Improving Braced Frame Modeling Application

by
Corey Groshong

A THESIS

submitted to

Oregon State University

Honors College

in partial fulfillment of
the requirements for the
degree of

Honors Baccalaureate of Science in Civil Engineering
(Honors Scholar)

Presented February 27, 2020
Commencement June 2020

AN ABSTRACT OF THE THESIS OF

Corey Groshong for the degree of Honors Baccalaureate of Science in Civil Engineering presented on February 27, 2020. Title: Improving Braced Frame Modeling Application.

Abstract approved:____

Barbara Simpson

This work analyzes the numerical curvature of buckling braces. An existing brace modeling learning tool, which in its current state allows users to change the inputs within a simulated model of a braced element, was modified to display the curvature diagram. The curvature diagram was verified through comparison to existing theories on elastic curvature. Numerical results from the tool were compared to the results from experiments. Parametric studies were conducted using the numerical modeling inputs available within the learning tool. The dependence of curvature on modeling inputs can be rationally explained using the assumptions for force-based and displacement-based elements. The addition of curvature to the existing learning tool will aid users in visualizing how numerical model inputs, including element types, number of sub-elements and integration points, distribution of sub-elements, and strain hardening, can affect the numerical results, such as curvature, compared to experimental values.

Key Words: Curvature, Brace Frame Modeling, Structural Modeling, Force-Based Elements, Displacement-Based Elements, Learning Tool

Corresponding e-mail address: groshoco@oregonstate.edu

Improving Braced Frame Modeling Application

by
Corey Groshong

A THESIS

submitted to

Oregon State University

Honors College

in partial fulfillment of
the requirements for the
degree of

Honors Baccalaureate of Science in Civil Engineering
(Honors Scholar)

Presented February 27, 2020
Commencement June 2020

Honors Baccalaureate of Science in Civil Engineering project of Corey Groshong presented on February 27, 2020.

APPROVED:

____

Barbara Simpson, Mentor, representing Civil & Construction Engineering

____

Michael Scott, Committee Member, representing Civil & Construction Engineering

____

Erica Fischer Committee Member, representing Civil & Construction Engineering

____

Toni Doolen, Dean, Oregon State University Honors College

I understand that my project will become part of the permanent collection of Oregon State University, Honors College. My signature below authorizes release of my project to any reader upon request.

____

Corey Groshong, Author

Table of Contents

List of Figures

List of Tables

1 Introduction

Many computer programs exist to help users with complex structural calculations. These programs, such as SAP or RISA, involve various geometric, material, section, and loading as inputs that result in forces, displacements, stresses, etc. as outputs. While useful, many of the calculations performed by these programs aren't visible to the user. The variables used in these calculations, such as the number of elements/integration points for an element, are often not accessible to the user even though their values can affect the numerical results. The braced frame modeling learning tool on the SimCenter website is a program that illustrates the impact of computer modeling on numerical response. It allows the user visualize how the results of the analysis of a brace element change with changing numerical inputs (Simpson, McKenna, & Gardner, 2018). Additions to this learning tool are the focus of this paper.

The learning tool in its original state was an already functioning program, which graphically showed changes to the displaced shape, axial force, and moment diagram of a brace element over time. The brace element could be divided into multiple elements as specified by the user for analysis. Those elements could be further separated into integration points for Gauss-Lobatto or similar integration over the element length. Each integration point was also defined by a section representing the cross-section of the brace, which can be separated into multiple fibers to integrate over the section. In the tool, each variable used to discretize the brace into smaller segments can be modified to see how the results are affected. The tool also allows for many other factors to be changed, such as the type of analysis, member size and material, connection type, etc. All of these changes allow the user to compare the numerical results to pre-uploaded experimental measurements. Each manipulation provides an interactive way for the user to visualize both significant and minor changes in the numerical results.

To further develop this program, a diagram showing curvature was added to the graphical user interface. Creating and analyzing the curvature diagram, which will be discussed in detail throughout this paper, involved the following steps. First, the diagram was added to the code for the program, involving multiple iterations and troubleshooting. After the diagram was added to the program and the code had been verified, a parametric study was conducted to observe how changing variables affect the curvature output of the learning tool. The variables manipulated included: number of integration points/number of elements used, strain hardening, concentrated vs. distributed elements, and force-based vs. displacement-based element types. Changing these variables had different effects on the results and overall accuracy of the model. Importantly, they also affected curvature in different ways. This paper will focus on the steps taken to add the curvature diagram to the existing learning tool software, and the unique ways the variables listed affect the overall curvature response displayed in the tool.

2 Background

Before analyzing the information extracted from the program, it is important to understand how the brace was discretized for analysis. It is also necessary to understand the mechanics of curvature, and how displacement-based and force-based beam-column element models can affect curvature results.

2.1 Discretization

In the tool, discretization of the brace for numerical analysis is comprised of three different levels, including: separating the brace into smaller sub-elements, separating those sub-elements into integration points, and separating the section at each integration point into fibers. In the learning tool, the user can manipulate each of these factors to determine how numerical results will be affected.

Figure 1 shows an example discretization of the brace. As shown in Figure 1, the brace has been separated into two sub-elements with five integration points per sub-element. The section at each integration point is a wide flange that has been separated into multiple fibers. The stress and strain is then monitored over time at the midpoint of each fiber.

The highest level of discretization includes separating the brace length into any number of sub-elements. At a minimum, two sub-elements are needed to enable symmetric buckling of the brace. This number can be increased, resulting in more sub-elements, nodes, and degrees of freedom. Increasing the number of sub-elements results in finer discretization of the brace element and potentially increased accuracy compared to the experimental results. However, the analysis also becomes more "computationally expensive" with increasing number of sub-elements (Simpson, 2018), i.e., using more sub-elements results in more degrees of freedom and more processing time for analysis, which might not be worth the increased accuracy in every situation.

The second level of discretization divides each sub-element with integration points. These points are used for numerical integration across the element length, e.g., Gauss-Lobatto quadrature assigns weights to points located at specific locations. The weight and value at the points can then be summed to numerically integrate across the element length. These points are typically located between the values -1 and 1 and are distributed with the respective weight coefficients as shown in Table 1. The analysis in the learning tool adjusts the values of -1 and 1 so that the locations are distributed between the end nodes of each sub-element.

The final level of discretization separates the section at each integration point into smaller fibers. The stress and strain are monitored at each fiber to determine the behavior of the section. Integration across the section, e.g., using the midpoint rule, results in the section deformation and force response at the integration points.

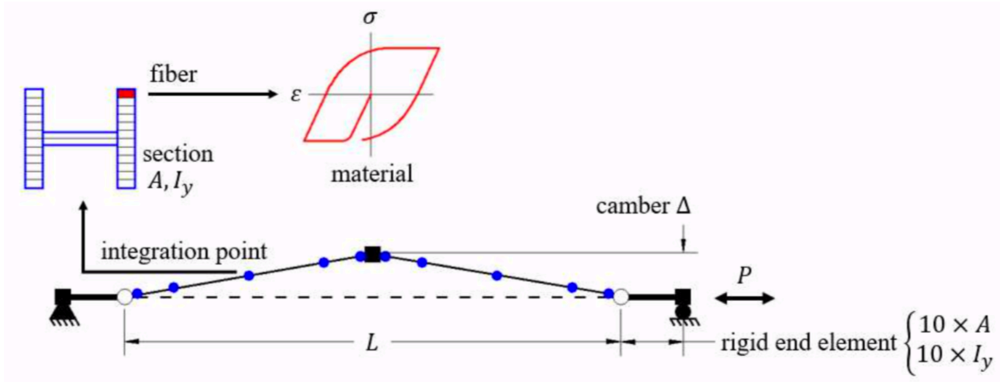

Figure 1: Brace Discretization in Brace Frame Learning Tool (Simpson, 2018)

Table 1: Gaussian Quadrature Nodes and Coefficients (Yew, 2011)

Gauss–Lobatto nodes and coefficients

n	x_1, roots of $P'_{n-1}(x)$, x_n	Coefficients c_i
2	-1	1
	1	1
3	-1	1/3
	0	4/3
	1	1/3
4	-1	1/6
	$-1/\sqrt{5}$	5/6
	$1/\sqrt{5}$	5/6
	1	1/6
5	-1	1/10
	$-\sqrt{3/7}$	49/90
	0	32/45
	$\sqrt{3/7}$	49/90
	1	1/10

2.2 Calculating Curvature

It is important to understand the meaning of curvature, i.e., how to calculate it and how numerical curvature was found in the numerical model. Curvature is defined as “the rate of change of the angle through which the tangent to a curve turns in moving along the curve and which for a circle is equal to the reciprocal of the radius” (Popov, Nagarajan, & Lu, 1976). In engineering applications, curvature, κ, is equal to the reciprocal of the radius of curvature, ρ, or $\kappa = \frac{1}{\rho}$; see Figure 2 for reference of the variable ρ.

In structural engineering, calculations of curvature typically assume that plane sections remain plane after deformations have occurred. For example, curvature derived from beam deformations often assumes that the section lines remain straight after bending has occurred. This concept is illustrated in Figure 2. Figure 2, which presents a beam separated by lines representing different sections, shows that even though deflection has occurred from the moment acting on a beam, grid lines A’B’, D’C’, etc. remain straight and in line with the direction of the radius of curvature as a result of the plane sections remain plane assumption. Note that this assumption neglects any shear deformation that occurs in a beam, which is reasonable if the shear deformation is small compared to flexural deformation.

The curvature equation, shown below, is derived from the variables shown in Figure 2:

$$\kappa = -\frac{\varepsilon}{y} \tag{1}$$

This equation indicates that the curvature depends on the strain, ε, of a point and its distance from the neutral axis, y. If the element response is elastic under pure bending moment, M, the stress, σ, at a point in the section is defined by:

$$\sigma = \frac{My}{I} \tag{2}$$

Using the relation, $\sigma = E\varepsilon$, curvature can also be represented in terms of moment by:

$$\kappa = -\frac{M}{EI} \tag{3}$$

This final equation is especially useful because it illustrates that a graph of curvature along an element reflects the shape of the bending moment diagram, so long as the element exhibits linear elastic behavior.

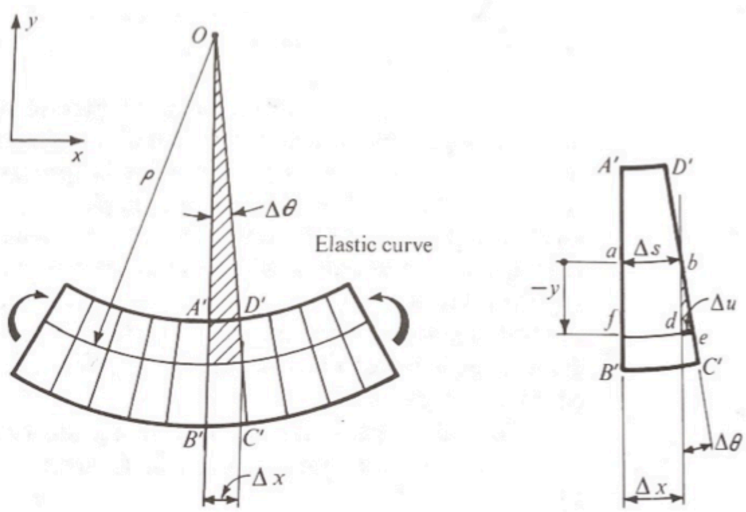

Figure 2: Beam Deflection Values for Curvature (Popov, Nagarajan, & Lu, 1976)

2.3 Force-Based vs. Displacement-Based Beam-Column Element Curvature

Understanding the differences between force- and displacement-based beam-column element models and their effects on curvature is necessary to interpret the curvature response obtained from the learning tool.

A displacement-based beam-column element modeling approach follows standard finite element procedures. Traditionally, these procedures approximate the deformation response by assuming axial strain is constant and the curvature is linear along the element length (Neuenhofer & Filippou, 1997). This method does not enforce equilibrium at the nodes, resulting in a discontinuous curvature response. As such, imposing constant axial deformation and linear curvature can result in poor approximation of the curvature response in the presence of nonlinear, inelastic behavior (Scott, 2019). Figure 3 illustrates the numerical curvature using displacement-based beam-column elements versus the exact curvature. Using four sub-elements, the exact curvature differs from the numerical approximation. Since the curvature is linear, using more sub-elements would result in a better approximation of nonlinear curvature response.

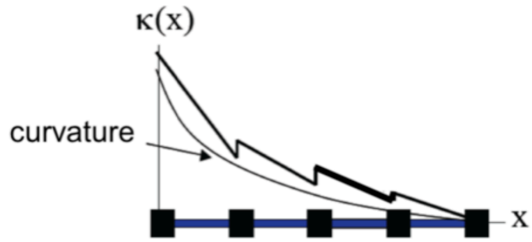

Figure 3: Idealized Curvature Values vs. Exact Curvature Using the Displacement-based Method (Terzic, 2011)

A force-based beam-column element modeling approach enforces equilibrium between the element and section forces, resulting in continuous curvature response along the element length (Neuenhofer & Filippou, 1997). This method iterates upon the axial strain and curvature at the section level until equilibrium is reached between the element and section forces. Use of force-based elements generally improves global and local response without the mesh refinement needed for the displacement-based method (Scott, 2019), resulting in better approximations of curvature for less sub-elements, as shown in Figure 4. This accuracy can come with more computational effort depending on the number of iterations needed to achieve the equilibrium requirement (Terzic, 2011).

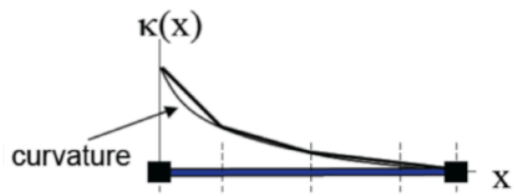

Figure 4: Idealized Curvature Values vs. Exact Curvature Using the Force-based Method (Terzic, 2011)

2.4 Curvature in the Program Model

Elements in the learning tool are distributed plasticity elements that assume plane sections remain plane. Using different element types, number of elements, and number of integration points in analysis will affect the brace curvature determined by the program. Deformations calculated within the program result from the sum of weighted curvature values at each integration point, e.g., using Gauss-Lobatto quadrature. Since equilibrium is satisfied using force-based beam-column elements, for the same deformation value, the curvature response will change depending on the number of integration points; i.e., curvature is not objective with the use of force-based beam-column elements. This is because the location, distance between points, and weights of the integration points change as integration points are added or removed, but the weighted sum used to calculate the deformation will remain unchanged because of the equilibrium requirement (Simpson, 2018). Even though moment, similar to deformation, is objective with regards to the number of integration points, the curvature response non-objectively depends on the number of integration points.

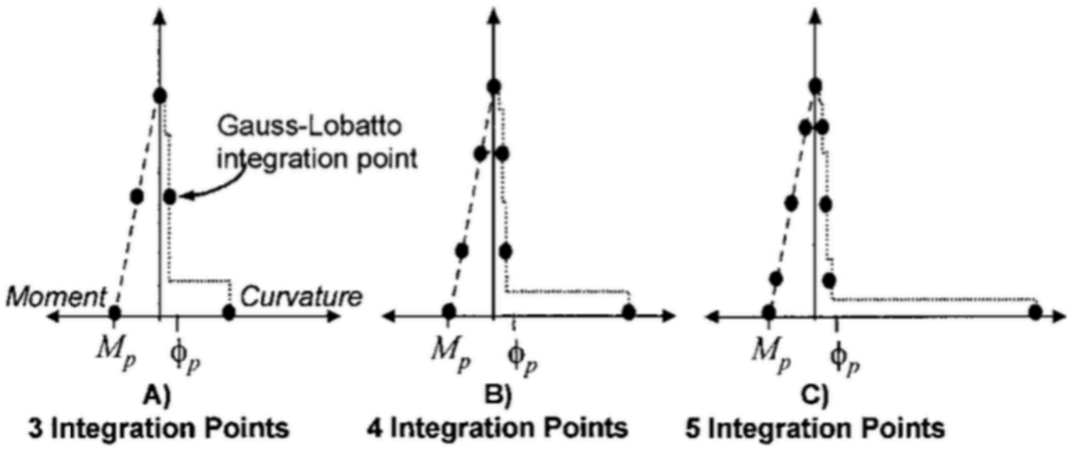

Figure 5: Moment and Curvature Values with Various Integration Points (Coleman & Spacone, 2001)

The change in curvature values based on the number of integration points used per element is illustrated in Figure 5, which shows a cantilever defined by a single element (this can be thought of as half the brace in the learning tool). Assuming perfectly plastic material behavior, the bending moment at the integration point that first exhibits inelastic behavior cannot increase beyond M_p. Without strain hardening, the inelastic response cannot spread to adjacent integration points, and the inelastic curvature localizes at the integration point where inelastic response was initiated (Coleman & Spacone, 2001). Increasing the number of integration points requires that the curvature value at that integration point increase to result in the same deformation and moment response. Although this does not affect the resulting moment, this localization results in changes to the curvature for the same deformation response.

The response of the cantilever is expected to be similar to the response of the brace in the learning tool. As the brace buckles, the integration point near the middle of the element length will be near its flexural strength. If the plasticity cannot spread to neighboring integration points, the strain and curvature near the point of buckling will become localized. Adding a curvature element to the existing program will be a way to further investigate this phenomenon.

3 Methodology

The process of adding a curvature diagram to the existing program involved many steps. It was important to learn how to code as well as understand the existing program format. After that, curvature could be retrieved and plotted within its own diagram. This required multiple attempts to successfully retrieve data, and each of these attempts involved efforts to debug and troubleshoot the additional code.

3.1 Existing Program

The braced-frame modeling program was already complete with many different features prior to the additions made for the purposes of this study. To ensure that the additions would not interfere with any existing functions, understanding this initial program was essential before making any additions to the code. It was also a useful way to become familiar with the current code and its structure.

In its original state, the software showed displays for how the displaced shape, axial force, and moment within a brace element would change over time. There was also a graph that showed the overall calculated response for axial deformation and force over time, which is compared to existing experimental values. Fatigue and strain hardening are also included in this program to show how the material behavior affects response.

These factors can be manipulated in many different ways. Some existing manipulations include changing the length of the braced element, modeling the brace using force-based or displacement-based beam-column elements, material or connection types, shape and severity of a camber, and shape of the brace section. A major manipulation within the existing program involves changing the number and distribution of sub-elements, as well as the number of integration points used in calculation. This is a key factor in graphing/analyzing the response because it changes how values are being calculated and visualized for the brace. Being able to change the number and distribution of sub-elements also changes the output graphs by adding nodes. It also gives the user the ability to distribute elements evenly across the brace or concentrate the majority of the elements at the center of the brace member. The application's input and output interfaces are illustrated in Figures 6 and 7 respectively.

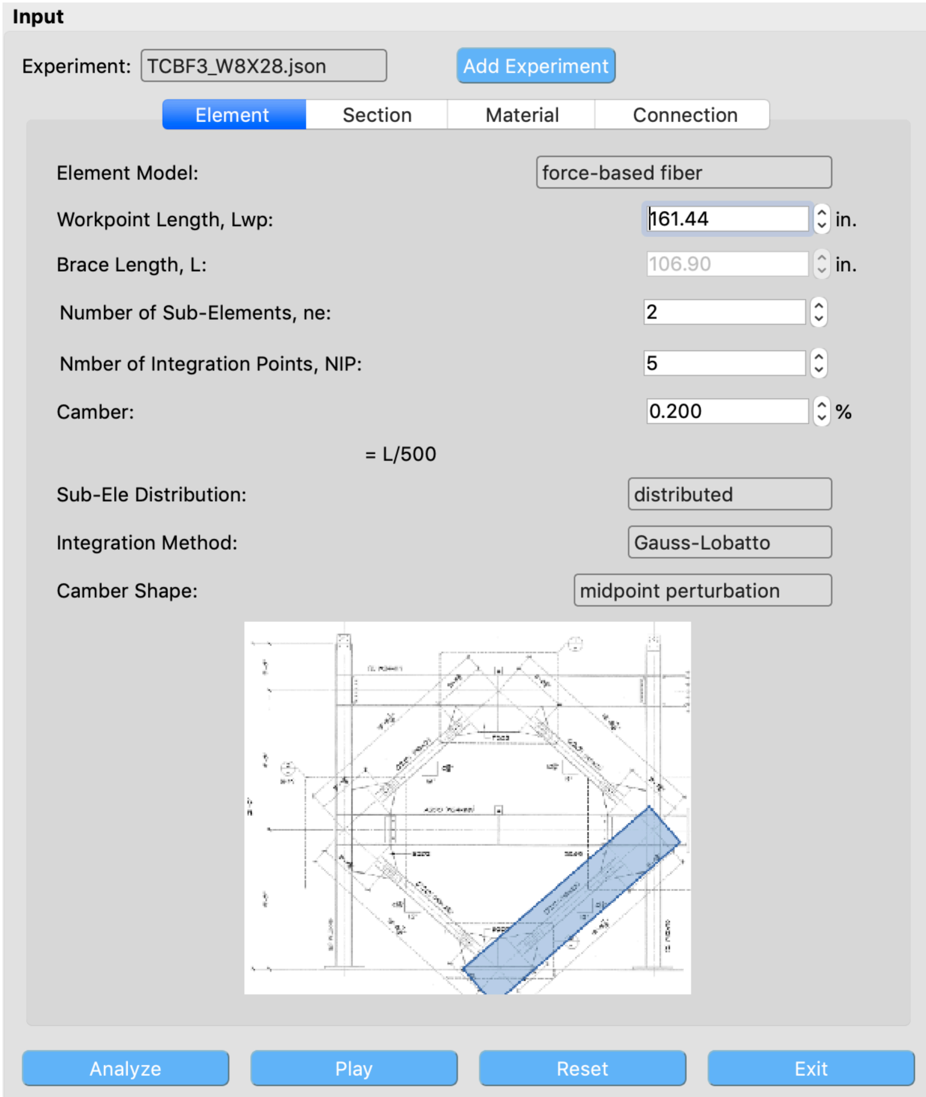

Figure 6: Application Input Interface

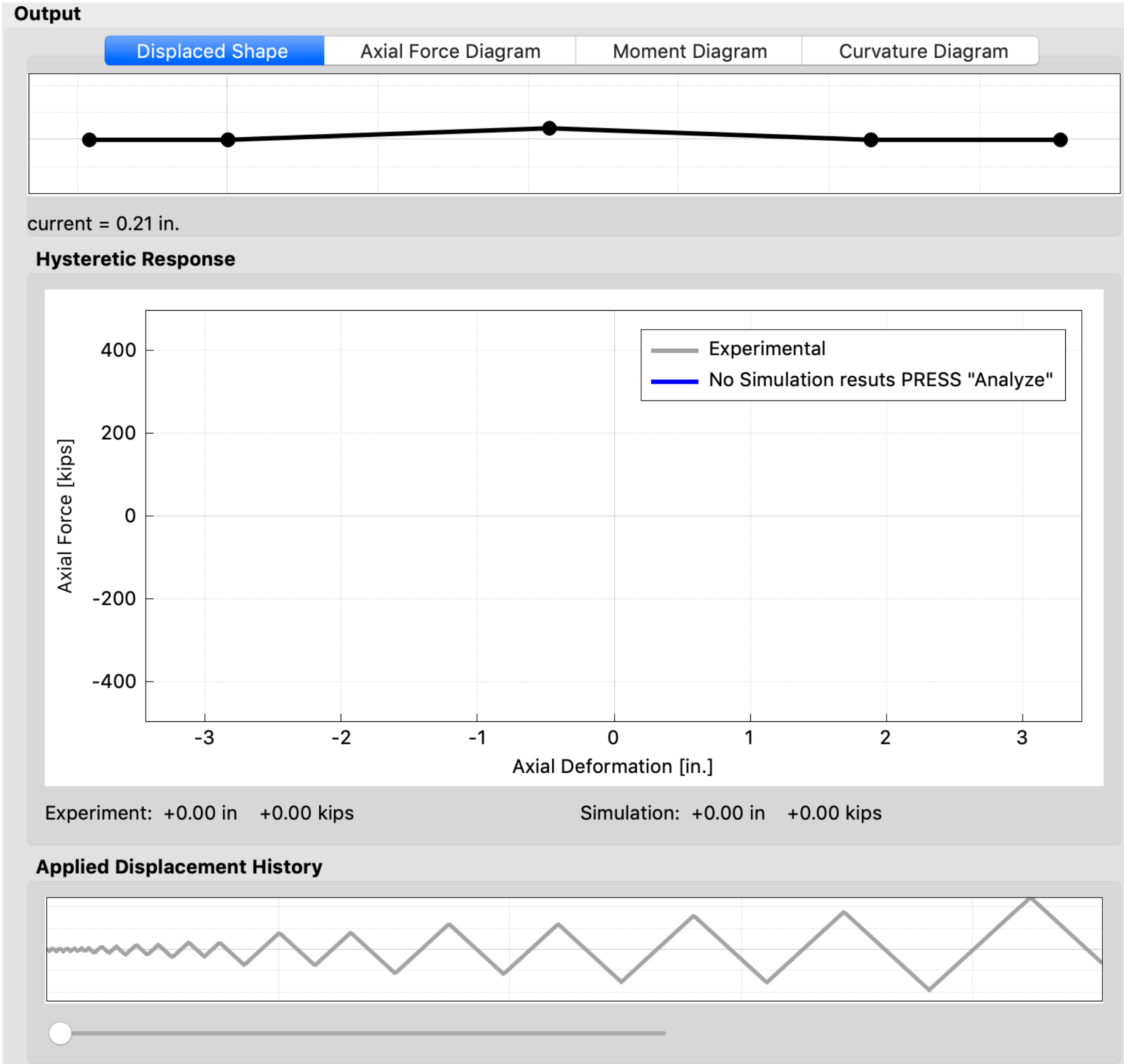

Figure 7: Application Output Interface

3.2 Learning to Program

Before any additions could be made to the braced-frame modeling program, knowledge of how to program using the C++ language and the QtCreator GUI application was needed. The first step in this learning process involved taking an online Udemy course that covered C++ concepts for beginners. This was a useful way to develop an understanding of the format used in C++ coding, which allowed for a deeper understanding of the current code within the program.

After learning the basics of the C++ language, the next step was to watch YouTube videos on the basics of QtCreator. Knowledge of using this application was essential to making changes to the existing program because the existing program was developed in QtCreator. Learning basic concepts for the C++ language and QtCreator application was not enough to become proficient in their use, but it was enough to allow for an understanding of the program and how it was constructed. These basic concepts were also essential to coding any of the additions made to the program as well as the troubleshooting involved in making these additions work. Another benefit of learning these concepts was that any complex functions could be broken down into basic concepts and understood with supplementary information from the Qt help function or online research.

3.3 Additions made to Program

The main goal of this project was to add a new graphical display showing curvature to the existing program. Changing the code to add in a curvature diagram involved many different steps, and there were multiple iterations involving troubleshooting.

The first step involved making an empty graph that would show all integration point nodes for each element. This was different from the existing program because the current software only showed nodes at the end points of each sub-element. Accurately showing how curvature was being calculated for each element required nodes at each integration point due to the fact that calculated curvature does not increase uniformly across an element as shown previously in Figure 5. Figure 8 shows the difference between the existing graph of moment showing sub-element nodes only (shown above) compared to the added graph of curvature showing the integration points for each sub-element (shown below) for a brace that has two elements and five integration points.

Figure 8: Moment Diagram Graph vs. Curvature Diagram Graph

After the graph had been created, the next step involved recovering the curvature values that were already being calculated by the program. These values were necessary to calculate the values shown at the end nodes for each sub-element. Because the values were previously calculated and embedded within different functions in the code, the primary purpose of this step was to find and recover these values to graph the curvature.

Once these values had been recovered, the final addition necessary within the code was to merge the graph and the curvature values to complete the diagram for curvature. This step initially involved attaching the curvature value at each integration point with the integration points acting as x-values and the curvature acting as y-values. The largest difficulty with this particular addition was making sure that the curvature values were attached to the correct integration points. To make this happen, some trial and error was needed to make sure that the values were added to the correct points along each sub-element and the duplicate points at the sub-element end nodes were removed. Once the curvature graph was functioning correctly for the initial requirements, the next step was to debug and troubleshoot any issues that occurred as the number of elements and/or integration points were modified.

3.4 Debugging and Troubleshooting

Debugging the additions made to the program involved correcting many little errors and issues, but there were two main issues that needed attention. The first issue was that even though the curvature was being graphed correctly at each point, there was an additional point being shown at the far left of the graph with zero value attached. This created an extra line that ran across the entire brace showing zero curvature. Removing this added value initially seemed simple because it only involved removing an extra node, but in reality it required some effort. The code couldn't initialize without the extra node due to the fact that it would read that there was a negative amount of nodes when no nodes actually existed. This was avoided by adding an absolute value to the number of nodes read by the program. Even though this wouldn't change anything shown in the final results other than removing the extra line shown, it would allow the program to initialize without the issues that were causing it to crash.

The other main issue that required some troubleshooting involved a software crash that would occur when the sub-elements were concentrated after six or more sub-elements were used. Initially it seemed like this issue was occurring due to the newest additions made to the code, but upon further investigation it was obvious that this was an issue that was already occurring in the original program. To avoid this issue, changes were made to the definitions of nodes that fixed the problem for the original program and the new additions that were made. This was necessary to prevent the crash that was occurring due to the fact that even though six or seven sub-elements were used for calculation, only four or five were displayed respectively for the concentrated requirement. Showing a reduced number of sub-elements was the desired result, but initially the program was not accounting for this reduction of sub-elements. Fixing this problem allowed the program to correctly plot results without crashing because it was no longer storing more data than allowable for the number of nodes shown. After these two main issues and other smaller issues had been debugged, curvature could be analyzed with the learning tool software.

4 Results

Changing the numerical modeling inputs changes the curvature diagram. Not all the possible changes to these inputs are addressed by this study. Focus was placed on key inputs affecting curvature, including changing the number of elements or integration points, strain hardening values, and the use of concentrated versus distributed sub-elements and displacement-based versus force-based beam-column elements. These changes were performed individually to see how the results were affected compared to a control test.

4.1 Parametric Study

To compare how each modeling change affects the results, it was necessary to create a control test. This control test used most of the default inputs in the learning tool, with the exception of the material inputs and camber shape. The original material, steel01 without strain hardening, does not converge. Since the purpose of this study is to observe the effects of inelastic behavior on brace response, a different material, steel02 with strain hardening, was specified instead. Remaining defaults included the following: two equally distributed sub-elements, five integration points, and a force-based element model. For all tests a sinusoidal camber shape was used, instead of the original midpoint perturbation setting. This was done to prevent concentrating the initial camber at the center of the brace.

Changes to the curvature were studied by changing one variable at a time. Key variables analyzed include number of sub-elements/integration points, strain hardening, distribution type, and modeling method. The values used for each test performed are shown in Table 2. Resulting values for peak displacement, axial force at peak displacement, tensile/compressive moment, and tensile/compressive curvature in each tested case are illustrated in Table 3.

Table 2: Parameters for Each Test Case

Case	Number of Sub-elements	Number of Integration Points	Strain Hardening Alpha Value	Concentrated v. Distributed	Force- v. Displacement-based
Control Test	2	5	0.003	Distributed	Force-based
20 Sub-elements	20	5	0.003	Distributed	Force-based
10 Integration Points	2	10	0.003	Distributed	Force-based
Strain Hardening x1/10	2	5	0.0003	Distributed	Force-based
Strain Hardening x10	2	5	0.03	Distributed	Force-based
Concentrated Distribution	6	5	0.003	Concentrated	Force-based
Displacement-based Method (2 sub-elements)	2	5	0.003	Distributed	Displacement-based
Displacement-based Method (20 sub-elements)	20	5	0.003	Distributed	Displacement-based

Table 3: Displacement, Axial Force, Moment, and Curvature for Each Test Case

Cases	Peak Displacement	Axial Force at Peak	Moment at Tensile Point	Moment at Compressive Point	Curvature at Tensile Point	Curvature at Compressive Point
Control Test	11.17 in	-54.98	-76.06	627.89	-0.0141	-0.0525
20 Sub-elements	10.96 in	-53.18	-121.00	584.67	-0.0033	-0.0309
10 Integration Points	11.21 in	-54.08	-92.18	619.63	-0.0107	-0.0477
Strain Hardening x1/10	11.41 in	-47.31	-104.21	552.28	-0.0325	-0.0861
Strain Hardening x10	10.58 in	-79.82	-82.28	861.29	0.0024	-0.0190
Concentrated Distribution	11.12 in	-53.51	-125.98	603.69	-0.0054	-0.0396
Displacement-based Method (2 sub-elements)	11.23 in	-63.60	-307.61	730.20	0.0058	-0.0146
Displacement-based Method (20 sub-elements)	10.97 in	-53.20	-123.67	585.03	-0.0061	-0.0308

4.2 Control Test Results

For the purpose of comparison, images used for results were captured at the same two points along the applied displacement history used to load the brace element. These points were determined somewhat arbitrarily, but they provide a positive (tensile axial deformation) and negative (compressive axial deformation) point within the displacement history that can be used for comparison. In subsequent figures, the compressive point (Figure 9) is shown on the top plot and the tensile point (Figure 10) is shown on the bottom plot.

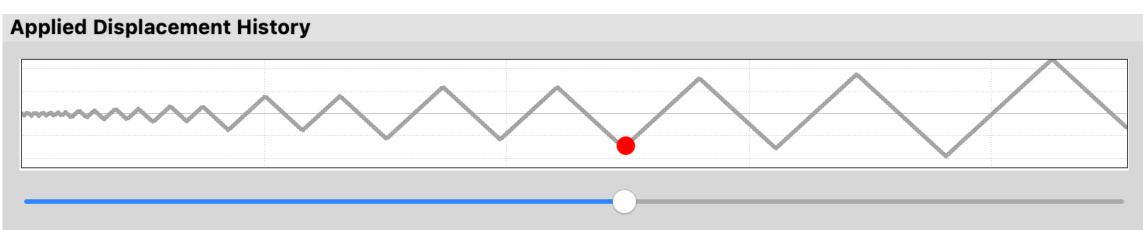

Figure 9: Chosen Compressive Point in Displacement History Graph Used for Analysis

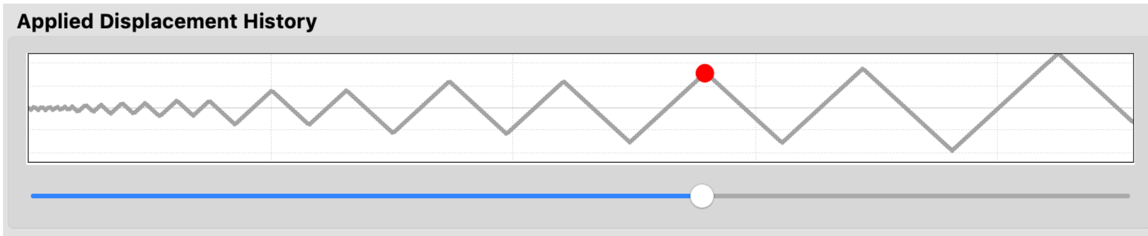

Figure 10: Chosen Tensile Point in Displacement History Graph Used for Analysis

The diagrams for displaced shape (Figure 11) and moment (Figure 12) at each chosen point for the control test are shown. The shape of the moment diagrams and displaced shape is fairly similar regardless of inputs as long as two sub-elements are used. This is due to the fact that both of these plots only have two linear sub-elements meeting at the center of the brace, and therefore both moment diagrams and displaced shape will not be attached unless they deviate substantially from the control plots. The displaced shape and moment plots follow the same convention, with the compressive point shown as the top diagram and the tensile point shown as the bottom diagram. The curvature diagram for the control test is shown in Figure 13.

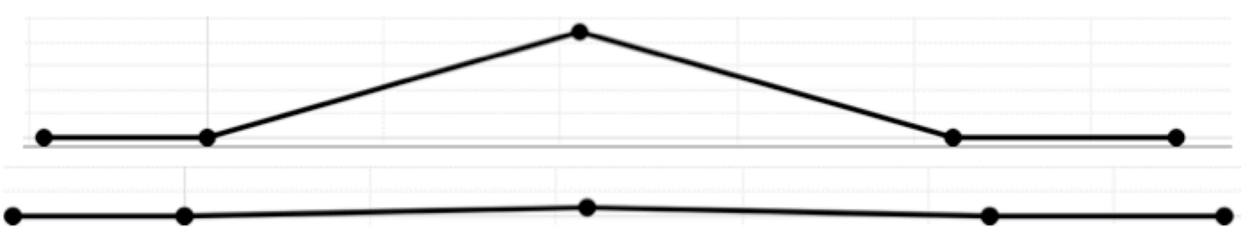

Figure 11: Control Test Displaced Shape Diagrams

Figure 12: Control Test Moment Diagrams

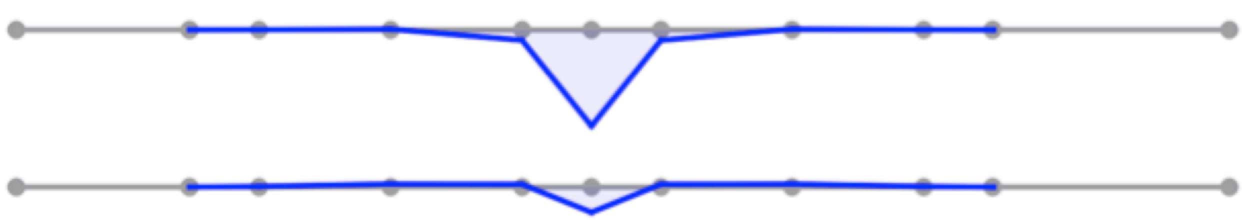

Figure 13: Control Test Curvature Diagrams

Figure 13 will serve as a baseline to compare how any variations made to inputs will change the results that are shown within the curvature diagram. The top diagram shows the curvature at the designated displacement history low point when the brace was in compression and the bottom diagram shows the designated high point when the brace was in tension. Note that all curvature values have been multiplied by a factor of 100 to display results. This was done to make the values large enough so that the effects were distinctive, since the original diagrams looked like flat lines.

4.3 Number of sub-elements

The number of sub-elements was changed to observe how the curvature diagram changed, see Figure 14. Five integration points and force-based beam-column elements were used for each sub-element. In compression, compared to the control test, the overall shape of the curvature field remained roughly the same, regardless of the number of integration points. Although the shape is similar, the curvature value is reduced closer to zero as shown in Table 3. Changes to this response are the result of a better approximation of a nonlinear curve due to mesh refinement. Curvature is also reduced in this case because there are points closer to the center of the brace where inelastic behavior originates. With strain hardening included in the steel02 material, the curvature is able to spread to adjacent points if they are close enough to the center of the brace, and in this case the spread of that curvature reduces the value reported overall.

In contrast, curvature results differed significantly when the brace was in tension. With increasing number of sub-elements, the curvature diagram exhibits a "kink". This is due to the residual deformation resulting from the brace buckling. This behavior is not well represented in the control test because there are only two sub-elements and not enough integration points located near the buckled region to adequately represent this response.

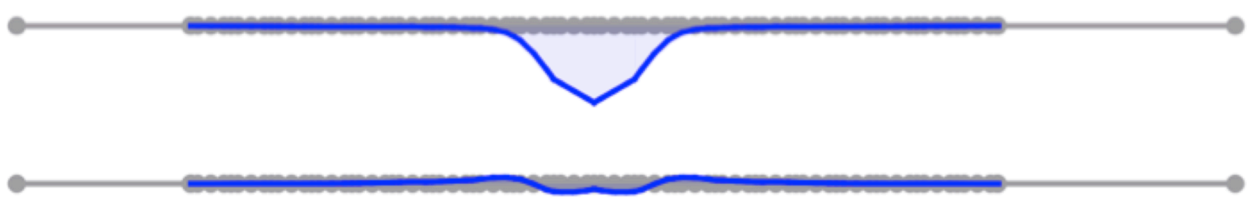

Figure 14: Increased Number of Sub-Elements Curvature Diagrams

The effects of increasing the number of sub-elements to 20 on displaced shape and moment are shown in Figures 15 and 16 respectively. It can be seen that all plots are no longer linear from the end to the center point. Table 3 shows that the peak displacement and compressive moment values are reduced. These are all due to mesh refinement, similar to the reduced curvature values in this case.

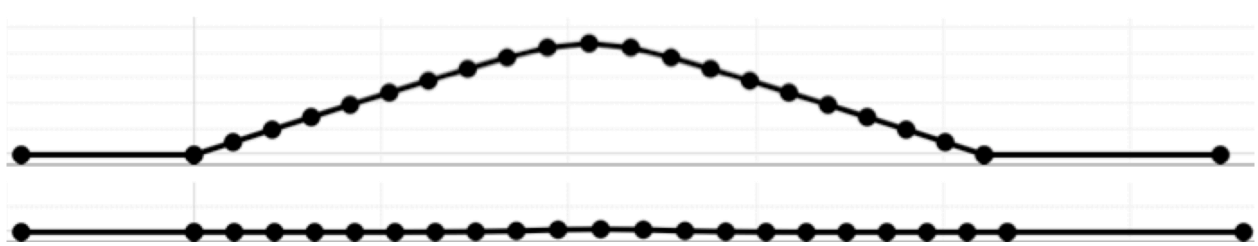

Figure 15: Increased Number of Sub-Elements Displaced Shape Diagrams

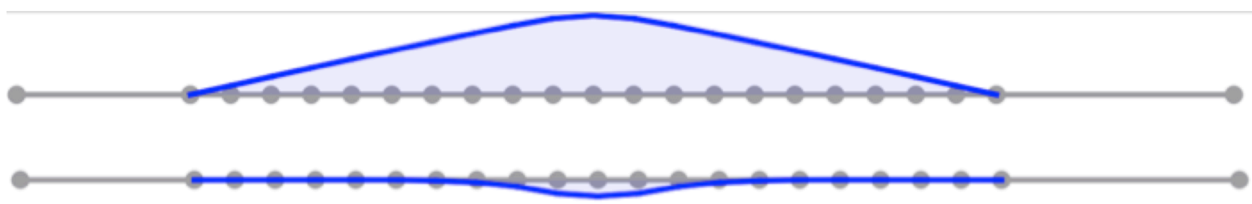

Figure 16: Increased Number of Sub-Elements Moment Diagrams

4.4 Number of Integration Points

The number of integration points was varied using only two sub-elements. The control test using five integration points was compared to a model using ten integration points, the maximum number allowed within the program. Curvature diagrams resulting from ten integration points are shown in Figure 17. In compression, curvature results were similar to those of the control test. Small discrepancies between these plots are from differences in the mesh refinement. Similar to the increased number of sub-elements, the curvature magnitude is reduced compared to the test values. Since increasing the integration points and number of sub-elements leads to a finer discretization, this makes sense. In tension, increasing the number of integration points has a similar effect on the curvature diagram shape as increasing the number of sub-elements. The moment and peak displaced shape in this test remained similar to the control results since only two sub-elements were used.

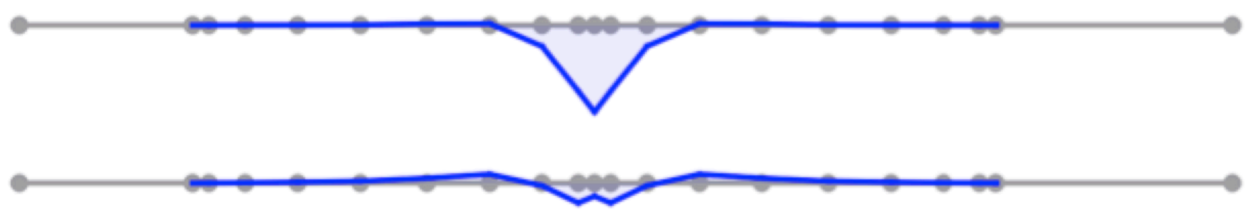

Figure 17: Increased Number of Integration Points Curvature Diagrams

4.5 Strain Hardening

Using the control test model, the effects of increasing and decreasing strain hardening were investigated. Kinematic hardening was reduced and increased by a factor of ten, see Figures 18 and 19 respectively.

Decreasing the kinematic hardening by a factor of 10 results in similar plots to the control. However, the magnitude values reported in Table 3 are fairly different. The peak-displacement value increased, moment decreased, and both curvature values are larger (farther from zero) than the original test. This is due to the decreased strength of the brace after buckling caused by a reduction in strain hardening, which also makes it less likely for curvature to spread to adjacent points near the point where inelastic behavior originates similar to the phenomenon shown in Figure 5.

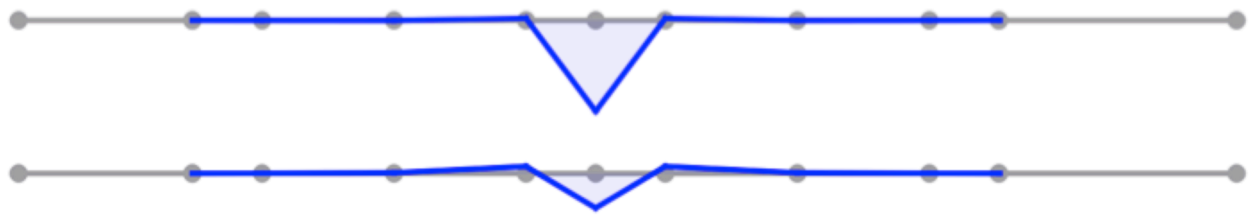

Figure 18: Strain Hardening Reduced Alpha Value

The effects of increasing the original kinematic hardening by a factor of ten are shown in Figure 19. This had the opposite effects of the reduced test. The peak-displacement value decreased, moment increased, and both curvature values are closer to zero as a result of the increased strength of the brace after buckling. This is a similar effect as the increased number sub-elements and integration points tests. Figure 19 shows a similar shape to the control, but curvature is more distributed along the brace. The additional distribution is a result of curvature being able to spread to adjacent integration points more easily with increased strength. There is also a positive value for curvature in the tensile portion of this test due to the absolute maximum value no longer being at the center, or points adjacent to the center, of the brace.

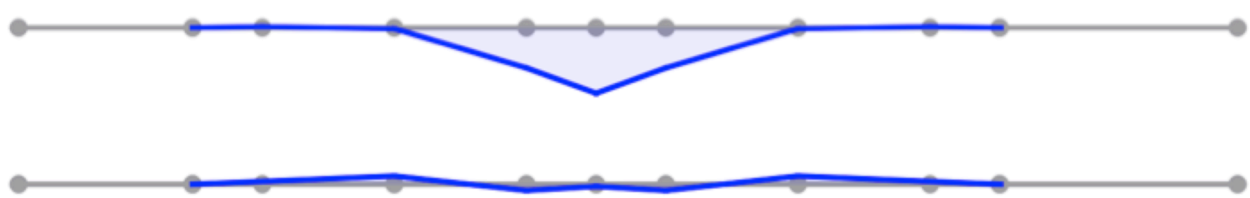

Figure 19: Strain Hardening Increased Alpha Value

4.6 Concentrated vs. Distributed Elements

The difference between having uniformly distributed sub-elements versus smaller sub-elements concentrated near the brace midpoint was also studied. Two concentrated sub-elements of length 1/6 of the brace length were used near the midpoint of the brace length. Results from using the concentrated distribution of elements are shown in Figure 20. Concentrating the sub-elements near the brace location results in similar curvature response to increasing the number of integration points or sub-elements. The use of concentrated sub-elements results in a larger number of integration points near the buckling location, resulting in greater resolution of the curvature near the buckling location and a reduced curvature value compared to the control test.

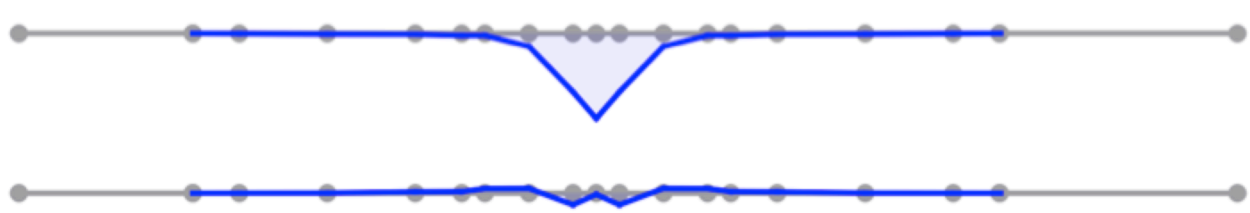

Figure 20: Concentrated Sub-Elements Curvature Diagrams

4.7 Displacement-Based vs. Force-Based Elements

Comparisons of curvature were also made using force-based versus displacement-based beam-column element models. As previously mentioned, the displacement-based formulation assumes linear curvature across each sub-element. Figure 21 shows the curvature diagrams for a displacement-based beam-column modeling assumption. These curvature diagrams are significantly different from those of the control tests, and the results reported in Table 3 are unlike any other test. This is due to the inaccuracy of only using two sub-elements in a displacement-based model. Mesh refinement would resolve these issues. Importantly, non-negligible curvature is observed near the brace ends. The discontinuity in the curvature diagram occurs because of the assumption of constant axial strain and linear curvature. A similar discontinuity illustrating this phenomenon was shown in Figure 3.

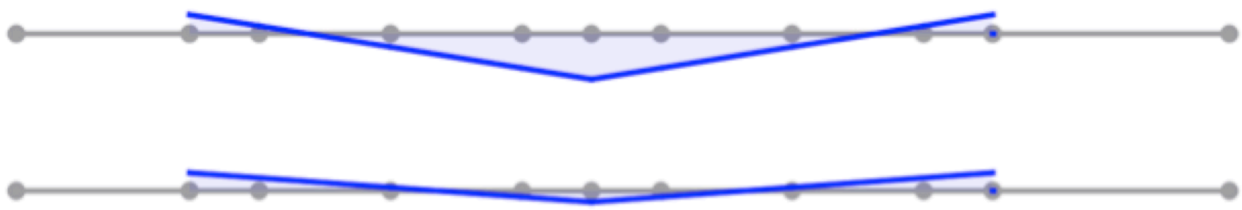

Figure 21: Displacement-Based Model Curvature Diagrams

Since the original displacement-based test was different from all other tests due to the inaccuracy of only including two sub-elements, mesh refinement is needed. Increasing the number of sub-elements in a displacement-based beam-column element results in a curvature response that approaches the exact curvature distribution. The effects of adding more sub-elements to a displacement-based model are shown in Figure 22, where twenty sub-elements are used instead of two. Even though the curvature still has discontinuities and is strictly linear, the resemblance to both the curvature diagram of the control test and the increased number of sub-elements test using force-based beam-column elements is noticeable.

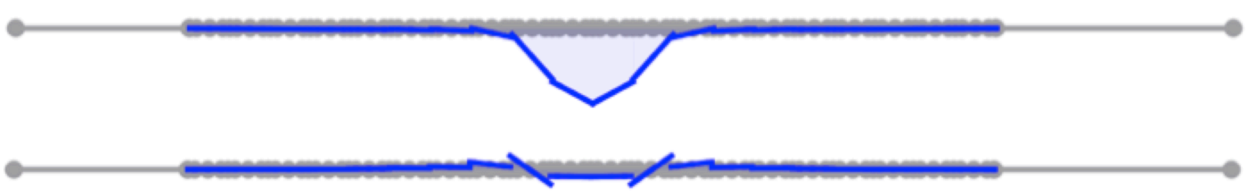

Figure 22: Displacement-Based Curvature with Increased Sub-elements

It is worth noting that, while it is not shown, increasing the number of sub-elements brought the global hysteretic response closer to experimental values. Also, the values for this test are almost identical to the increased number of sub-elements test from Table 3, and the plots are very similar to those from Figure 14 other than the assumptions for linear curvature across the sub-element. The moment diagram and displaced shape were also changed as the number of sub-elements increased, and these plots looked identical to those from Figures 15 and 16. This shows that mesh refinement for a displacement-based model does in fact bring the curvature values, and therefore other calculated values, closer to the exact solution.

5 Discussion

As illustrated by the previous section, curvature depends on the modeling assumptions. Visualizing the curvature response allows the user to gain insight on how modeling assumptions affect numerical results. Future additions to the program other than curvature would result in even greater insights.

5.1 Results Analysis

Although the overall hysteretic response representing global behavior compared well to the experimental data, the curvature diagrams significantly depend on the numerical modeling inputs.

For all cases considered, the location of inelastic behavior near the buckling location resulted in the largest curvature value plotted as it exhibited the most significant inelastic response. Without significant strain hardening, inelastic behavior near the buckled location cannot easily spread to adjacent integration points (Coleman & Spacone, 2001). The tests for increasing the number of sub-elements or integration points, increasing strain hardening, and using a concentrated distribution of sub-elements showed that it is still possible for curvature to spread to adjacent integration points with some strain hardening included in the material. This is confirmed by the reduced compressive curvature values, reported in Table 3, which show that the increase of curvature values near the center of the brace cause curvature at the center of the brace to decrease to maintain equilibrium and the same weighted sum of values.

Changing the number of sub-elements, number of integration points, and sub-element distribution affected the results in similar ways. For the point of compressive deformations in the brace, shown on the top portion of each figure, the curvature shapes were fairly similar to the control test. Slight changes occurred with finer meshes as the model gained higher resolution. In tension, the brace was unable to entirely straighten resulting in a “kink” in the curvature diagram that is illustrated in the curvature response. This occurs because of the presence of residual deformations from brace buckling. This effect on curvature was only present when a finer mesh was utilized. Changing the camber would also affect the shape of this “kink”.

Almost identical values resulted from the force-based and displacement-based models using twenty sub-elements. These values, shown again in Table 4, were similar except for curvature measured at the tensile point. The curvature at the tensile point is fairly different between these model types due to the linear curvature assumption made for displacement-based beam-column elements. Displacement-based beam-column elements cannot capture this kink unless a finer mesh is utilized.

Table 4: Reiterated Values for Increased Sub-Elements and Displacement-Based Method

Cases	Peak out-of-plane displacement	Moment at Tensile Point	Moment at Compressive Point	Curvature at Tensile Point	Curvature at Compressive Point
Increased Sub-elements	10.96 in	-121.00	584.67	-0.33	-3.09
Displacement-based Method Modified	10.97 in	-123.67	585.03	-0.61	-3.08

5.2 Importance of Program and Study

Although the initial program was useful for many reasons, adding a curvature diagram as a result of this study was still important to allow the user to see how much inputs could change curvature overall. This new diagram adds to the educational aspect of the first version of the learning tool, and even though it wasn't necessary originally it is still a powerful tool for users.

The existing learning tool (without curvature response) is important for many reasons. It allows the user to manipulate the modeling of an individual brace element to see how the overall analysis is affected, and it provides a comparative diagram to show both an experimental and simulated hysteretic response of the brace as it cycles between loading an unloading. Not only is this useful for learning from and experimenting with the manipulation of results through a simulated model, but it can be implemented to show how individual braces should be simulated in larger simulations of an entire structure. The introductory page for the existing tool discusses the importance of determining the response in large buildings that experience high magnitude seismic events through simulations, and the learning tool helps users determine how to model braces in large scale simulations (Simpson, McKenna, & Gardner, 2018). This shows that the program is not only useful to learn about how changing inputs, model types, and assumptions can have an effect on results, but it is also a way to learn what the optimal way to model braced elements is within larger programs and simulations.

Adding a curvature diagram to the program was not an essential step initially because it did not affect the global behavior results or the ability to show users how inputs could change the hysteretic response. Users could still use the tool to learn how brace elements should be modeled without visualizing the curvature response. However, although originally not a feature within the first version of the program, showing curvature diagrams with the results is still important. The largest importance of allowing the user to see curvature is to show how much it can be changed by the input values. For example, even though the hysteretic response might only change slightly and become a little more accurate with an increased number of sub-elements, the curvature diagram can change dramatically as shown by the "kink" in Figure 14.

Allowing the user to see the effects on curvature can also help add to the educational aspect of this tool. Displacement-based and force-based modeling can be fairly complicated concepts when learning about all of the assumptions used and the pros and cons of each model type. The added curvature diagram helps to explain some of this, and the effects of the assumptions used become clearer. For example, the hysteretic results did change somewhat when changing from a force-based model to a displacement-based model, but the curvature really changed due to the linear curvature assumption used for the displacement-based method. This concept is already known, as shown by the difference in Figures 3 and 4, but the addition of the diagram within the learning tool helps the user visualize this concept in an actual simulation. While curvature might not have been vital to the original program to achieve its necessary goals, it is still an important addition that can help improve the information provided as a learning tool for users.

5.3 Future Additions to Program

Although the braced frame modeling tool is currently available and functional, there are still other additions that can be made to improve the overall user experience, similar to the addition of curvature. Some of the planned additions that will be made to the program over time are listed in a table within the learning tool's user manual, see Table 5 (Simpson, McKenna, & Gardner, 2018). The key additions listed in this table include: allowing the user to create sections, creating an image to show fiber discretization, allowing the user to define a moment-rotation response representing the connection details, and adding a setting for concrete-filled brace sections.

Each of these additions would be beneficial to the learning tool and its users in different ways. Creating a way for users to define sections for analysis would help to show how custom sections that aren't within the AISC database behave when subjected to a seismic event and buckling. Adding an image for fiber discretization will help to explain how the section is being divided at each integration point, similar to how showing curvature supplements the theories from strength of materials and numerical modeling. Including a moment-rotation response will allow users to manipulate connections and visualize their impact on response, which can be important when more specialized connection types are needed. Finally, adding a setting to use concrete-filled tubes will create another useful way for users to see how composite brace elements respond to a seismic event with loading and unloading. Although none of these additions are necessary for the functionality of the program and its usefulness as a learning tool, they would still be interesting additions that would improve the tool in the future.

Table 5: Learning Tool Existing and Future Desired Functions (Simpson, McKenna, & Gardner, 2018)

#	Description	Priority	Version
1	Ability to compare simulation results with experimental results for a number of different experimental datasets	**M**	**1.0**
2	Ability of user to modify the element type and discretization used for the numerical simulation	**M**	**1.0**
3	Ability of user to change the extent of the initial deformation, camber, and camber shape when using beam-column elements	**M**	**1.0**
4	Ability to change section type to include: Wide Flange HSS Round HSS and PIPE	**M**	**1.0**
5	Ability to model concrete-filled tube braces	**P**	**-**
6	Ability to define user-defined sections	**D**	**2.0**
7	Ability to change section discretization in the simulation model, i.e. number of fibers in in steel section	**M**	**1.0**
8	Ability to look at fiber discretization in an image	**D**	**2.0**
9	Ability to study effect of different materials and their material properties on the simulation response	**M**	**1.0**
10	Ability to modify connection details in the simulation mode, including end rotational constraints and connection length.	**M**	**1.0**
11	Ability to define a user-defined moment-rotation response representing the connection details	**D**	**2.0**
12	Ability to look at deformed shape of model as the analysis progresses	**M**	**1.0**
13	Ability to compare experimental and simulation measured response with respect to axial deformation and axial force	**M**	**1.0**
14	Ability to compare experimental and simulation measured responses with respect to curvature	**D**	**2.0**
15	Ability to view the moment diagram for the simulation model	**M**	**1.0**
16	Tool to provide a number of preloaded experiments, 2 at a minimum	**M**	**1.0**
17	Tool to allow user to store current configuration and reload it later	**M**	**1.0**
18	User able to save current settings and subsequently reload at a later date	**D**	**1.0**

M=Mandatory, **D**=Desirable, **O**=Optional, **P**=Possible Future

6 Conclusion

This project involved many different steps that required both patience and self-study to complete. The most important aspect of completing the additions to the program and analyzing results was to gain background knowledge on both coding and the existing program's design. Without an understanding of how to program it would have been impossible to make any additions. Without knowledge of the theory behind numerical analysis, discretization, and model types, the finalized curvature diagrams would have seemed nonsensical.

Overall, this project was successful in developing a curvature diagram addition for the existing braced frame modeling tool. There are no definitive values to show that these diagrams are accurate due to the complexity of the calculations required to confirm values for inelastic behavior along with buckling in the brace, but the diagrams do appear to be correct based on existing theories regarding curvature in program models. The "kink" in curvature at the center of the brace, even with small residual deformations, can only be observed with increasing number of integration points and sub-elements. Furthermore, the displacement-based diagram illustrated that using more sub-elements results in higher resolution and more accurate results.

The braced frame learning tool can continue to be improved by providing more ways for users to test and manipulate the modeling inputs and results. In its current state the tool provides many insights on how modeling affects numerical behavior. Curvature was not an essential part of this program when the original version was developed, but its addition provides another means for users to gain insights on how model inputs affect curvature. It also shows how discretization affects the curvature results. Above all else, the curvature diagram has improved the learning tool, and provides one more way for users to visualize concepts behind existing numerical models and methods.

7 References

Coleman, J., & Spacone, E. (2001). Localization Issues in Force-Based Frame Elements. *Journal of Structural Engineering , 127* (11), 1257-1265. doi: 10.1061/(asce)0733-9445(2001)127:11(1257)

Neuenhofer, A., & Filippou, F. C. (1997). Evaluation of Nonlinear Frame Finite-Element Models. *Journal of Structural Engineering , 123* (7), 958-966. doi: 10.1061/(asce)0733-9445(1997)123:7(958)

Popov, E. P., Nagarajan, S., & Lu, Z. A. (1976). *Mechanics of Materials.* Englewood Cliffs, NJ: Prentice-Hall.

Scott, M. (2019). *Displacement-Based Beam-Column Finite Elements.* Retrieved January 10, 2020, from Oregon State University Ecampus, CE 537 Lecture 10.

Scott, M. (2019). *Force-Based Beam-Column Finite Element.* Retrieved January 10, 2020, from Oregon State University Ecampus, CE 537 Lecture 11.

Simpson, B. (2018). *Design Development for Steel Strongback Braced Frames to Mitigate Concentrations of Damage.* Civil and Environmental Engineering. University of California, Berkeley.

Simpson, B., McKenna, F., & Gardner, M. (2018, September 28). *Braced Frame Modeling*. Retrieved January 21, 2020, from https://simcenter.designsafe-ci.org/learning-tools/braced-frame-modeling/

Terzic, V. (2011, December). *Discovering OpenSees -- Force-based Element vs. Displacement-based Element.* Retrieved January 10, 2020, from OpenSees: https://opensees.berkeley.edu/wiki/index.php/Discovering_OpenSees_--_Force-based_Element_vs._Displacement-based_Element

Yew, A. (2011). *Numerical Integration: Gaussian quadrature rules*. Retrieved January 20, 2020, from http://www.dam.brown.edu/people/alcyew/handouts/GLquad.pdf