Development of the Web-Based Structure and Form Analysis System (SAFAS) for Architectural Education

M. Setareh[1], F. Bacim[2], N. Polys[2], and B. Jones[3]
[1]School of Architecture and Design, Virginia Tech, Blacksburg, Virginia, USA
[2]Department of Computer Science, Virginia Tech, Blacksburg, Virginia, USA
[3]School of Education, Virginia Tech, Blacksburg, Virginia, USA

Abstract - *This paper presents a collaborative effort among the Schools of Architecture and Design, Computer Science, and Education at Virginia Tech, Blacksburg, Virginia, to develop the web-based Structure And Form Analysis System (SAFAS) for the education of architects. The details of the software architecture, operations, and graphical user interface of SAFAS are discussed.*

Keywords: SAFAS, Web-Based Application, Architectural Structures, Architectural Education, Spatial Structures

1 Introduction

During the past decade there have been a number of attempts to use computers to enhance building design education. The target audiences for these software tools have been architecture, building construction, and engineering students (Messner and Horman [1]; Moloney and Amor [2]; Sulbaran and Crosby [3]; Kalisperis, et. al. [4]; Chou, et. al. [5]; Stojadinovic [6], and others).

Before the beginning of modern architecture, buildings were designed and built by master builders. They were in charge of creating the building from preliminary sketches to the final design and construction. However, due to the complexity of modern construction, the design process is now fragmented among several trades with each expert professional being responsible for his/her own part of the overall design and construction. Therefore, the modern building design process is an integrated collaborative effort, in which architects still have a major role; however, their main responsibility is limited to issues related to building aesthetics and function.

Even though structural engineers are generally responsible for the overall design of structural elements, architects must be familiar with the concepts related to building structures, and in particular, the inter-relationship between form and structure. For this reason the National Architecture Accreditation Board (NAAB) has mandated that all students in accredited architecture programs have instruction related to structural analysis and design.

There have been a few attempts and experimentations with the application of computer visualization and simulation as tools to teach structural behavior to architecture students with some success (Black and Duff [7]; Vassigh [8]; Setareh, et. al. [9,10]).

A collaborative effort supported by the National Science Foundation started in 2009 among the faculty members from the Schools of Architecture, Computer Science, and Education at Virginia Tech with the overall goal of integration of structures in architecture education. For the next three years, the team developed and tested the Structure and Form Analysis System (SAFAS). SAFAS is mainly the result of an attempt to close the gap between architecture and engineering education using an intuitive three dimensional (3D) graphical application. To ensure that SAFAS will be accessible to as many students as possible, a web-based approach was adopted. Upon the completion of the SAFAS development, it was used and tested by students of three architecture programs at the University of Illinois, Urbana-Champaign, Hampton University, and Virginia Tech.

This paper discusses the overall design of the SAFAS and focuses on the various aspects of software architecture and its graphical user interface. The application of the software in several architectural structures courses and different rounds of changes made to the SAFAS based on the analysis of the formative and summative evaluations can be found elsewhere (Setareh, et. al. [11]; Jones, et. al. [12]).

2 SAFAS and Spatial Structures

The main objective of this project is to help architecture students and architects to better comprehend the inter-relationship between building structure and form. The reader should note that in the context of this paper, "spatial structures" refer to structural systems made of interconnected linear elements. These systems are made of steel, aluminum, plastic or wood. They create architecturally appealing forms and are mostly used as long-span roof systems. It is assumed that the spatial structures are pin-connected and loads are applied at their connections (nodes or joints) only, thus, their members are under axial (tensile or compressive) forces only. This aspect of the spatial structures make them ideal candidates to be used for visualization of the inter-relationship between the form and structural behavior by the novice users.

3 Design of SAFAS

SAFAS consists of two main modules: Module One (Knowledgebase) and Module Two (Structure and Form Experimentation). SAFAS can be found at: http://legacy.caus.vt.edu/setareh/archresearch/

3.1 Module One (Knowledgebase)

This module includes several web pages with informative and educational materials as related to the various aspects of spatial structures.

Figure 1. Links to the Different Sections of the SAFAS-Module One

As mentioned by Setareh, et. al. [11], this includes: Introduction, History, Design, System, Advantages and Disadvantages, Assembly and Erection, Case Studies, Bibliography, and Fundamentals. Figure 1 shows the webpage which includes the links to the above mentioned sections of the SAFAS Module One.

A Dublin-Core metadata was used as a means to publish these resources in an accessible and searchable format (Web Consortium, www.w3c.org; National Science Digital Library, www.nsdl.org).

3.2 Module Two (Structure and Form Experimentation)

This module consists of software that can be used to: (1) create computer models of spatial structures, (2) subject them to various loading conditions, and (3) simulate the structural behavior by displaying the internal member forces and joint deformations. All the spatial structures used here are made of double-layer grids. This module was developed using Web3D standards and open source libraries using Java (Web3D Consortium, www. Web3d.org; Xj3D Java Toolkit, www.xj3d.org) to create a versatile software to be used on a wide variety of client platforms. The project development platform used Net Beans and SVN.

The main objective was to design user friendly software since the target audience was undergraduate architecture students with limited knowledge of structures and computer science. Another important software design goal was to create a portable, durable, and interoperable software tool. The interactive 3D contents of this module is portable as the open-source Xj3d rendering library (Java and OpenGL) provides cross-platform client application to visualize spatial structural models and their behavior under the applied loads. The 3D model visualization uses Extensible 3D (X3D) application, which is an open, royalty-free standard developed through the Web3D Consortium and ratified through the International Standardization Organization (ISO). This has made SAFAS durable as the models created through this application will be reproducible for many years to come. The import/export of the models using the X3D and VRML emphasizes the software interoperability.

SAFAS Module Two consists of two modes:

(1) Pre-Analysis Mode - This mode is used to define the structure and the applied loads.

(2) Post-Analysis Mode - This mode is used to simulate the effects of loads on spatial structures to assist student learning.

The software launches in the "Pre-Analysis" mode. Here, the user can build models of rectangular flat double-layer grid spatial structures using several different configurations. The software allows the user to create his/her own structural model, define the section properties, apply loads, and show the internal forces and structural deformations based on the results of the analysis by the structural analysis software. The user can also compare the results of the analysis of two different spatial structures.

Two structural analysis programs were used as the analytical engines for this module of SAFAS: (1) The open-source, PC-SAP4 [13], and (2) The commercial software, SAP2000 [14]. The development of SAFAS started with using the PC-SAP4 as a proof of concept, and was later replaced by the SAP2000. Both computer software reside on a remote server, and a username and password are required to gain access to them. The structural analysis program PC-SAP4 was originally developed in the early 1970's at the University of California, Berkeley, California. The SAP2000 is the commercial version of PC-SAP4 developed by the Computers and Structures, Inc. (CSI), Berkeley, California.

The simulation service is managed by a queue and each user's results are saved on both the client and the server. The communications between the SAFAS and SAP2000 are conducted using the SAP2000 API functions within a Visual Basic application.

3.2.1 SAFAS Module Two User Interface and Software Operations

Pre-Analysis Mode

Upon launching the software, the user can create a new project by clicking on the "create new structure" icon or by selecting "New" from the "File" menu bar. This opens a dialog box, as shown in Figure 2, which includes eleven different configurations of spatial structures in four groups, according to the top and bottom layer grid patterns.

The user can select one of the configurations and specify the total length, width, depth, and height. He/she can also define the number of modules to be used in each direction. The superimposed (S.I.) dead load and snow load will be entered and the support locations and types can be selected (corner, pyramid, tree or edge).

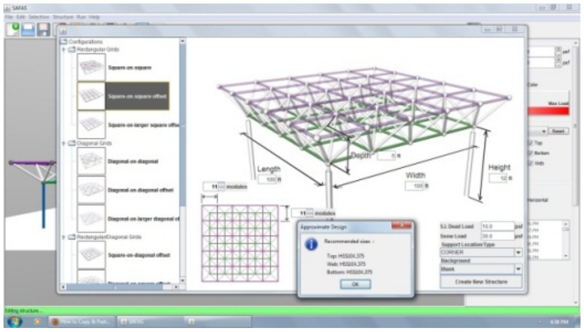


Figure 2. Initial Model Definition and Data Entry Control Panel

Also, several background options are provided for enhanced visualization of the model. These include: blank, grass, grass with trees, and user-specified. Based on the entered information, SAFAS conducts an approximate design and recommends member sizes for the top, web and bottom layers. At this point, the structural model is shown in the SAFAS main display (Figure 3). The load control panel (Figure 4), located on the right side of the display, allows the user to change the values of the applied loads. In addition, the nodal loads can be shown using the Glyph representation (arrows) or by coloring the nodes.

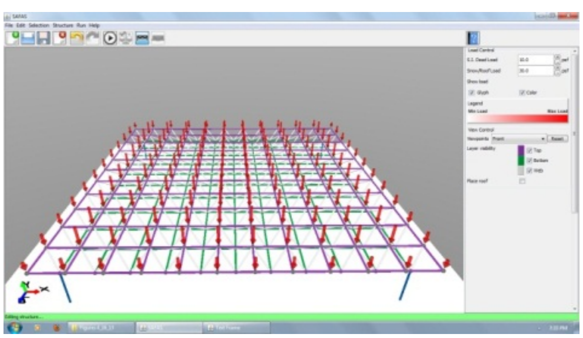


Figure 3. Complete Model of Spatial Structure

The heaviest-loaded node is colored in dark red and the lightest-loaded node in white. Various shades between these two limits are used for the remainder of the nodes.

The View Control panel (Figure 4) allows the user to define the viewpoint (front, rear, left, right or top) and the layer to be visible (top, bottom or web). These options can help the user turn on and off selected layers of the structure for enhanced visualization. The user can also place a continuous membrane on the top layer which helps visualization of the roof profile.

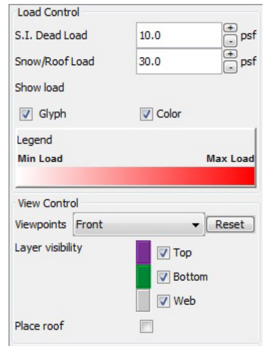


Figure 4. Load and View Control Panels

The user can manipulate the created structure using the options in the toolbars and control panels. The software has a conventional user interface in which the main menu, located at the top left corner of the window, includes: File, Edit, Selection, Structure, Run, and Help menu bars. At the bottom of the window, a status bar indicates the current state of the operations of the software. There is also a toolbar under the main menu bar which includes active buttons for the most commonly used tasks included in the main menu bar.

The following is a brief description of the various functions of the main menu bar (note that most actions within each drop-down menu include shortcut equivalents, as indicated below):

File: When this drop-down menu is selected, the following options will appear (Figure 5a):

- New <Ctrl/N>: This command creates a new file.
- Open/Manage <Ctrl/O>: This command opens an existing file, previously created by the software.
- Save <Ctrl/S>: This saves an existing file.
- Close <Ctrl/W>: This closes an existing file.
- Export: This saves the created file in a user-specified directory to be submitted to the SAP for analysis. This feature facilitates portability of the generated files to other instances of the software or a separate computer.
- Export X3D: This allows the software output to be viewed by any other X3D viewer application. It can also be imported into a Computer-Aided-Design (CAD) application.

Edit: This drop-down menu includes (Figure 5b):

- Undo <Ctrl/Z>: Undoes the last five tasks.
- Redo <Ctrl/Shift/Z>: Redoes the last five tasks.

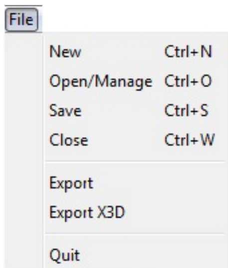
Selection: This drop-down menu allows different selection options (Figure 5c):

- Select All <Ctrl/A>: Selects the entire model including all the nodes and members.
- Clear Selection <Escape>: Clears the selected parts of the structure.

- Select Top, Select Web, Select Bottom: Selects the top, web, and bottom layers of the structure, respectively.
- Select Columns: Selects all the columns in the structure.
- Remove Selection <Delete>: Deletes the selected members of the spatial structure. It has to be noted that in reality the selected members are not removed from the structural model, rather their structural properties are substantially reduced.
- Selection by Stress: This option opens a dialog box, which allows selection based on the user-defined range of stresses for the top/bottom/web layer members (Figure 5d).

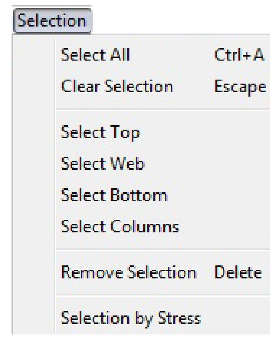
The user can also select the individual members and nodes by clicking on them or drawing a box around them to select a group of members. The color of the elements turns into red upon selection. Structure: This drop-down menu allows changes to the structure geometry, support conditions and applied loads, assigning member sizes, and computing the total structural weight. The various options include (see Figure 5e):

- Add single Column(s) to Selection <Ctrl/J>: After selecting node(s) from the bottom layer, using this option adds a column from the ground to the selected node(s).
- Place Columns Manually <Ctrl/K>: After selecting this option, clicking on a bottom layer node, places a column from the ground to the node at the location.

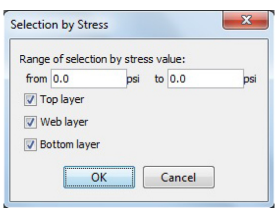


(a) File

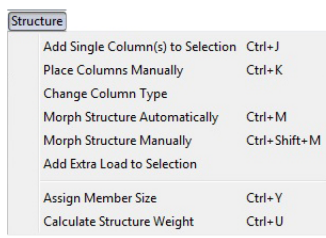
(b) Edit



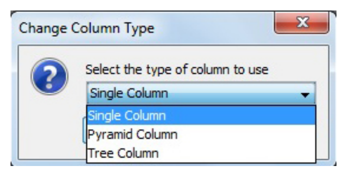
(c) Selection



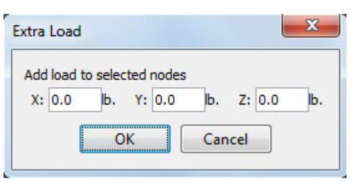
(d) Selection by Stress (Selection)



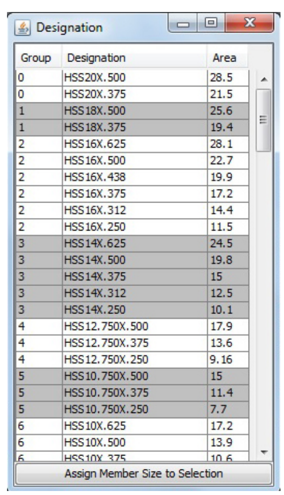
(e) Structure



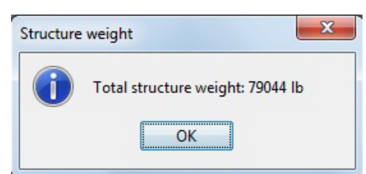
(f) Column Type (Structure)



(g) Add Extra Load to Selection (Structure)



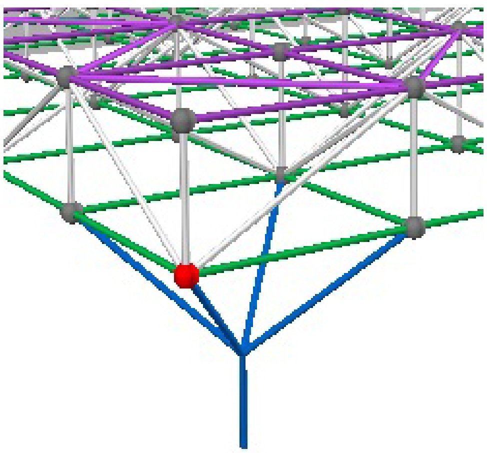
(h) Assign Member Size (Structure)



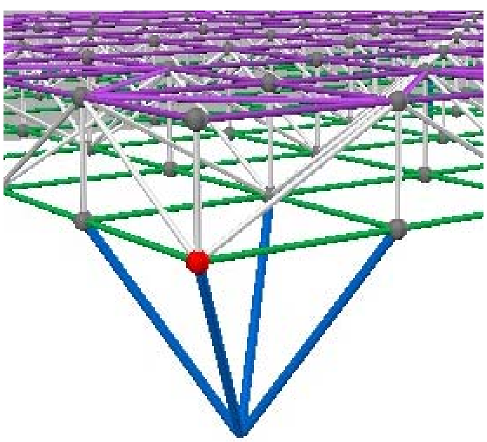
(i) Calculate Structure Weight (Structure)

Figure 5. Pre-Analysis Mode Menus and Options

- Change Column Type (see Figure 5f): This works in conjunction with the above two options for placing columns, and has to be selected first before adding columns. There are three possible choices: Single Column (a straight column from the ground to the node), Pyramid Column (a straight column connected to a module), or a Tree Column. Figure 6 shows the pyramid and tree column options
- Morph Structure Automatically <Ctrl/M> (Figure 5e): Activating this option, opens the "Morph Control Panel" (Figure 7a.), on the right side of the window. The software allows two morph geometries: Dome (representing a sphere-like deformation) and Vault (representing a cylindrical deformation). Each morph option includes three pre-defined functions for morphing the structure (see Figure 7a): Smooth (a bell-curve roof profile), Linear (a linear, double-pitch roof profile), Barrel Vault (a parabolic roof profile), and Uniform (a flat roof profile).



(a) A Pyramid Column



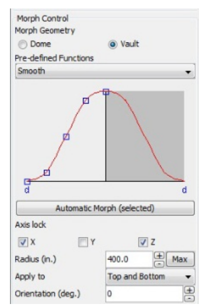
(b) A Tree Column

Figure 6. Pyramid and Tree Columns

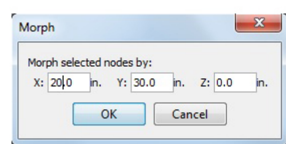
The morph control panel allows the changes to the shape of the selected parts of the structure to be applied along a particular axis by using the "Axis Lock" option. The user can define the morph radius from the selected node (extent that the morph function is applied). The "Max" button selects the entire roof structure. The morphing function can be applied to an individual layer or both layers by selecting the choice from the "Apply to" option. The default orientation of morph is along the x-axis; however, it is possible to modify this by changing the value in the "Orientation (deg)" box. This changes the orientation of morph with respect to the x-axis.

The automatic morph allows the user to change the shape of the structure by moving the selected nodes by a fixed value along the x, y, and z axes. Upon selecting the "Automatic Morph (selected)" in the "Morph Control Panel", a dialog box opens (Figure 7b.), which allows the user to enter the desired amount of morphing along the three axes.

- Morph Structure Manually <Ctrl/Shift/M> (Figure 5e): This allows the user to place the cursor on a node from which an area is selected (this is identified by changing the color of the nodes), set by radius. The user can then move the cursor and deform the structure using the various curve options of the automatic morphing that was explained above.



(a) Morph Control Panel



(b) Automatic Morph Dialog box

Figure 7. Morph Control Panel and Automatic Morph Dialog box

The amount of displacements along the three axes and the coordinates of the selected node during the morphing process are shown in a text box located at the upper right corner of the window. The status bar at the bottom of the screen shows the current mode of the software. Figure 8 shows a morphed spatial structure.

- Extra Load to Selection (Figure 5e): This option allows the user to apply additional loads (in addition to the gravity loads) along the x, y, and z axes. The user first selects the individual or a group of nodes, and then uses this option, which opens a dialog box (Figure 5g) to enter additional loads on the structure.

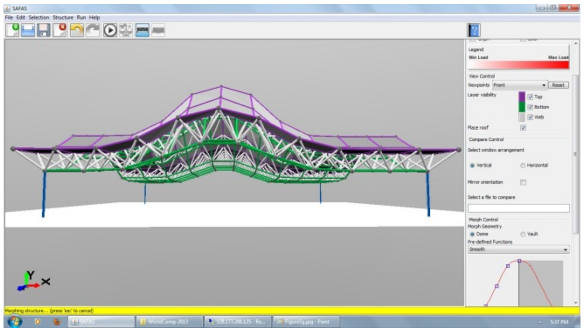


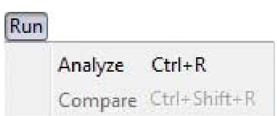
Figure 8. A Morphed Spatial Structure

- Assign Member Size <Ctrl/Y> (Figure 5e): Upon selection of members and using this option, a "Designation" dialog box opens (see Figure 5h). This dialog box includes a list of the available standard steel Hollow Structural Shapes (HSS) that can be used for the spatial structure members. This list includes the designation and cross-sectional area of each shape. All the shapes are placed in groups that have the same nominal outside diameter, as specified by the Manual of Steel Construction published by American Institute of Steel Construction [15]. The user can sort this table based on different parameters such as group number, designation or the cross sectional area by clicking each header. To complete the member size assignment, the user must select "Assign Member Size to Selection" in the Designations dialog box.
- Calculate Structure Weight <Ctrl/U> (Figure 5e.): This option computes the total weight of the structure based on the member sizes used (Figure 5i).

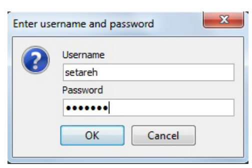
<u>Post-Analysis Mode</u>

<u>Run</u>: This option includes the submission of the model to SAP for the structural analysis, and comparison of the results of two spatial structures:

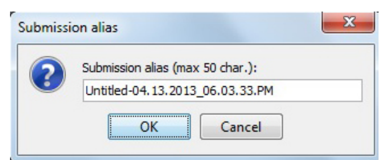
- Analyze <Ctrl/R> (Figure 9a): This options is used when the modeling is complete and the user intends to submit the structure to the SAP, structural analysis program, for computation of nodal displacements and member forces. Upon choosing this option, a dialog box opens in which the user enters his/her assigned username and password (Figure 9b). This feature ensures the security of access to the server. This dialog box is followed by a second box which contains the default file name, consisting of the word "untitled", and the date and time that the model was created (Figure 9c). The users can change this default name as they wish. This dialog box is followed by a third box, which indicates that "the submission is in progress."(Figure 9d).



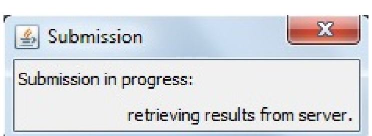
(a) Run



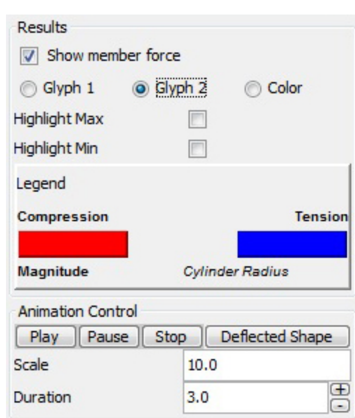
(b) User Information Check for file Submission



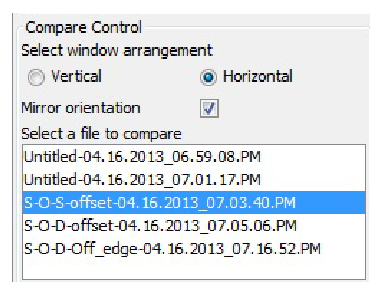
(c) Setting Filename



(d) SAP Analysis in Progress



(e) Results and Animation Control Panels



(f) Compare Control Panel

Figure 9. Post-Analysis Mode Menus and Options

At this time, the SAP input file is created and submitted to the server that the PC-SAP4 or SAP2000 computer software resides on. Upon the completion of the analysis, this dialog box is automatically closed. This also changes the status of the software to the "post-analysis" mode, and adds the post-analysis controls to the panel on the right side of the screen. This includes the "Results" and "Animation" controls. The "Results" allow the user to select how the member forces are to be shown. These include two glyph options: Glyph 1 (using cones to demonstrate if the member is in tension or compression and their size to indicate the relative size of the internal force) (Figure 10a), Glyph 2 (shows the member forces by coloring them, "red" to represent compression members and "blue" for tensile members (Figure 10b). The diameter of cylinders represents the relative internal force magnitude). The third option is "color", which colors the compression members in red and tensile members in blue. The color shading changes based on the member force magnitude (Figure 10c). The "Highlight Max", and "Highlight Min" in the Results Control Panel can be used to identify the members having the largest and smallest forces (Figure 9e.)

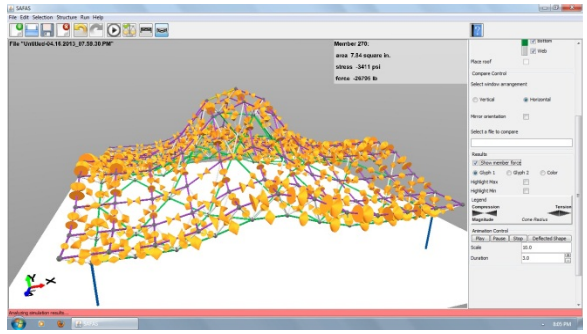
Upon placing the cursor on a member, the internal forces and stresses are given in a textbox located at the top right corner of the screen. Also, when the cursor is placed on a node, the deflection and position of node are shown (see Figure 10.)

To help visualize the deformation of the structure, the animation control shows moving images of the structure when subjected to loads. The video can be started, stopped, and paused at any time by the user. The static deflected shape can also be displayed. The amplitude of the deformation can be increased for very stiff systems or reduced for unusually flexible structures. The animation duration can also be changed by the user for better visualization (see Figure 9e).

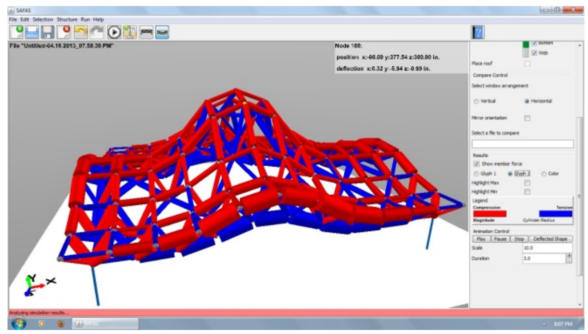
- Compare <Ctrl/Shift/R> (Figure 9a): Once the analysis is complete, the user can compare the results of the behavior of two structures using the Compare control panel (Figure 9f). Upon activating the compare option (Figure 9a), a split window appears. The user can then select a second structural model from the compare control panel "Select a file to compare" (see Figure 9f). The split window can be oriented horizontally or vertically. The images in the two windows can be linked so that the two structures will be observed from the same viewpoint. This is accomplished by selecting the "mirror orientation" in the compare control panel (Figure 9f). Figure 11 show the comparison of two structural models in a horizontally split window.

4 Summary and Conclusion

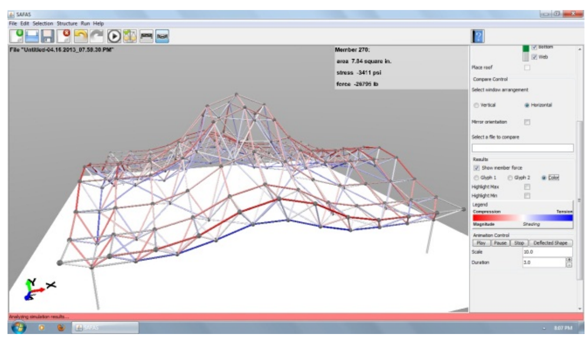
This paper presented the details of the software architecture, operations, and the graphical user interface of the "Structure And Form Analysis System (SAFAS)." SAFAS was developed through a collaborative efforts by a team of researchers at Virginia Tech to help architecture students better understand the inter-relationships between structure and form. This web-based software was used in several architectural structures courses at Virginia Tech and two other universities. The analysis of students' performance showed that SAFAS is an effective educational tool for learning about structural performance.



(a) Glyph 1



(b) Glyph 2



(c) Color

Figure 10. Various Modes of SAFAS Results Representations

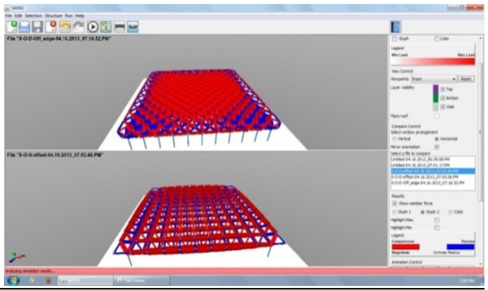


Figure 11. Comparing Two Structural Models in SAFAS

5 Acknowledgements

This study was supported by the National Science Foundation under Grant No. CCLI/TUES 0817106. This support is gratefully acknowledged. Any opinions, findings, and conclusions expressed in this paper are those of the writers and do not necessarily reflect the views of the National Science Foundation.

References

[1] Messner, J., and Horman, M., "Using Advanced Visualization Tools to Improve Construction Education", In *Proceedings of the Conference on Construction Applications of Virtual Reality-CONVR 2003,* Blacksburg, Virginia, pp. 145-155, 2003.

[2] Moloney, J., and Amor, R., "StringCVE: Advances in a Game Engine-Based Collaborative Virtual Environment for Architectural Design", In *Proceedings of the Conference on Construction Application of Virtual Reality-CONVR 2003,* Blacksburg, Virginia, pp. 156-168, 2003.

[3] Sulbaran, T., and Crosby, W., "Creating a Distributed Virtual Reality Environment to Enhance Engineering Students' Abilities in Crane Selection", In *Proceedings of the Conference on Construction Application of Virtual Reality-CONVR 2003,* Blacksburg, Virginia, pp. 169-178, 2003.

[4] Kalisperis, L., Otto, G., Muramoto, K., Gundrum, J., Masters, R., and Orland, B., "An Affordable Immersive Environment in Beginning Design Studio Education", In *Proceedings of the 2002 Annual Conference of the Association for Computer Aided Design in Architecture,* Ponoma, California, pp. 49-56, 2002.

[5] Chou, C., Hsu, H., and Yao, Y., "Construction of a Virtual Reality Learning Environment for Teaching Structural Analysis", *Computer Applications in Engineering Education*, Vol. 5, pp. 223-230, 1997.

[6] Stojadinovic, B., "Conceptual Design of a Haptic Interface for Structural Analysis Software", In *Proceedings of the ASCE 2000 Structures Congress,* Philadelphia, Pennsylvania, 2000.

[7] Black, R. G., and Duff, S., "A Model for Teaching Structures: Finite Element Analysis in Architectural Education", *Journal of Architectural Education*, Vol. 48, No. 1, September, 1994.

[8] Vassigh, S., "Interactive Structures: Visualizing Structural Behavior", CD-ROM, *John Wiley and Sons, Inc.*, Hoboken, New Jersey, 2005.

[9] Setareh, M., Bowman, D., and Kalita, A., Gracey, M., and Lucas, J. "Application of the Virtual Environments in Building Sciences Education and Practice", *Journal of Architectural Engineering*, American Society of Civil Engineers, Vol. 11, No. 4, pp. 165-172, 2005.

[10] Setareh, M., Bowman, D., and Kalita, A., "Development of a Virtual Reality Structural Analysis System", *Journal of Architectural Engineering*, American Society of Civil Engineers, Vol. 11, No. 4, pp. 156-164, 2005.

[11] Setareh, M., Bacim, F., Jones, B., Polys, N., Geng, T., and Orsa, B., "Integrating Web-Based Visualization with Structural System Understanding to Improve the Technical Education of Architects", *Journal of Online Engineering Education*, Vol. 3, No. 1, 2012.

[12] Jones, B., Setareh, M., Polys, N., and Bacim, F., "Application of an Online Interactive Simulation Tool to Teach Engineering Concepts Using 3D Spatial Structures", *International Journal of Technology and Design Education*, Springer Publishing Company, under review, 2013.

[13] Maison, B., "PC-SAP4. A Computer Program for Linear Structural Analysis", *Earthquake Engineering Research Center,* University of California, Berkeley, California, 1994.

[14] Computers and Structures, “SAP2000 – Linear and Nonlinear Static and Dynamic Analysis and Design of Three-Dimensional Structures”, *Computers and Structures, Inc.*, Berkeley, California, 2011.

[15] American Institute of Steel Construction, "Manual of Steel Construction", *AISC*, Chicago, Illinois, 2010.