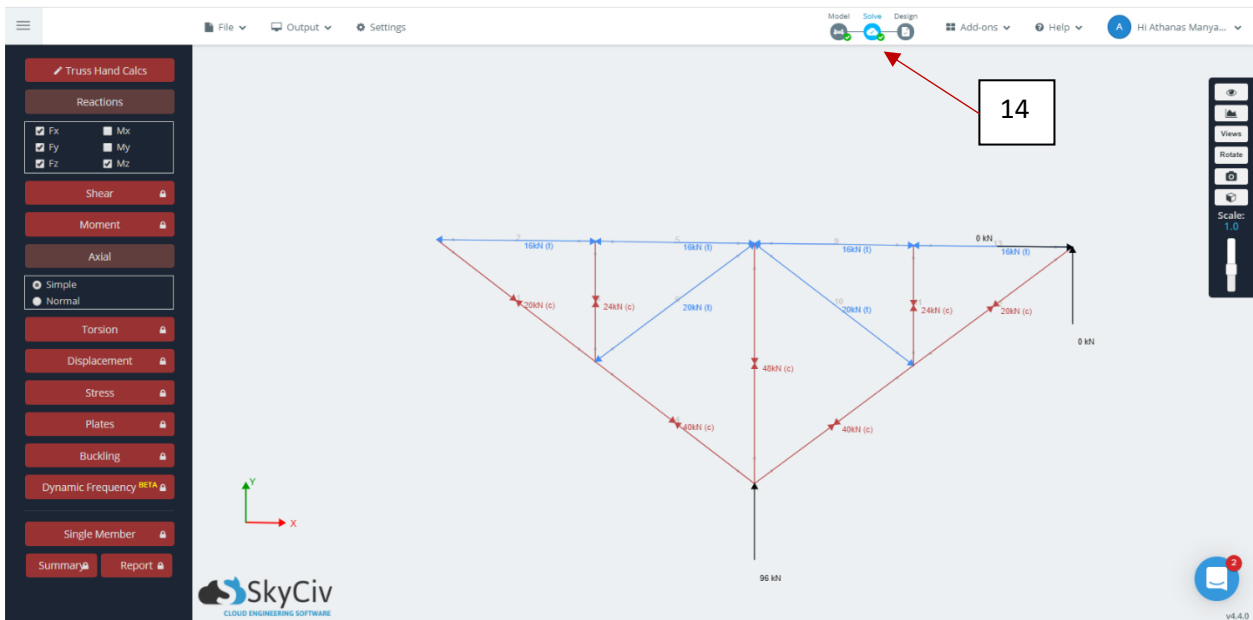


13. Click on “back” and add points loads on nodes 1 (12 kN), 3 (24 kN), 5 (24 kN), 7 (24 kN) and 8 (12 kN). Specify the sense of the point loads.





14. Select “Solve” to perform the linear static analysis to generate the axial forces in the members of the truss and the reactions at the supports as shown below.



Detailed calculations by SkyCiv

SkyCiv generates report of various results such as support reactions, member forces, displacements, stresses, etc. The support reactions and member forces by the method of joints are reported in this paper. For the sake of brevity, the output of two joints is given.

(a) Support reactions

Reactions

All forces in kN. All lengths in m.

$$\rightarrow \sum F_x = 0$$

$$(R_{2,x}) = 0$$

$$\therefore R_{2,x} = 0 \text{ kN}$$

$$+\uparrow \sum F_y = 0$$

$$(R_{1,y}) + (R_{2,y}) + (-12) + (-24) + (-24) + (-24) + (-12) = 0$$

$+\circlearrowleft \sum M = 0$ about Support 2 located at Node 8

$$(4)(R_{2,y}) + (-1)(3)(R_{2,x}) + (4)(12) + (2)(24) + (-1)(2)(24) + (-1)(4)(12) = 0$$

$$(4)(R_{2,y}) + (-3)(R_{2,x}) + (48) + (48) + (-48) + (-48) = 0$$

Substituting from previous values,

$$(4)(R_{2,y}) + (0)(-3) + (48) + (48) + (-48) + (-48) = 0$$

$$\therefore R_{2,y} = 0 \text{ kN}$$

New values can be substituted into force equation F_y

$$(R_{1,y}) + (0) + (-12) + (-24) + (-24) + (-24) + (-12) = 0$$

$$\therefore R_{1,y} = 96 \text{ kN}$$

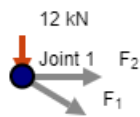
(b) Member forces by the method of joints.

Joint 1

All forces in kN. All lengths in m.

Known Reactions: $R_{1,y} = 96$, $R_{2,x} = 0$, $R_{2,y} = 0$,

Known Axial Forces:



- 0 + ← ↓ ↑ →

$$\rightarrow \sum F_x = 0$$

$$(F_1)(\cos 36.87) + (F_2) = 0$$

$$(0.8)(F_1) + (F_2) = 0$$

$$+\uparrow \sum F_y = 0$$

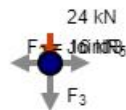
$$(-12) + (-1)(F_1)(\sin 36.87) = 0$$

Joint 3

All forces in kN. All lengths in m.

Known Reactions: $R_{1,y} = 96$, $R_{2,x} = 0$, $R_{2,y} = 0$,

Known Axial Forces: $F_1 = -20$, $F_2 = 16$,



- 0 + ← ↓ ↑ →

$$\rightarrow \sum F_x = 0$$

$$(-1)(F_2) + (F_5) = 0$$

Substituting the previously found member (axial) forces,

$$(-1)(16) + (F_5) = 0$$

$$\therefore F_5 = 16 \text{ kN}$$

Summary of Axial Forces

All axial forces have been solved*. They are:

$$F_1 = -20 \text{ kN}$$

$$F_2 = 16 \text{ kN}$$

$$F_3 = -24 \text{ kN}$$

$$F_4 = -40 \text{ kN}$$

$$F_5 = 16 \text{ kN}$$

$$F_6 = 20 \text{ kN}$$

$$F_7 = -48 \text{ kN}$$

$$F_8 = -40 \text{ kN}$$

$$F_9 = 16 \text{ kN}$$

$$F_{10} = 20 \text{ kN}$$

$$F_{11} = -24 \text{ kN}$$

$$F_{12} = -20 \text{ kN}$$

$$F_{13} = 16 \text{ kN}$$

*Note that:

- positive numbers represent tension
- negative numbers represent compression
- zero represents a zero force member

All reaction forces have been solved. They are:

$$R_{1,y} = 96 \text{ kN}$$

$$R_{2,x} = 0 \text{ kN}$$

$$R_{2,y} = 0 \text{ kN}$$

Learning Module 2: Beam analysis

A simple beam [7], carrying distributed and concentrated/point loads, shown in figure 2, is analyzed. The analysis involves the computation of support reactions, shear forces, bending moments, and plotting shear and moment diagrams. Phase 1 requires the identification of the shape of the shear and moment diagrams. By inspection, the shear forces drops suddenly at locations of concentrated load, as one moves from left end to the right end of the beam. The sudden drop is equal to the magnitude of the concentrated load. In addition, the shear force is constant in the domain AC due to no distributed load acting, and varies linearly in the remaining domains due to the uniform distributed load. Consequently, the bending moment varies linearly

in the domain AC and parabolically in domain CB. The horizontal reaction component at the pin support A is zero as there are no applied loads in the horizontal reaction. The sum of the vertical components of the support reactions must be equal to the total downward load of 48.2 kip. Detailed hand calculations are carried out in phase 2 by the method of sections as discussed in the lectures, and equilibrium check is performed. The computer analysis is performed in phase 3, and here are the steps to solve the problem using SkyCiv.

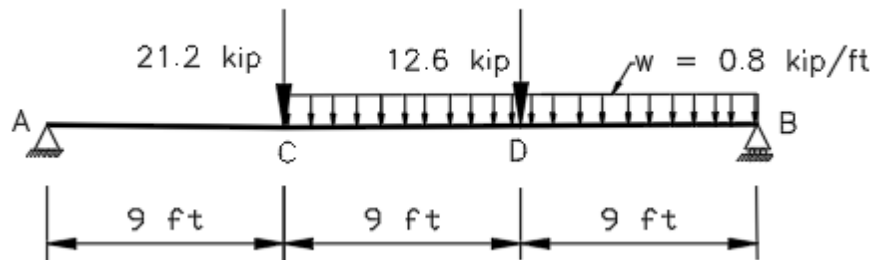


Figure 2: Simple beam

- **SkyCiv implementation**

Create the beam

1. Launch the SkyCiv website and login to the dashboard.

<https://skyciv.com/>

Modeling the Structure:

2. Select > BEAM > near the top left side of the screen.
3. Specify the force units < kip > and length units < ft >, then select “Add Beam”.
4. Fill in all fields without units > Beam length = 27 > ADD IT. The beam appears on the screen.
5. Select > Add Support > PIN SUPPORT > Number of Supports = 1 > Support Position = 0 > ADD IT. Select > Add Support > ROLLER SUPPORT > Number of Supports = 1 > Support Position = 27 > ADD IT.

Applying Loads:

6. Select > Add Point Load > Arrow pointing downward > Number of Loads = 1 > Load Position = 9 > Load Magnitude = 21.2 > ADD IT.
7. Select > Add Point Load > Arrow pointing downward > Number of Loads = 1 > Load position = 18 > Load Magnitude = 12.6 > ADD IT.

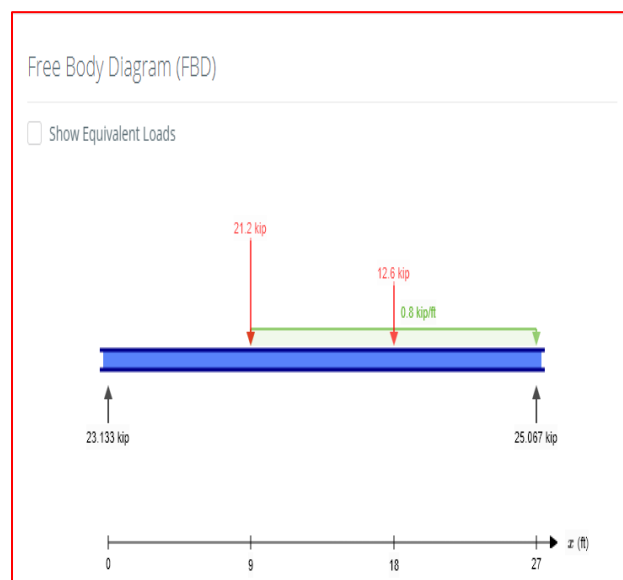
8. Select > Add Distributed Load > Arrow pointing downward > Start Position = 9 > End Position = 27 > Start Magnitude = 0.8 > End Magnitude = 0.8 > ADD IT.



9. Click on > File > Save > to save the model.

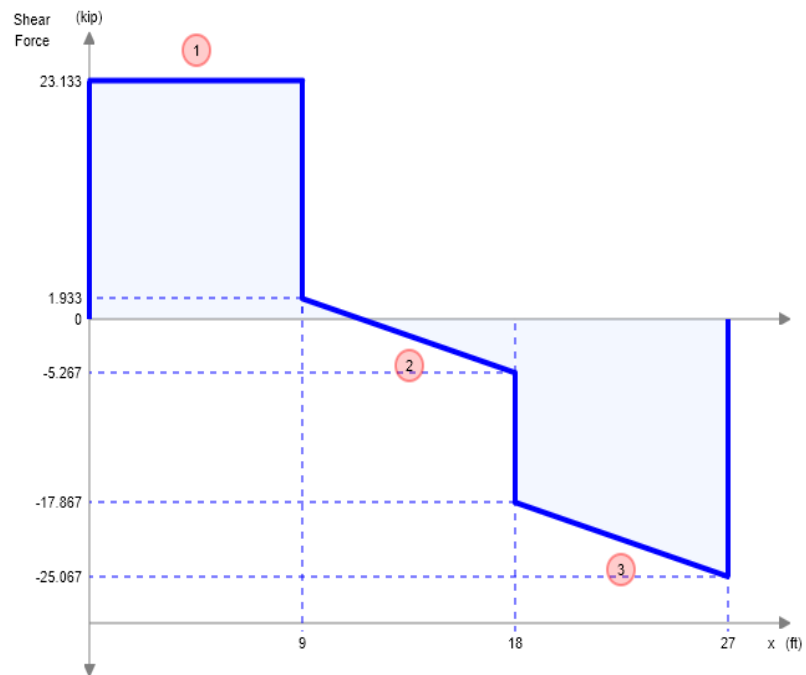
Solving the Structure / Post Processing:

10. Click on > Solve > to generate the free body diagram (FBD), shear force diagram (SFD) and bending moment diagram (BMD)



Shear Force Diagram (SFD)

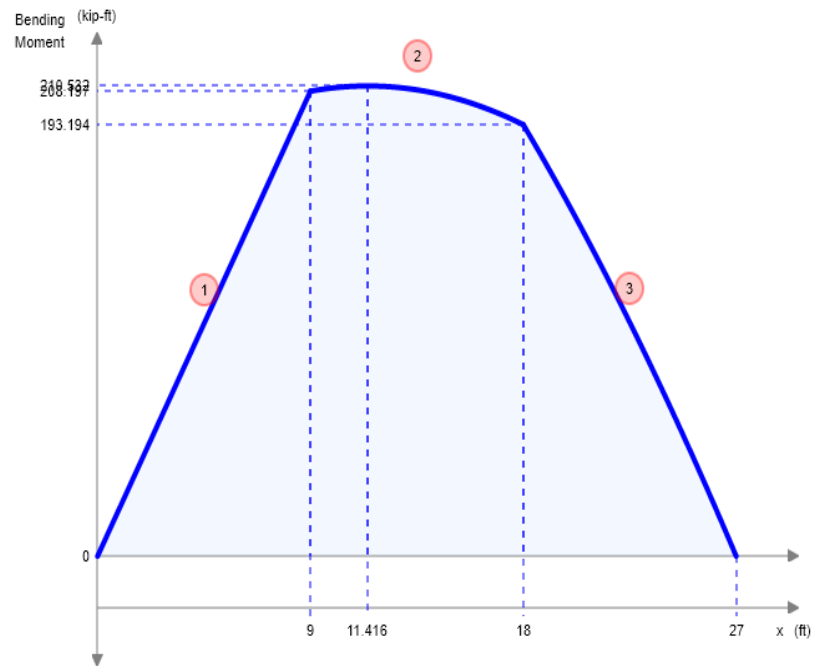
The Shear Force (V) at ft along the beam is: (No Position Entered)



Bending Moment Diagram (BMD)

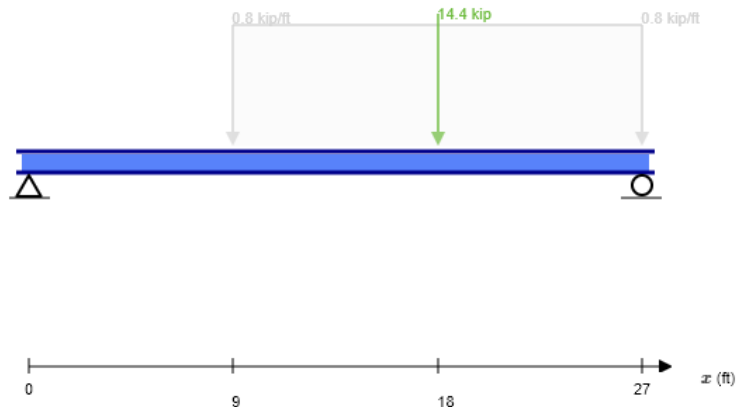
Reverse BMD Sign Convention

The Bending Moment (M) at ft along the beam is: (No Position Entered)



Detailed calculations by SkyCiv

Equivalent load due to the distributed load



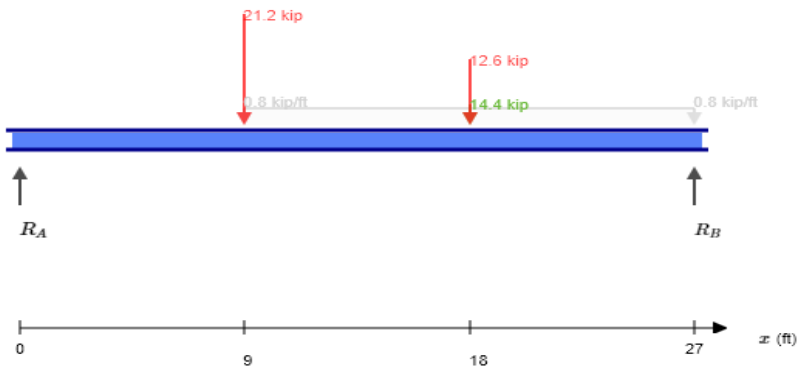
Start by finding the equivalent loads caused by the distributed loads (DLs).

Remember that the equivalent load of a DL has a magnitude equal to its area, acting through the area's centroid.

Equivalent load(s) of the DL from $x = 9\text{ft}$ to $x = 27\text{ft}$:

Rectangular Load Magnitude: $(27\text{ft} - 9\text{ft}) \times (-0.8\text{kip/ft}) = -14.4\text{kip}$

Rectangular Load Position: $9\text{ft} + \frac{27\text{ft} - 9\text{ft}}{2} = +18\text{ft}$



Let the left support have a vertical reaction R_A and the right support have a vertical reaction R_B .

Sum of forces along the y-axis is equal to zero for static equilibrium:

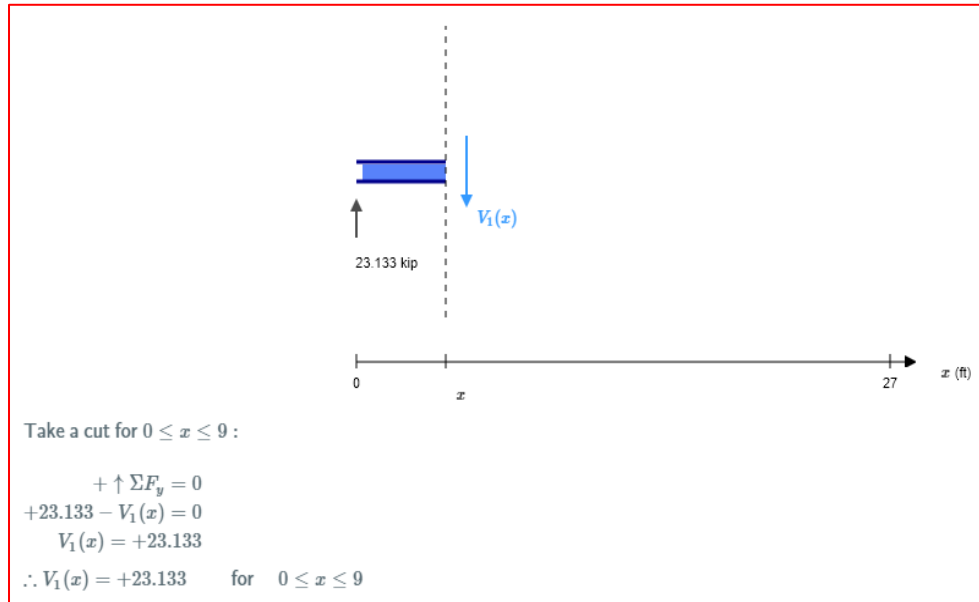
$$\begin{aligned} + \uparrow \Sigma F_y &= 0 \\ R_A + R_B - 21.2 - 12.6 - 14.4 &= 0 \\ R_A + R_B - 21.2 - 12.6 - 14.4 &= 0 \\ R_A + R_B &= +48.2\text{kip} \quad \text{--- (1)} \end{aligned}$$

Sum of moments about the left support is equal to zero for static equilibrium:

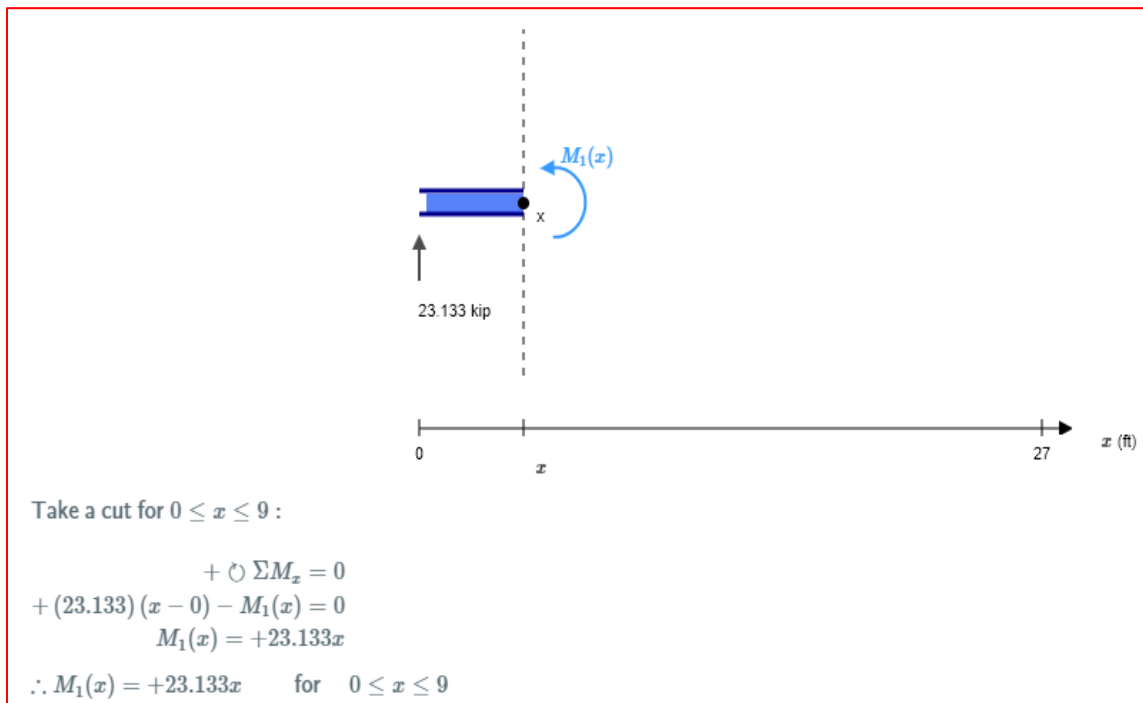
$$\begin{aligned} + \circlearrowleft \Sigma M &= 0 \\ R_B (27 - 0) + (-21.2) (9) + (-12.6) (18) + (-14.4) (+18) &= 0 \\ R_B (27 - 0) + (-21.2) (+9) + (-12.6) (+18) + (-14.4) (+18) &= 0 \\ R_B &= +25.067\text{kip} \quad \text{--- (2)} \end{aligned}$$

SkyCiv reports shear force calculations using shear equations, and bending moment calculations using bending moment equations for section cuts $0 < x \leq 9$ ft; $9 \leq x \leq 18$ ft; and $18 \leq x \leq 27$ ft. To keep the length of the paper short, only the calculations for section cut $0 < x \leq 9$ ft is reported here.

Shear: $0 < x \leq 9$ ft



Moment: $0 \leq x \leq 9$ ft.



Learning Module 3: Rigid frame analysis

The plan and section views of a reinforced concrete multi-story building [8] discussed in the AT 321 structural design course is shown in figures 3-5. The building has double tee-beams (DT), girders (G), and columns (C). In the hand solution method, to simplify the analysis and design, the beams, girders, and columns were analyzed by assuming the ends as simply supported thus resulting in approximate solutions. However, accurate results can be obtained using the SkyCiv software.

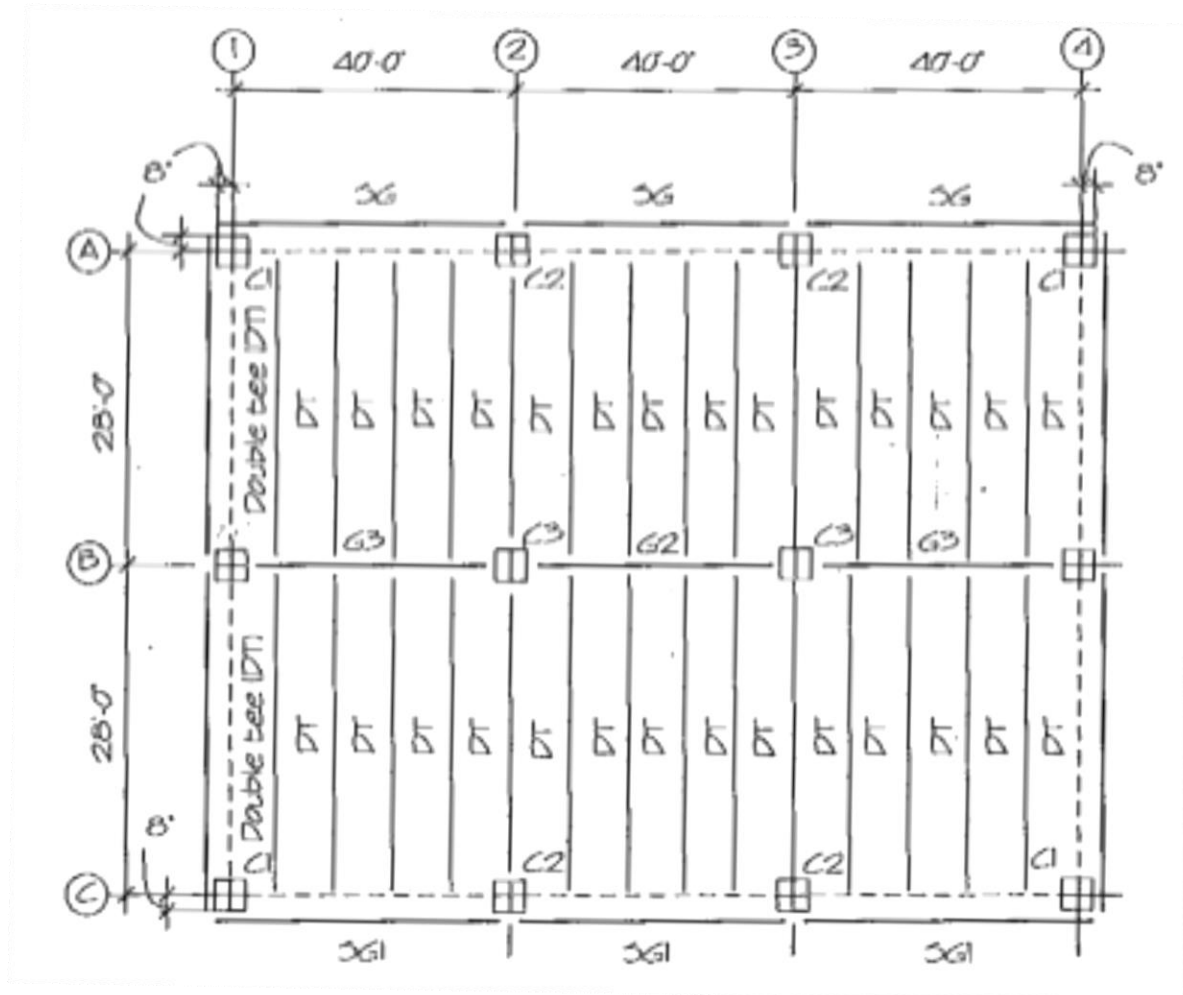


Figure 3: Roof framing plan

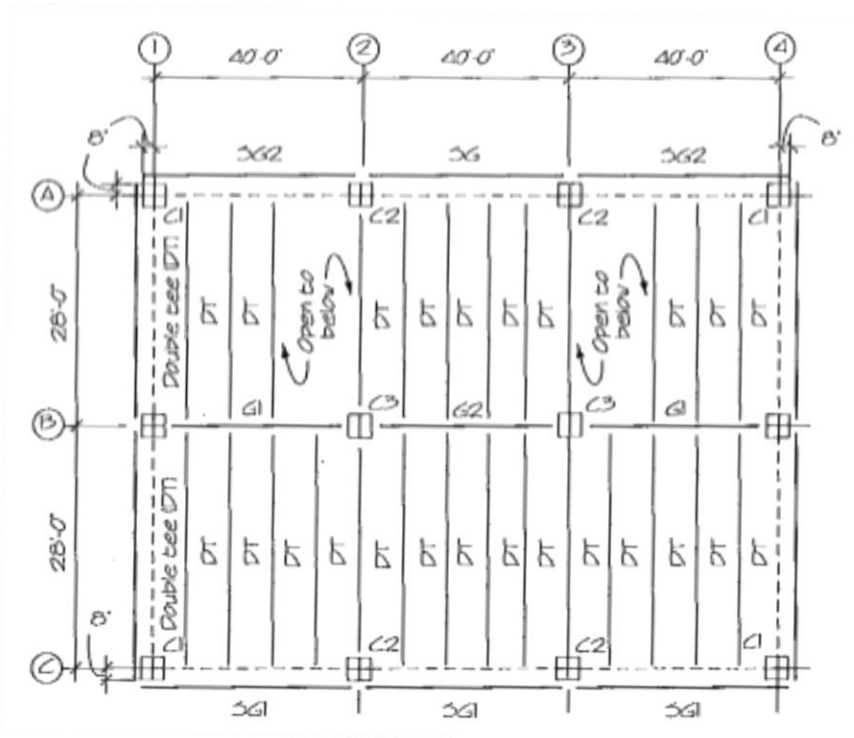


Figure 4: Floor framing plan

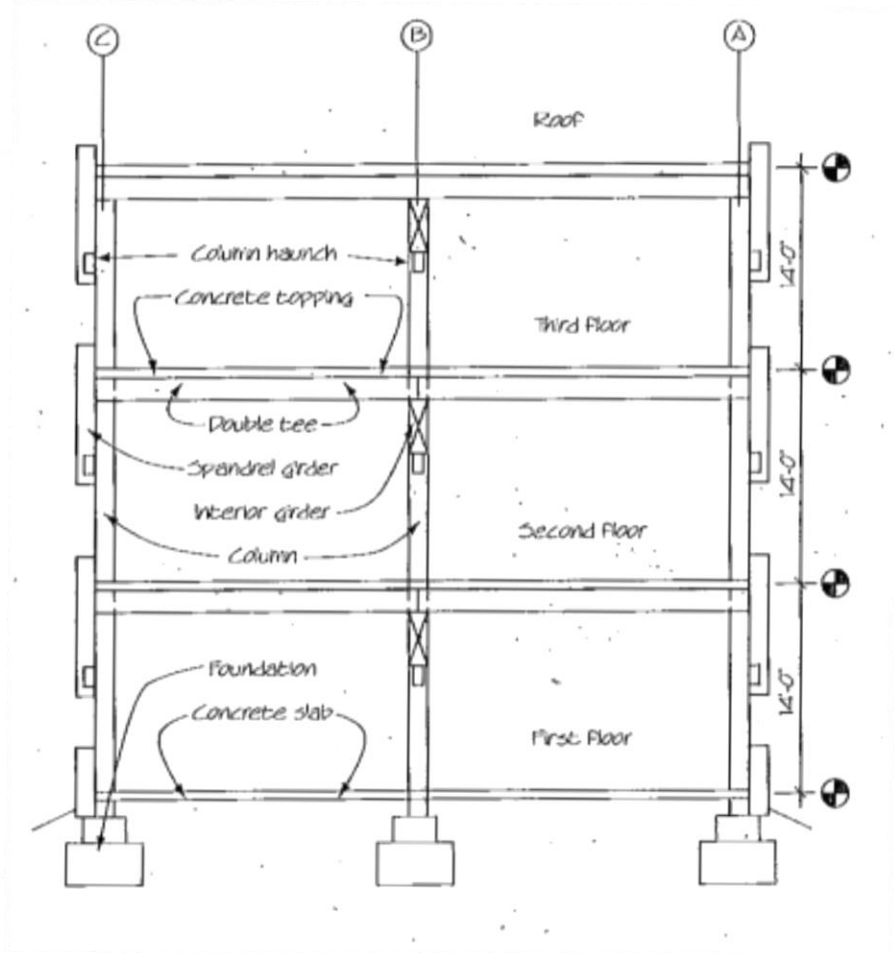


Figure 5: Transverse building section

A simple two-member rigid frame [9] with a horizontal beam and an inclined column shown in figure 6 was considered in the development of the learning module. The frame is fixed at the ends and has a distributed load. Following the learning module any multi-story building can be analyzed. One can also design the building using SkyCiv.

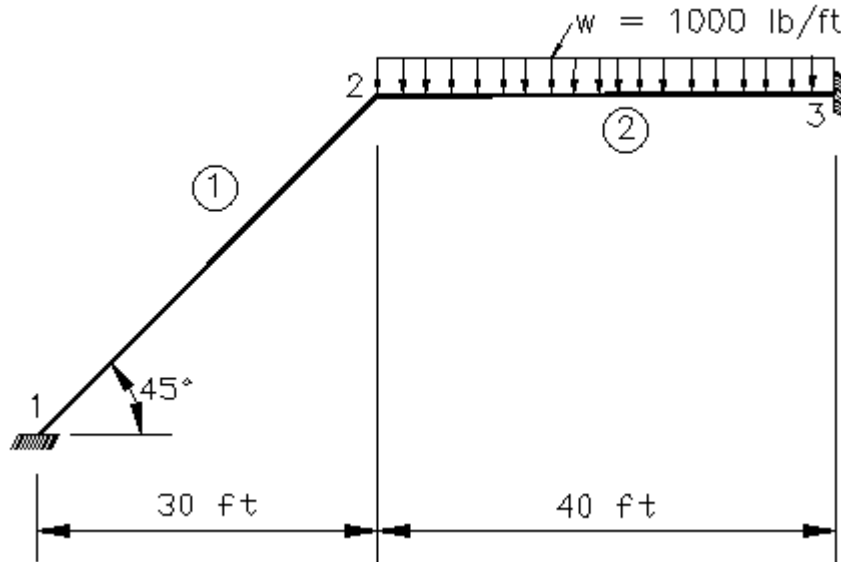
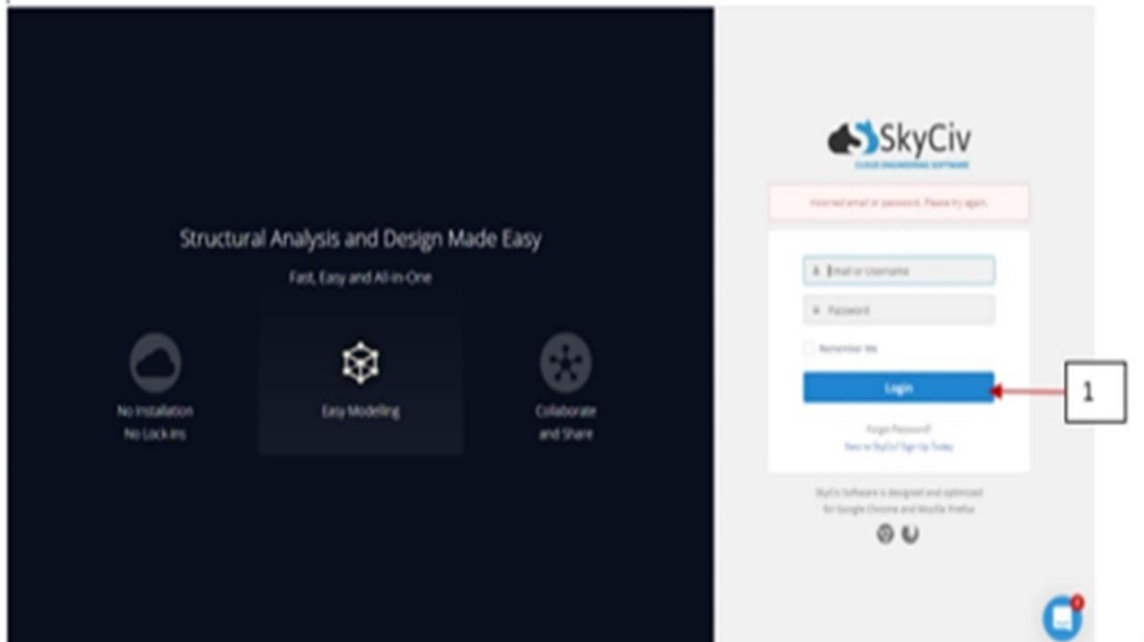


Figure 6: Rigid frame

The analysis involves the computation of support reactions, axial forces, shear forces, bending moments, and plotting shear and moment diagrams. Phase 1 requires the drawing of the deflected shape by hand and identifying the shape of the shear and moment diagrams. By inspection, the shear force is constant in member 1 due to no distributed load on it and varies linearly in member 2 due to the uniform distributed load. Consequently, the bending moment varies linearly and parabolically in the two members respectively. Each fixed end has three support reactions (two force and one moment reactions). For equilibrium, the sum of the vertical components of the support reactions must be equal to the total downward load of 40 kip. The algebraic sum of the moment reactions must be equal and opposite to the moment due the distributed load. The hand calculations assume the loaded member as simply supported at the ends. The computer analysis is performed in phase 3, and here are the steps to solve the problem using SkyCiv.

- **SkyCiv implementation**
Create the rigid frame

1. Launch the SkyCiv website and login to the dashboard.
<https://skyciv.com/>



Modeling the Structure:

2. Click on settings to select unit system.
 > Unit System = Imperial > Length = ft > Section Length = in > Material Strength = ksi > Force = kip > Moment = kip-ft >
3. Select > FRAME > near the bottom left side of the screen.
4. Create nodes > Nodes > Node ID = 1 > X position = 0 > Y position = 0 > Apply > Node ID = 2 > X position = 30 > Y Position = 30 > Apply . Node ID = 3 > X Position = 70 > Y Position = 30 > Apply > Back.

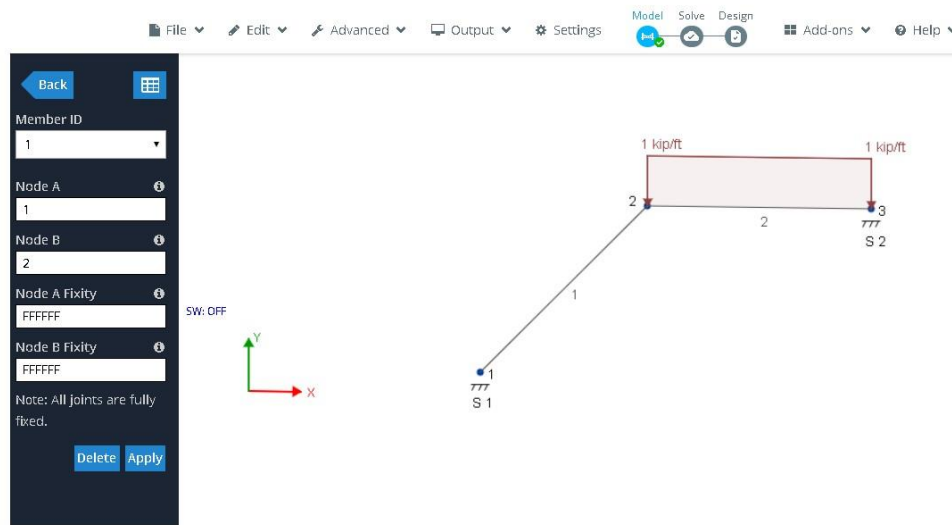
Nodes	X-Coordinates	Y-Coordinate
1	0	0
2	30	30
3	70	30

Table 3: Nodal coordinates for the frame

5. Select > Members > Member ID = 1 > Node A = 1 > Node B = 2 > Apply > Member ID = 2 > Node A = 2 > Node B = 3 > Apply > Back.
6. Select > Supports > Support ID = 1 > Node ID = 1 > Fixed Support > Apply > Support ID = 2 > Node ID = 3 > Fixed Support > Apply > Back.

Applying Loads:

7. Select > Distributed Loads > Distributed Load ID = 1 > Member ID = 2 > Start X-Mag = 0 > End X-Mag = 0 > Start Y-Mag = -1 > End Y-Mag = -1 > Start Position 0 > Finish Position = 100 > Apply.

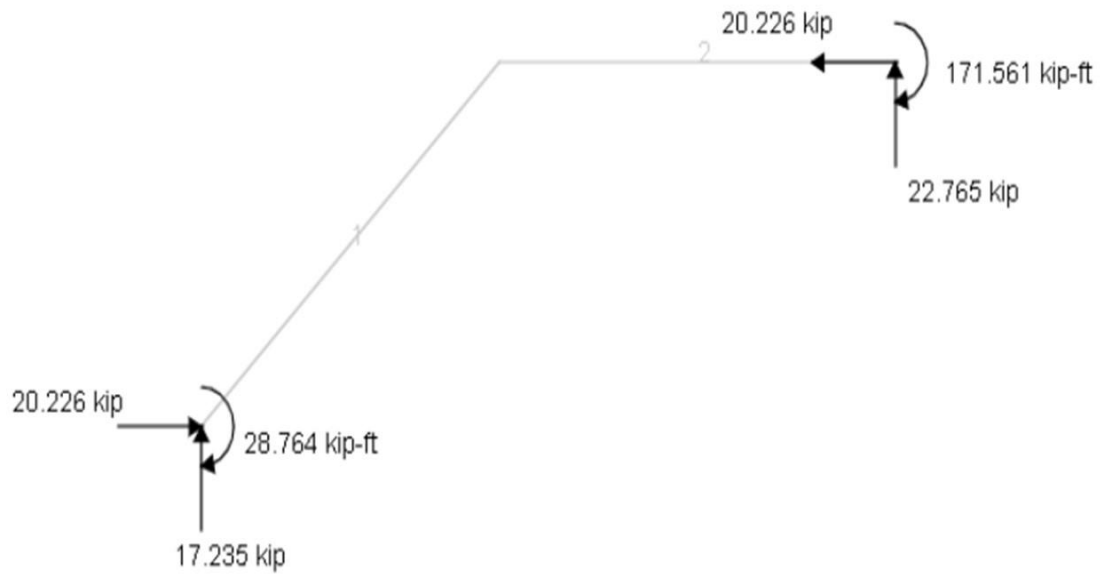


8. Click on > File > Save > to save the model.

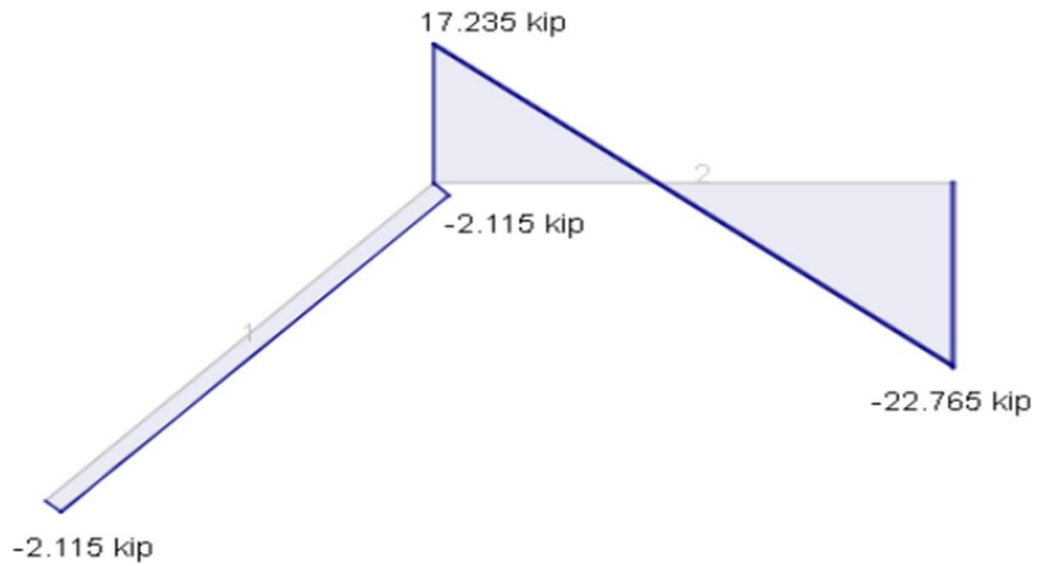
Solving the Structure / Post Processing:

9. Click on > Solve > to generate the free body diagram (FBD), shear force diagram (SFD) and bending moment diagram (BMD).

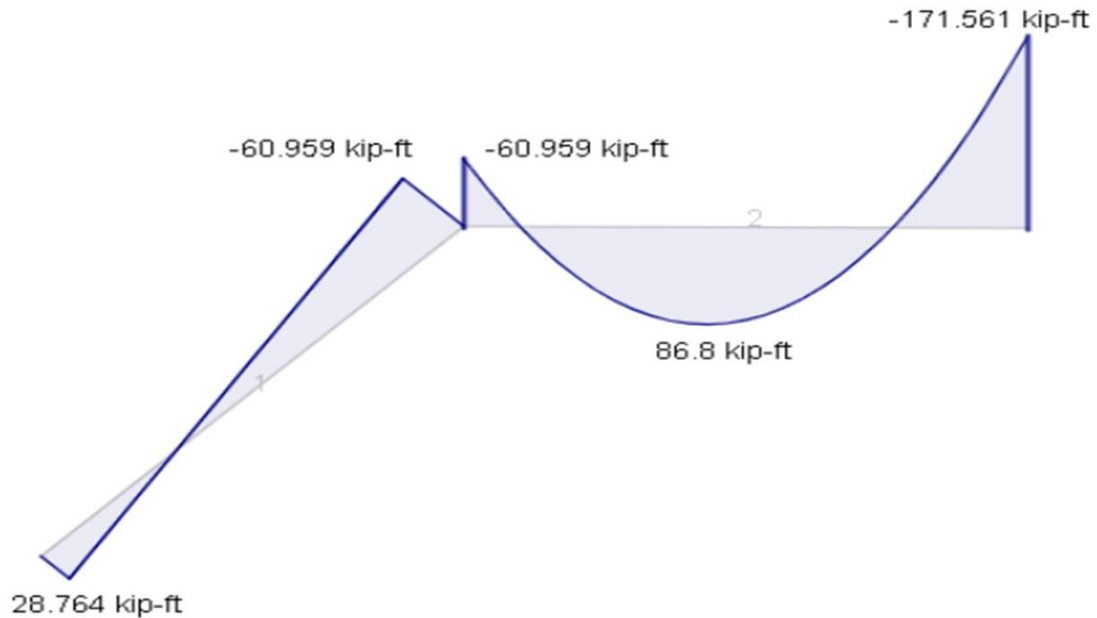
Support reactions:



Shear Force Diagrams:



Bending Moment Diagrams:



Conclusions:

Three GUI (Graphic User Interface) learning modules were developed and utilized in three courses of the architectural technology program curriculum at Baker College Flint for the computer structural analysis by SkyCiv commercial structural analysis software. These learning modules, with complete step-by-step instructions for solving trusses, beams, and rigid frames, are fully described in the paper. The GUI enables easy flow of steps in the application of the modules. In an effort to prevent students using the software merely as a black box, the structural analysis is divided into three phases. Phase 1 uses the method of inspection to enable students to develop a feel for the structural behavior and anticipate some results. Phase 2 deals with detailed hand calculations and phase 3 involves the application of SkyCiv. For large structures, phase 3 can be applied directly and some results can be verified by phases 2 and 3.

SkyCiv was chosen over other software owing to its additional capability of reporting detailed hand calculations, as well its integration of structural design codes for solving structural design problems. Our future plans include utilizing the software to solve problems on buckling of columns and structural design, as design codes are integrated in SkyCiv. The main aim of the learning modules is to develop student competency in the application of commercial software for structural analysis for solving challenging capstone and other course projects, better preparation for workforce and graduate studies, complementing hand solution methods, and enhancing student engagement. Since SkyCiv was applied to discrete structures, the theoretical knowledge of FEA is not necessary. Learning or migrating to another FEA software should not be difficult as the key steps and menu features are similar.

The current paper is the first publication in the computer analysis of structures for architectural technology students. The advantage of SkyCiv software over other FEA software is that it is cloud based and reports detailed calculations identical to hand solutions, a feature that is not available in other commercial software. Therefore, one could use the software for verifying hand solutions too. The modules will be shared with other faculty members and made available to instructors of four engineering courses: two Mechanical Engineering courses- ME2210 Statics and ME 3210 Solid Mechanics, and two civil engineering courses- CE 3210 Structural Analysis I and CE 4220 Structural Analysis II.. The learning modules are currently used in the Architectural technology course CAD 2150: Statics and Strength of Materials. Student surveys will be administered for feedback and the effectiveness of the learning modules will be quantified.

Acknowledgements

The authors thank the three anonymous reviewers for their suggestions for minor revisions of the draft paper. The encouragement by Dr. Anca Sala, Dean of College of Engineering and Information Technology at Baker College Flint is greatly appreciated.

References

- [1] G. Yazici and Y.E. Yazici, "The Use of Finite Element Analysis Applications in Architectural Education," *International Journal on New Trends in Education and Their Implications*, Vol 3, no. 2, article 14, October 2012. [Online]. Available: <http://ijonte.org/FileUpload/ks63207/File/14.yazici.pdf>. [Accessed Jan. 2, 2020].
- [2] F.U.A. Shaikh, "Role of Commercial Software in Teaching Finite Element Analysis at Undergraduate Level 2012 a Case Study:," *Journal of Engineering Education, a Journal of the Higher Education Academy*, Vol. 7, no. 2, 2012. [Online]. Available: <https://www.tandfonline.com/doi/full/10.11120/ened.2012.07020002>. [Accessed Jan. 2, 2020].
- [3] R.G. Black and S. Duff, "A Model for Teaching Structures: Finite Element Analysis in Architectural Education," *Journal of Architectural Education*, Vol. 48, no. 1, pp 38- 55, 1994. . [Online]. Available: <https://www.tandfonline.com/doi/abs/10.1080/10464883.1994.10734621>. [Accessed Jan. 1, 2020].
- [4] G. Loreto and H. Reynoso, "Hybrid Method for Enhancement of Structural Understanding in Architecture Students," *Proceedings of the ASEE Annual Conference & Exposition, Tampa, FL, USA, June 15 -19, 2019*.
<https://peer.asee.org/hybrid-method-for-enhancement-of-structural-understanding-in-architecture-students>
- [5] SkyCiv, *Cloud Engineering Software*.

[6] R.E. Shaefer, *Elementary Structures for Architects and Builders*. Upper Saddle River, NJ: Prentice Hall, 2007.

[7] B.Onouye and K. Kane, *Statics and Strength of Materials for Architects and Building Construction*. Upper Saddle River, NJ: Prentice Hall, 2012.

[8] J. B. Wujek, *Applied Statics, Strength of Materials, and Building Structure Design*. Upper Saddle River, NJ: Prentice Hall, Inc., 1999.

[9] D.L. Logan, *A First Course in the Finite Element Method*. Boston, MA: Cengage Learning, 2012.