Purdue University
Purdue e-Pubs

ASEE IL-IN Section Conference | 2019 ASEE IL-IN Section Conference

Application of Computational Tools to Spaghetti-Based Truss Bridge Design

Jin Xu
Purdue University Northwest, xu1265@pnw.edu

Jiliang Li
Purdue University Northwest, jiliang.li@pnw.edu

Nuri Zeytinoglu
Purdue University Northwest, nz@pnw.edu

Jinyuan Zhai
Purdue University Northwest, Zhai33@pnw.edu

Follow this and additional works at: https://docs.lib.purdue.edu/aseeil-insectionconference

Part of the Applied Mechanics Commons, Civil Engineering Commons, Construction Engineering and Management Commons, Engineering Education Commons, Engineering Mechanics Commons, Mechanics of Materials Commons, Polymer and Organic Materials Commons, Structural Engineering Commons, and the Structural Materials Commons

Xu, Jin; Li, Jiliang; Zeytinoglu, Nuri; and Zhai, Jinyuan, "Application of Computational Tools to Spaghetti-Based Truss Bridge Design" (2019). *ASEE IL-IN Section Conference*. 4.
https://docs.lib.purdue.edu/aseeil-insectionconference/2019/technology/4

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Application of Computational Tools to Spaghetti-Based Truss Bridge Design

Abstract *Statics* and *Strength of Materials* are two foundational courses for Mechanical/Civil Engineering. In order to assist students in better understanding and applying concepts to a meaningful design task, *SolidWorks* and theoretical calculation were used for a spaghetti-bridge design contest with the constraints of given maximum weight and allowable support-material weight. As the first step of this iterative designing process, both extrude feature and structural member were introduced to model planar bridge trusses. Then *SolidWorks*' Statics module was used to run FEA analysis of the structural performance in efforts to optimize the load-carrying capacity of the structure. To make simulation possible, a universal material-response testing apparatus was used to measure the key mechanical properties of the bridge material, namely spaghetti bundles, and add it to *SolidWorks*' material database. The building stage started upon completion of design refinement, and the project culminated with performance prediction (as to the weakest spots of the structure) and testing.

The theoretical calculation went down two paths—A full truss analysis was performed based on the method of joints, along with more thorough FEA analysis through coding, before comparing the internal forces, displacements, etc., with the simulation results. Through the holistic design process, the course turned out to be more engaging and students gained experience of solving a typical real-life engineering problem involved with trade-off between economy and quality.

Keywords: *SolidWorks*' Statics, Spaghetti-Based Truss Bridge, Finite Element Analysis

I. Introduction

Statics and *Strength of Materials* are two foundational courses in the Mechanical/Civil Engineering curricula [1]. One of the course objectives is to enable students to perform structural analysis and predict failure, given material properties. Nonetheless, unless the two separate semester-long courses are compressed into a single course, it is after completion of solid mechanics that students can apply concepts to such projects that their main purpose is to put forward a cost-effective structural design with the required load-carrying capability. This is because concepts like structural analysis, stress/strain, failure prediction, *etc*. must be introduced beforehand. Our study at Purdue University Northwest (PNW) has been an exploration of using a spaghetti-bridge design contest towards the end of the *Statics* semester to fulfill the aforementioned goals. In furnishing students guidance of design optimization, *SolidWorks* simulation and theoretical calculations were taught, both of which are based on finite-element analysis (*FEA*). Regarding the simulation option, *SolidWorks Statics* module [2] was used to run *FEA* analysis of the structural performance in order to maximize the load capacity. To make simulation possible, a universal material-response testing apparatus was used to measure the elastic modulus and yield strength of the bridge material, namely spaghetti, before creating a new material in *SolidWorks*. Regarding the theoretical-calculation option, students concurrently taking or had taken *FEA* (an elective at PNW) could opt for the coding-based design tool.

Through the iterative design process, students gained experience of solving a typical real-life engineering problem involving trade-off between economy and quality. Post-project survey indicated that the course had turned out to be more engaging and students had developed more consciousness of and confidence in using Engineering tools in their design projects for other courses.

II. Methodology

A design project is usually used to facilitate the application of concepts to a meaningful engineering task. Our project assignment was to design, build and test a bridge structure made out of spaghetti up to 2 *lbs* to support a minimum load of 20 *lbs*, applied at the mid-span. The bridge should span 20~24 inches and be 4~6 inches wide. The majority of the structure must be spaghetti, and the support material, which usually is certain super glue, must be kept below 2% of the total weight. The weight and dimension constraints were placed to mimic the trade-off situation between economy and quality that a typical real-life Engineering problem presents. Students could start with commonly seen bridge structures, such as Pratt, Howe, Warren, *etc*., before modifying them and arriving at their own design. In the iterative design process, *FEA* could be used as a tool to increase the structure strength and refine the structure. Fig. 1 shows the design that won this year’s contest.

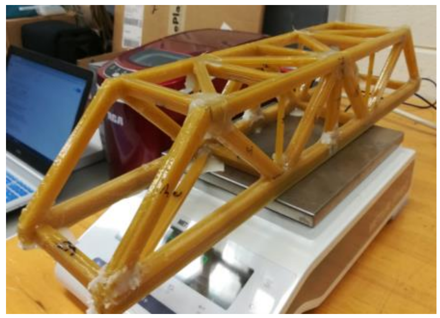


Fig. 1 Contest winner with 611-*lb* load-carrying capability

Since there is no course exclusively devoted to computer-aided engineering or simulation at PNW, a *SolidWorks* simulation lab was given to prepare students for the class project. More specifically, simple 2D truss systems were modeled, using both the Extrude and Structural Member features in *SolidWorks*, prior to the use of *Static* module for computation of reactions, internal forces, axial stresses, *etc*. Fig. 2 shows the displaced truss subjected to two concentrated loads, along with the axial stress distribution, in the lab tutorial.

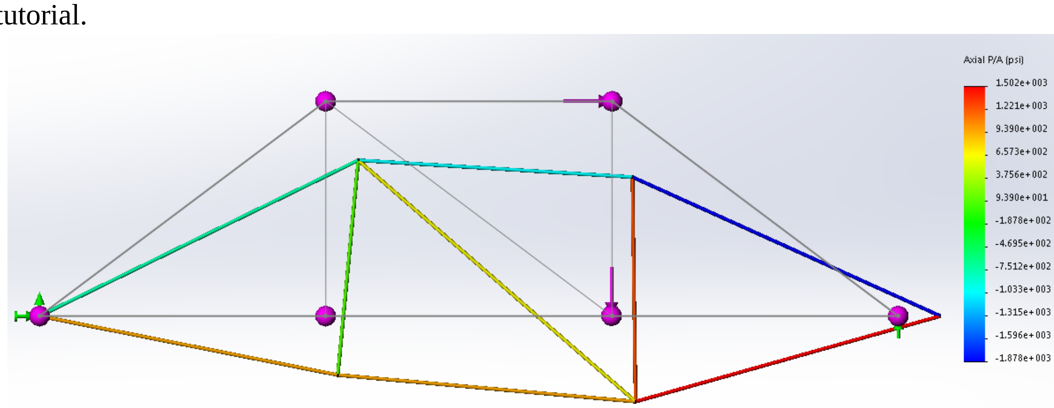


Fig. 2 Plane truss simulation used in the *SolidWorks FEA* simulation lab

Like any *FEA* software, *SolidWorks Static* module divides the entire body (either a single part or an assembly) in a process termed meshing [3], into many geometrically small simple bodies termed elements. Each element leads to an algebraic equation based on the theory of elasticity. Then all the equations are simultaneously solved. One thing students need to pay attention to is mesh convergence, *i.e.*, grid sensitivity. This means that each simulation task is iterative — once preliminary results are obtained using a relatively coarse mesh, mesh control should be applied whenever there are singularities and/or great gradients in key quantities like stresses. Moreover, the average mesh size should be decreased to an extent that no noticeable change in the results, *e.g.*, stress distribution, could be observed.

In addition to the *FEA* simulation as a design tool, students could opt to perform theoretical calculations for a full truss analysis and determine the bar subjected to the greatest internal force. This is also based on *FEA* and entails a bit of coding. Its principal steps include (a) Discretization, for which the simplest line element is chosen, namely one element for each bar. (b) Selection of a displacement function, for which a linear one is chosen for simplicity; (c) Derivation of the element stiffness matrix [*k*] and equation for each bar. Eq. (1) is the element stiffness equation for a generic bar arbitrarily oriented as indicated in Fig. 3 wherein θ is positive when measured counterclockwise from x to x' and c and s are short for cosine and sine of θ, respectively. In addition, u and v represent the two global displacement components.

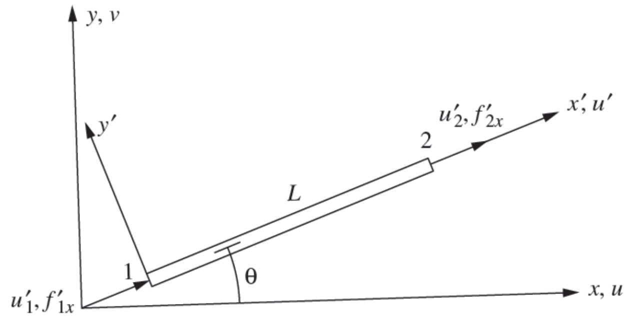
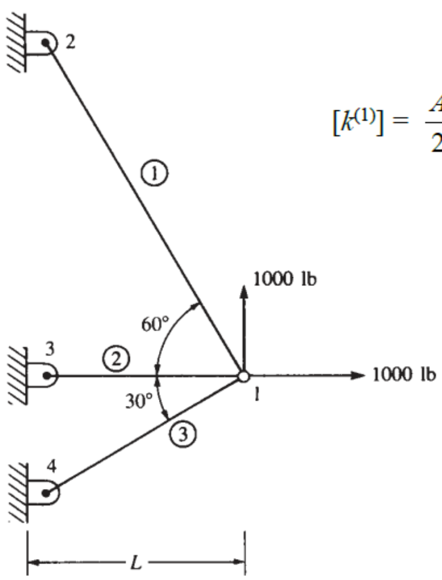


Fig. 3 A bar element arbitrarily oriented in the global x –y plane [4]

$$\begin{Bmatrix} f_{1x} \\ f_{1y} \\ f_{2x} \\ f_{2y} \end{Bmatrix} = [k]\{d\} = \frac{AE}{L}\begin{bmatrix} c^2 & cs & -c^2 & -cs \\ & s^2 & -cs & -s^2 \\ sym & & c^2 & cs \\ & & & s^2 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} \qquad \text{Eq. (1)}$$

In the above equation, the f 's are the force components applied at either end and expressed in the global coordinate system [5]. Moreover, A is the cross-sectional area, E Young's modulus, and L the length. The next step is: (d) Assembling of all element stiffness equations, using the direct stiffness method, and enforcement of load and boundary conditions. For illustration, Fig. 4 shows the assembling process through which to obtain the global equation for a three-bar truss.



$$[k^{(1)}] = \frac{AE}{2L}\begin{bmatrix} \frac{1}{4} & -\frac{\sqrt{3}}{4} & -\frac{1}{4} & \frac{\sqrt{3}}{4} \\ -\frac{\sqrt{3}}{4} & \frac{3}{4} & \frac{\sqrt{3}}{4} & -\frac{3}{4} \\ -\frac{1}{4} & \frac{\sqrt{3}}{4} & \frac{1}{4} & -\frac{\sqrt{3}}{4} \\ \frac{\sqrt{3}}{4} & -\frac{3}{4} & -\frac{\sqrt{3}}{4} & \frac{3}{4} \end{bmatrix} \qquad [k^{(2)}] = \frac{AE}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$[k^{(3)}] = \frac{\sqrt{3}AE}{2\,L}\begin{bmatrix} \frac{3}{4} & \frac{\sqrt{3}}{4} & -\frac{3}{4} & -\frac{\sqrt{3}}{4} \\ \frac{\sqrt{3}}{4} & \frac{1}{4} & -\frac{\sqrt{3}}{4} & -\frac{1}{4} \\ -\frac{3}{4} & -\frac{\sqrt{3}}{4} & \frac{3}{4} & \frac{\sqrt{3}}{4} \\ -\frac{\sqrt{3}}{4} & -\frac{1}{4} & \frac{\sqrt{3}}{4} & \frac{1}{4} \end{bmatrix}$$

$$\begin{Bmatrix} 1000 \\ 1000 \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} \frac{1}{8}+1+\frac{3\sqrt{3}}{8} & -\frac{\sqrt{3}}{8}+0+\frac{3}{8} \\ -\frac{\sqrt{3}}{8}+0+\frac{3}{8} & \frac{3}{8}+0+\frac{\sqrt{3}}{8} \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \end{Bmatrix}$$

Fig. 4 A truss along with the assembling step for the global equation [4]

The last two steps are: (e) Solution for the unknown displacements, using such numerical methods as Gauss-Seidel [6], and (f) Solution for secondary quantities, *e.g*., the element strains and stresses. For example, the axial stress σ equals $E/L \cdot [-c \quad -s \quad c \quad s] \cdot d$ where the displacement vector *d* is the 4 by 1 column vector as in Eq. (1).

III. Discussion and Results

Our *Statics* class was divided into fourteen teams of 2~3 for a project. In principle, each student should devote 10 hours to the project which accounts for five points for the course. The project assignment was to design, build and test a bridge structure made out of spaghetti up to 2 *lb* to support a minimum of 20-*lb* load. *FEA* was used as the design tool, and a *SolidWorks* simulation lab using the *Statics* module was given for truss analysis to prepare students for the project. In the lab, both extrude feature and structural-member feature were introduced to model planar trusses. Considering that not everyone was proficient with *SolidWorks* modeling, two-dimensional systems were chosen for the 75-min lab. The two other reasons for such choice are: (1) many 3D structures can be approximated to 2D ones without considerable loss in accuracy of the results; and (2) students can easily extend the procedure to 3D cases, assuming they can spend less than half an hour outside the classroom practicing 3D modeling. Only one team actually modeled the *3D* truss structure that they built and simulated its performance, though.

Besides *SolidWorks* simulation as the *FEA* design tool, a coding-based alternative was given and the training was offered on an available-upon-request basis. To encourage students to choose this *FEA* option, admittedly we limited the *FEA* case scenario to the simplest possible one; for instance, it took only one line element for each bar and the linear displacement function was selected. However, the choice was based upon its practicality, given the permissible time to train the students. Moreover, one team chose the method of joints for truss analysis, which had been covered in *Statics*. Its gist is applying equilibrium of a particle to each of the joints and producing a linear set of equations. Nonetheless, it could be used to compute only the internal force for each bar. Thus *FEA* as a more thorough analysis tool should be recommended because the class project would also serve as a great opportunity to introduce *FEA* as a useful computational tool for mechanics.

FEA was performed in the design process to help maximize the load-carrying capability of each structure. Whether it is in the form of *SolidWorks* simulation or theoretical calculations, it is inevitable to quantify the necessary elastic modulus for the spaghetti, since student designs vary in terms of the “material” and the dimension of each “bar”. For instance, some designs might bundle quite a few strands of angel hair together, whereas others may choose fewer strands of spaghetti. Besides, strands might stick together, using epoxy or by being dipped into boiling water. As such, a universal stress/strain apparatus [7] was used to obtain the material response primarily for elastic modulus for our purposes. Fig. 5 shows the sample material response under tensile testing for a single strand of *dry* angel hair. Once again, student designs may use multiple strands bundled together for the “bars”. Post-processing of such curve involves computation of elastic modulus (equal to 410 *ksi* for the sample) and tensile strength (estimated to be 3 *ksi* for the sample), prior to their use in the *FEA* procedure. To enter the key mechanical properties for *SolidWorks* simulation, as an example, nylon could be used as the base material for the above sample to create a custom material and supplement the material library for *SolidWorks*.

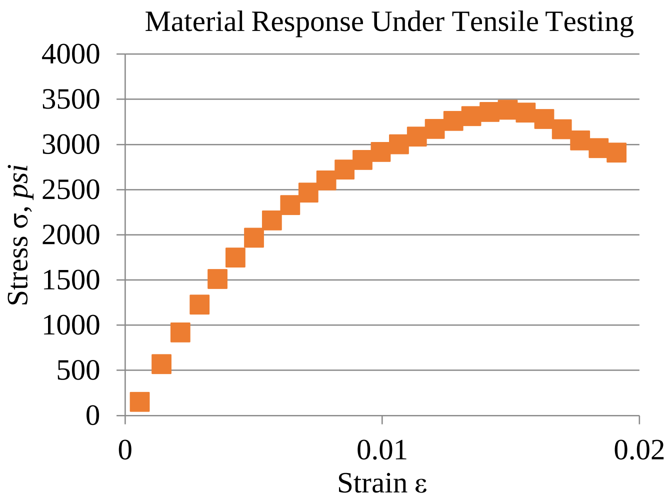


Fig. 5 Stress-strain curve for one strand of angel hair

The building stage started upon completion of design refinement, and the project culminated with performance prediction (*e.g.*, the weakest bar and the max load of the structure) prior to testing. Fig. 6 shows the loading machine for testing and a design in test. Each bridge was held fixed at one end, and the load, applied at the mid-span, was increased slowly until failure in the form of fracture or considerable deformation. Survey was conducted on the site, which indicated that 86% of students found they could better link structural-analysis concepts to their physical meanings after the project. All students reported that the project increased the engagement of the course and that they had become more conscious of using computer-aided Engineering software to aid in their later projects or tasks. Moreover, 71% expressed more confidence in learning new software or new modules/functionality in a familiar software package. Things that could be improved on include that more time than three weeks should have been given for the project and that the instructor should have devoted more time to explanation of structural failure.

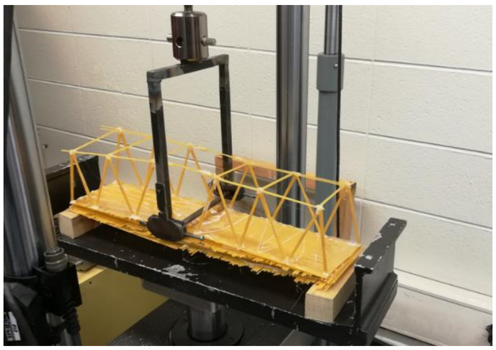


Fig. 6 A bridge design in test

In general, the project served its purpose well; that is, a single *Statics* course was sufficient to assist students in understanding a typical structural analysis and failure prediction. It is worth noting that *Elementary Engineering Design*, a freshman-level course at PNW, contributed to the success of our exploration. For instance, it helped visualize force transfer along the two-force bars, and piqued students' interests in bridge design. On the other hand, through our project students were further intrigued by bridge design, taking material selection into account. Furthermore, such a project in *Statics* or *Mechanics of Materials* could make student better appreciate their later concrete or steel bridge design competitions, *etc*. Also, students could better understand bridge material test and bridge structural test, which are important as it often is the weakest link during a structural test that dominates the bridge failure.

IV. Concluding Remarks

A design project was used to facilitate the understanding and application of concepts covered in *Statics*. To design a spaghetti bridge, students learned to perform structural analysis and predict failure, without having to wait until completion of both *Statics* and *Strength of Materials*. In the iterative design process, *FEA* was used as a design tool in the form of *SolidWorks* simulation or coding-based calculations. Whichever was chosen, the elastic modulus and yield strength of the bridge material were measured with a universal material testing machine. And these mechanical properties made a truss analysis possible, which produced such information as internal forces, displacements, stresses, *etc*. Through the holistic design process, students gained experience of solving a typical engineering problem involving trade-off between economy and quality. Survey indicated that students had developed more consciousness of and confidence in using engineering tools in their design projects for other courses.

REFERENCES

1) William C. Oakes and Les L. Leone, *Engineering Your Future: A Comprehensive Introduction to Engineering (9th Ed.)*, Oxford, 2016.
2) Radostina V. Petrova, *Introduction to Static Analysis Using SolidWorks Simulation (1st Ed.)*, CRC Press, 2014.
3) Robert D. Cook, and *et al*, *Concepts and Applications of Finite Element Analysis (4th Ed.)*, Wiley, 2001.
4) Daryl Logan, *A First Course in the Finite Element Method (6th Ed.)*, Cengage, 2016.
5) Tirupathi R. Chandrupatla and Ashok D. Belegundu, *Introduction to Finite Elements in Engineering (4th Ed.)*, Pearson, 2012.
6) James F. Epperson, *An Introduction to Numerical Methods and Analysis (2nd Ed.)*, Wiley, 2013.
7) R. C. Hibbeler, *Statics and Mechanics of Materials (5th Ed.)*, Pearson, 2016.