NUMERICAL ANALYSIS OF THE STABILITY OF A COLUMN

LATERALLY RESTRAINED BY A FLEXIBLE BRACE

By

Natalia Kulchitskaya

RECOMMENDED: ______________________

Mr. Liao Yang

____

Dr. Andrew Metzger

____

Dr. Yuri Shur

____

Dr. Leroy Hulsey
Advisory Committee Chair

____

Dr. Robert Perkins
Department Chair, Civil and Environmental Engineering

APPROVED: ______________________

Dr. Douglas Goering
Dean, College of Engineering and Mines

____

Dr. John Eichelberger
Dean of the Graduate School

8/15/14

Date

NUMERICAL ANALYSIS OF THE STABILITY OF A COLUMN LATERALLY RESTRAINED BY A FLEXIBLE BRACE

A

THESIS

Presented to the Faculty

of the University of Alaska Fairbanks

in Partial Fulfillment of the Requirements

for the Degree of

MASTER OF SCIENCE

By

Natalia Kulchitskaya, M.S.

Fairbanks, Alaska

August 2014

Abstract

The paper analyses the behavior of a structure which includes a classically restrained steel column under an axial load and a single flexible brace attached at an arbitrary point along the column to restrict its lateral deformation. The column is assumed to have an initial imperfection limited according to the current code requirements. Focusing on lateral deformations only, the paper studies the maximum load the system can resist before failure, as well as a brace force arisen at this load. Due to the complexity of the problem when it is extended from the elastic region to the plastic domain, a numerical solution is utilized. In the current work, a student version of Abaqus™ provides results of finite-element analysis implemented for a variety of ASTM A992 steel W-Shaped columns. The results confirm that the failure load and brace force highly depend on brace location and its stiffness. It is also shown that the current code provision of a brace load is not always conservative for braces shifted from the center of the column.

Table of Contents

List of Figures

Page

List of Tables

List of Appendices

Acknowledgments

The author would like to express the deepest appreciation to her committee chair, Professor J. Leroy Hulsey, for the valuable guidance and advice. His willingness to motivate the author contributed tremendously to the project.

The author also thanks the committee members, Dr. Yuri Shur, Dr. Andrew Metzger and especially Mr. Liao Yang for continuous support and important recommendations for the work improvement.

The author would like to acknowledge the patience and help provided by the family members especially her husband Anton Kulchitsky.

This study could not be completed without assistance from the Abaqus Internet community where anonymous researchers were ready to help with very valuable advice.

INTRODUCTION

The ultimate strength of a steel column can be significantly increased by bracing it with an intermediate elastic restraint attached at an arbitrary position along the column.

The greatest effect can be attained if the restraint is at the mid-height; however, it is often convenient to locate the brace higher or lower. After an axial load is applied to a column, a certain percentage of this load is transferred to the brace. Obviously, braces can experience greater or smaller loads depending on their position along the column. At this moment, the percentage of the load taken by the brace is not studied very well whereas the brace forces should be closely examined in order to evaluate the satisfactory requirements for restraint strength and stability. Current code brace stiffness and strength requirements are mainly conservative for braces located at mid-height and should be reconsidered for braces situated at other positions along the column length.

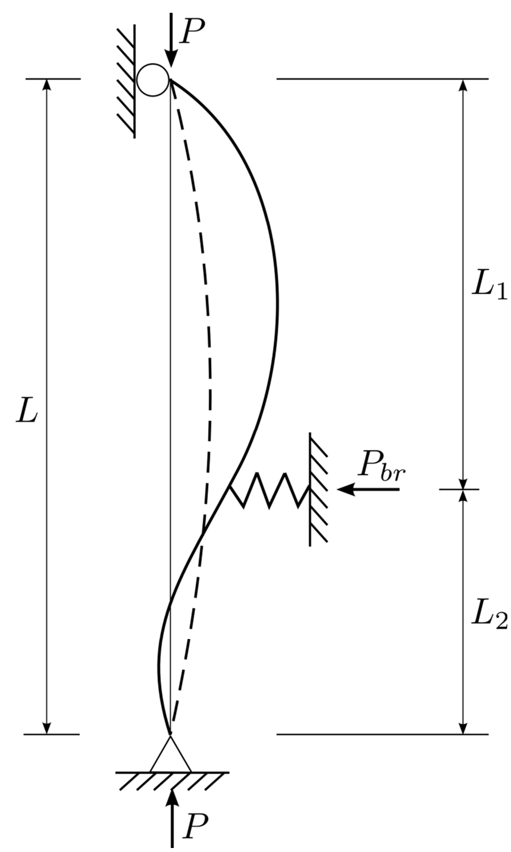


Figure 1. Initially Imperfect Column with the Brace at Arbitrary Position

The current recommendations for the brace strength and stiffness are mostly originated from Winter's model described in his paper in 1960 [1]. This paper was one of the first works that analyzed characteristics required for adequate lateral bracing. Previous works, for example Timoshenko and Gere (1936) [2], already established equations connecting column critical load and brace stiffness for classically supported straight elastic beams. However, Winter was able to provide important conclusions regarding the required brace stiffness and acceptable brace strength. In his work the author analyzed a simple model of an axially loaded column classically supported and restricted by the elastic brace attached in the middle of the column span. The specific detail of Winter's model is a fictitious hinge at the brace point. The presence of the hinge sets the moment to zero that helps to keep buckling equations very simple even in case of initial column imperfection. Using this method Winter received the same results for the ideal brace stiffness (required stiffness for full bracing) as rigorous elastic theory for perfectly straight columns. This confirmed the validity of the model and the author used it to develop new results: he showed that the brace stiffness required to produce full bracing in an imperfect column exceeded the stiffness required for the straight column (p.812).

By that time Winter had already distinguished the stiffness and the strength of the brace. He stated that full bracing can be achieved when both: brace stiffness and brace force satisfied certain criteria (p.813). Thus, the brace force was chosen to be less than 1% of the strength of the column. This recommendation was based on the serious of tests in which column failure occurred due to the fracture of the braces. However it is remarkable that among the experiment results there is a case when the brace force reached 2.2% of the column strength at failure (Table 1, p.809). It happened while a column was supported by a single brace at the mid-height. Winter ignored this result only because in this case the column buckled first before any brace damage occurred (p.810). Nevertheless, Winter contributed to the development of relationship between the brace-column parameters and emphasized the importance of both brace properties: stiffness and strength.

In 1979, O'Connor [3] continued investigation of the relationship between bracing parameters and a column's critical buckling load. He built a finite-elements model for a typical W-shape column axially loaded, classically supported and restricted by a brace at about the mid-point of the column. Thus, O'Connor analyzed the brace slightly shifted from the column's mid-height and how variations in the brace location influence the value of the maximum column strength. He was one of the first who discussed column's unequal spans. He found that the brace stiffness dose not reach the ideal value if the brace dose not locate in the center of the column (p.70). So there is no such a term as "ideal stiffness" for unequally braced columns. Using his results O'Connor also concluded that the brace position greatly affects the critical buckling load (p.74).

In 1992, Stanway [4] and others also applied the finite-element analysis for the initially imperfect rectangular column with an intermediate elastic restraint at an arbitrary position. In comparison with the previous work an initial imperfection was added. Using elasto-plastic analysis Stanway et al were able to receive results for any range of the column's slenderness ratio and demonstrated that column flexure could be a very significant contributor to the brace force. Stanway et al calculated that under certain conditions the brace force created in the restraint can be relatively large compared to the typical estimation. Thus, brace force reached 3 percent of the axial load for the column with the large slenderness ratio and when the restraint was significantly shifted from the center. For the mid-height brace its load exceeded 2.2 percent (Part 1, Table 5, p. 214).

The next significant step was made by Plaut and Yang in 1992 [5] when they presented an analytical analysis of the buckling behavior of the column laterally restricted by the flexible brace at arbitrary position along the column. The authors solved linear elastic equations with various boundary conditions for both straight and imperfect columns. The results illustrated that the brace stiffness and brace position greatly affected the column critical load and the brace force. Plaut and Yang also concluded that brace forces were only small percentages of the axial load while the axial load was relatively low. If the axial load exceeded the critical load of the elastic column, bracing forces significantly grew (p. 2910, Fig. 13 p. 2909).

In 1993, Clark and Bridge [6] continued investigation of the topic and examined column-brace behavior using nonlinear numerical method which allowed them to cover as elastic as plastic domains. They also took into account column residual stresses besides its initial crookedness. At the same time, the authors limited their investigation by analyzing only one type of classically supported W-Shaped steel columns - 200UC46.2 (equivalent of W8x31) restrained by a central brace only. Clark and Bridge obtained that the brace force value varied from approximately 0.5% to 2% of the ultimate (maximum) axial load depending on the brace stiffness (p.82, Fig.7 p.83). They also made an important conclusion regarding brace positioning since they found that a small offset from the perfectly central position could cause a significant decrease in the ultimate strength (p.77, Fig.2-3 p.78) and as a result alter the brace force value (Fig.7 p.83).

In 1994, Yura [7] expanded Winter's model to investigate how brace stiffness could affect brace force. He found that increasing of the brace stiffness caused less lateral deflection and as a result reduced corresponding brace force and its percentage of the critical load. Thus, according to the Winter's model, the brace force $k_{br} = 2k_{ideal} = \frac{4P_e}{L_{unbr}} = \frac{4\pi^2 EI}{L_{unbr}^3}$ can keep the brace force less than 1% of the critical Euler load where L_{unbr} is the unbraced span length or half of the column (pp. 821-823). This result became the base for the current code recommendation for adequate bracing.

In 1996, Yang [8] analyzed steel W-shaped classically supported columns under the axial load restrained by a flexural brace. Similarly with Clark and Bridge [6], Yang took into account all types of possible column imperfections: initial crookedness and residual stresses. But then he also considered the situation when the brace is located at other positions than column mid-height. And using numerical finite-elements approach, Yang was able to implement inelastic model and describe column-brace behavior for any type of slenderness ratio. He found that the brace force reached the highest percentage of the maximum axial load for the column with the large slenderness ratio and when the brace is significantly shifted to one end of the column. The author found that for described situation the brace force could be higher than 3% while the column is fully braced (Table 3 p.81). Yang also developed design charts for the required brace strength for W10 columns made of ASTM A36 steel. This chart demonstrated that the brace force could exceed 1 or even 2% of the maximum axial load (which is achieved at failure) in many practical cases (Fig. 52 p.111).

The current study also analyzes brace forces arisen at failure and tries to verify if the current recommendation for brace stiffness and strength are conservative for the whole range of the column's possible axial load. For this purpose, a simple model of classically supported column under the axial load with the elastic restrain at arbitrary position (Fig. 1) is built using Abaqus™ software. The column is assumed to have a maximum allowed initial imperfection equaled to L/1000 and to be in the form of a half sine wave. The study was limited by considering only A992 steel columns with W-shaped cross-sections. Several compacted W-shapes of various geometry were chosen for the current work.

FAILURE LOAD

As mentioned above the increasing of the axial load, P, can cause the increasing of the brace force, P_{br}, and the ratio P_{br}/P may grow too. According to Plaut and Yang [5], the brace force measured as a percentage of the axial load intensively grows with the increasing of the axial load for elastic columns regardless of brace stiffness and brace location (Fig. 13 p. 2909). It is important to understand what would be the maximum axial load the column can resist before failure. Since the ratio P_{br}/P will also reach its maximum value at failure, the brace should be designed to resist this maximum percentage of the axial load. This simple fact is a subject of discussion because typically braces are designed for the loads significantly smaller than the failure load, P_{fail}. It happens since the critical load for the unbraced span is calculated with the assumption that this span has the same boundary conditions as the column has. For example, in the case of a classically supported column with the rigid brace, the longest unbraced span will also be assumed classically supported to estimate its critical load. This is a convenient way of estimation but it is not accurate because a *pin-brace* span capacity is significantly higher compared to the capacity of a *pin-roller* column. This is because the span rotation at brace point is limited and the rotational moment is not zero at that point while a classically supported column can freely rotate at both ends and the bending moments equal to zero there. Figures 2 and 3 demonstrate the difference between a typically estimated critical load and a real failure load for different brace locations. Lambda at the abscissa axis represents slender ratio $\lambda = \frac{1}{\pi}\frac{L_1}{r}\sqrt{\frac{F_y}{E}}$ where L_1 is the longest unbraced span, r is radius of gyration of the column's weak axis, F_y is column yield stress equal to 50 ksi and E is Young's Modulus of steel. P_y is the yield strength of the column and ratio P_{fail}/P_y represents unit less load at failure. The plots were obtained for steel columns with W10x19 cross-section using Abaqus™ software. See Appendix A for the numerical model description.

The bottom line of these plots is constructed according to the equations (1) provided at Ziemian [9] for SSRC column strength curve 2P (Group II columns) assuming the initial imperfection to be equal to L/1000 and yielding stress to 50 ksi. These equations describe the average critical load for classically supported inelastic columns which can be used as the first estimation of column's span failure load when the span is supported by a brace at one end. This will be referred to as the estimated critical load.

$$
\begin{aligned}
&0 \leq \lambda \leq 0.15 && \frac{P}{P_y} = 1 \\
&0.15 \leq \lambda \leq 1.2 && \frac{P}{P_y} = (0.979 + 0.205\lambda - 0.423\lambda^2) \\
&1.2 \leq \lambda \leq 1.8 && \frac{P}{P_y} = (0.030 + 0.842\lambda^{-2}) \qquad (1) \\
&1.8 \leq \lambda \leq 2.6 && \frac{P}{P_y} = (0.018 + 0.881\lambda^{-2}) \\
&2.6 \leq \lambda && \frac{P}{P_y} = \lambda^{-2} \text{ (Euler curve)}
\end{aligned}
$$

The top lines of Figures 2 and 3 represent the real failure load for the spans created by the brace attached exactly at the mid-height, $L_1 = 0.5L$, and calculated numerically. The failure load is significantly higher in the case of $L_1 = 0.5L$ which is explained by the fact that the column can resist an axial load longer when it experiences symmetrical deformation. Symmetrical and asymmetrical column deformations can be seen in Figure 4. Technically the column fails through the symmetrical mode when the restrain is located exactly at the center of the column and can easily switch to the asymmetrical mode if the brace is slightly shifted from the center (Clark and Bridge (1993) p. 77-79 Fig. 2, Fig. 3). Since insignificant shifts are possible in practice, it would be more conservative to consider the asymmetrical failure mode when the maximum resisting loads are lower. However, according to Stanway et al (1992), the assumption of antisymmetry is conservative for column design but is non-conservative for the estimation of the brace

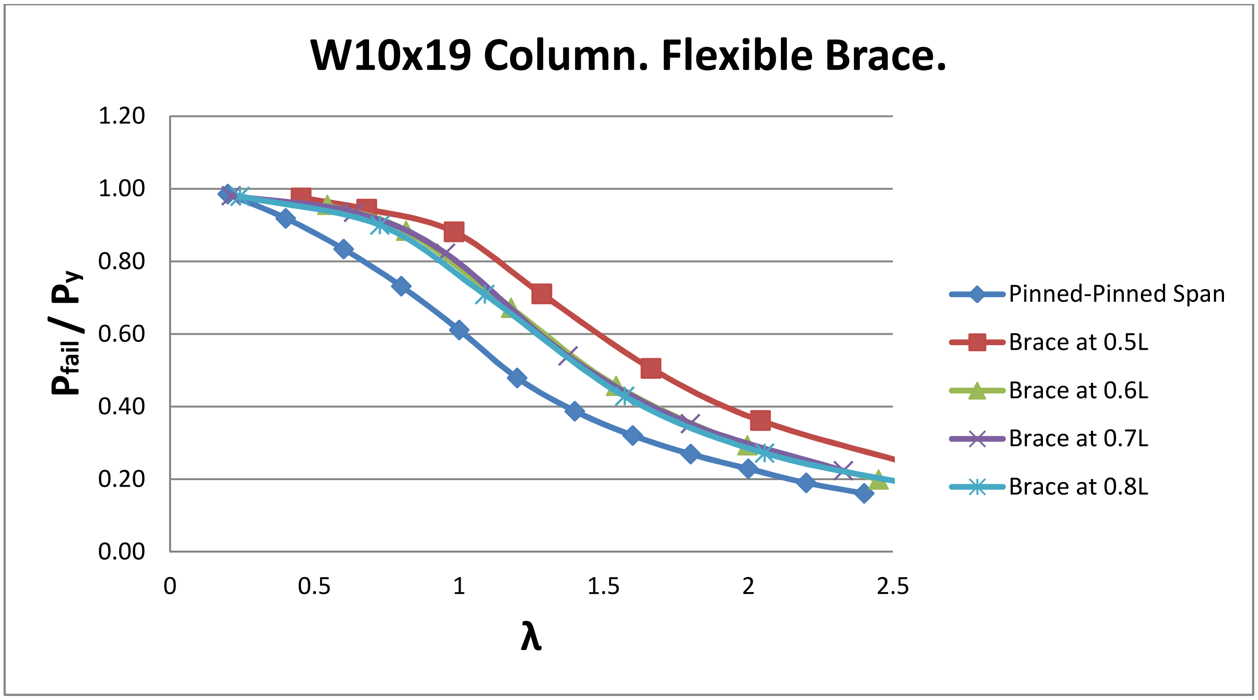


Figure 2. Difference Between Estimated Critical Load for Classically Supported Column and Failure Load of the Same Length Span but Supported with the Flexible Brace at One End.

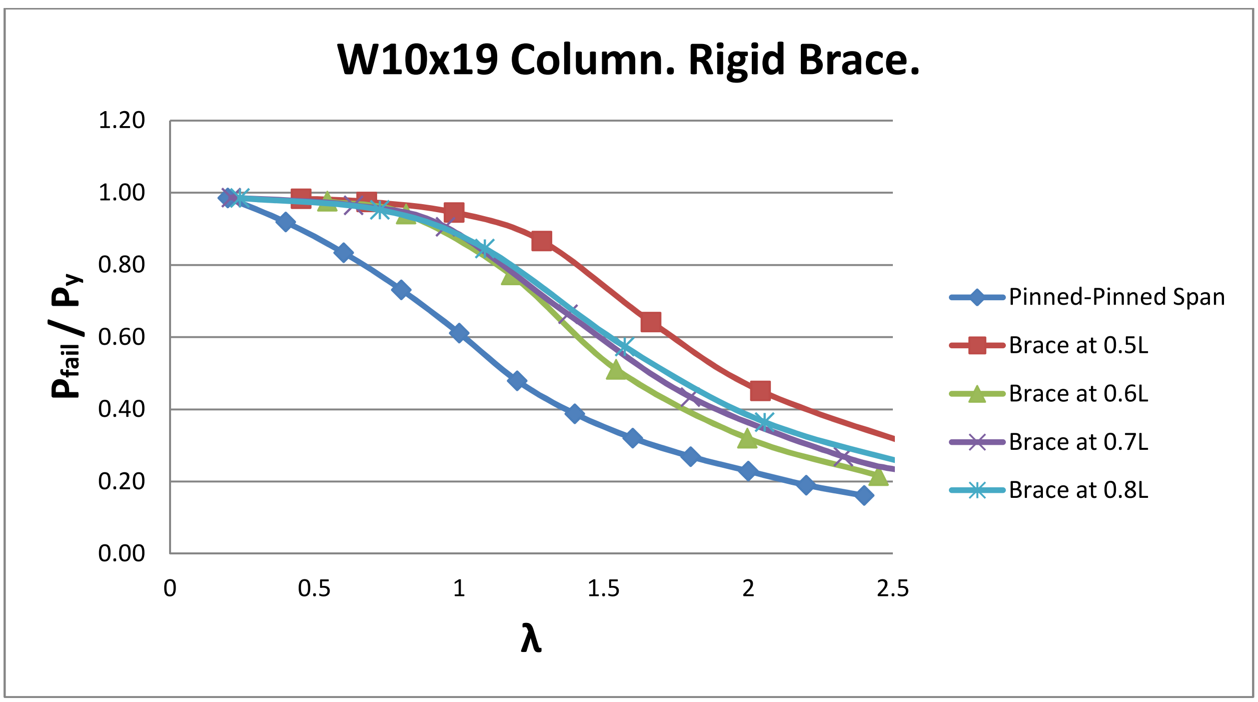


Figure 3. Difference Between Estimated Critical Load for Classically Supported Column and Failure Load of the Same Length Span but Supported with the Rigid Brace at One End.

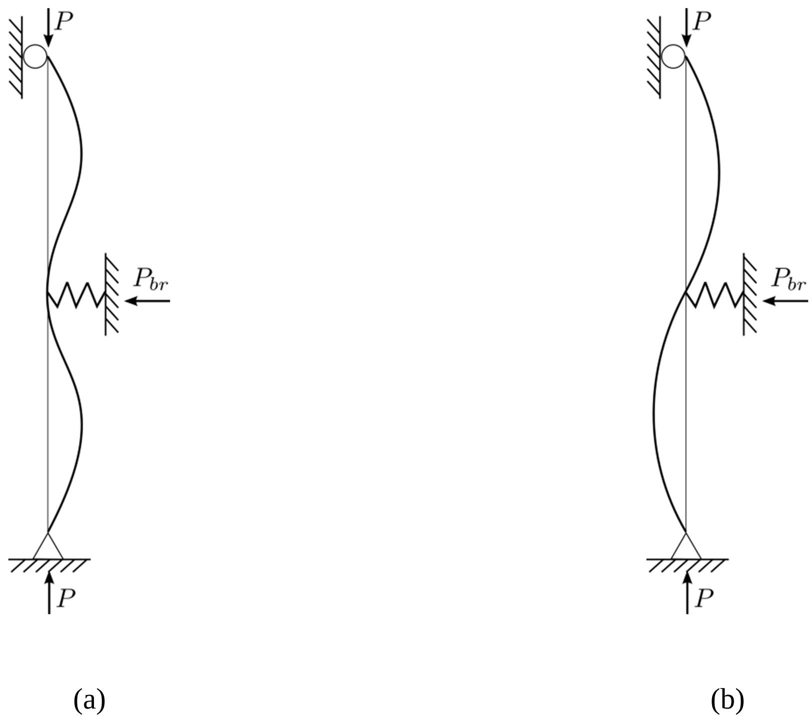


Figure 4. Symmetrical (a) and Asymmetrical (b) Shapes of the Column Laterally Deformed Under the Axial Load.

force (p. 205). When the brace is attached at any other position except the mid-height, the failure load tends to be very similar for spans of the same length.

The plots at Figures 2 and 3 differ due to the brace stiffness: the brace is flexible or has smaller stiffness for the first plot and the brace is rigid, has higher stiffness, for the second plot. This stiffness value heavily influences the failure load. In practice, the brace stiffness depends on the type of brace connection. Figure 5 (a) shows that the brace can be situated along the weak axis of the W-shape and in this case column lateral deformation causes brace bending. When the brace is attached perpendicular to the cross-sectional weak axis, Figure 5 (b), column lateral deflection will cause brace tension or compaction and possibly buckling but the brace capacity is much greater than that of the bending brace. So the brace can be called flexible for the case (a) and rigid for the case (b).

It is important to study the current recommendations for brace stiffness and stress. According to Ziemian [9], the design (LRFD) recommendation for discrete bracing is

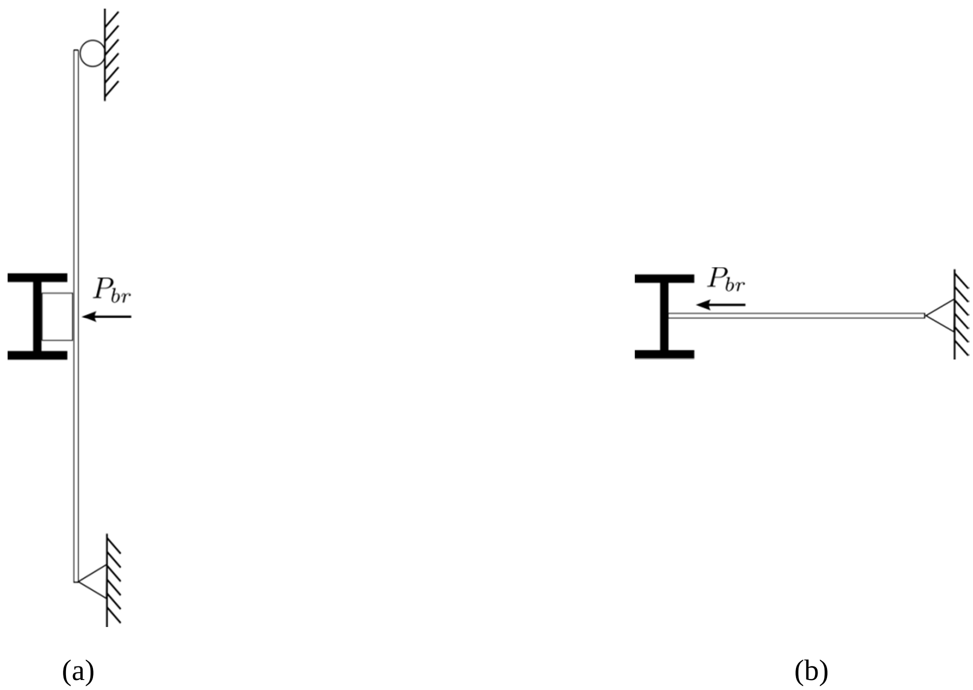


Figure 5. Top View of Column Bracing: (a) Flexible Brace Positioning with Respect to the Column Cross-Section; (b) Rigid Brace Positioning.

$$\phi = 0.75 \qquad k_{req} = N_i \frac{2P}{\phi L_1} \qquad P_{br} = 0.01P \qquad (2)$$

where P is the factored column axial load, L_1 the required brace spacing, $N_i = 1 + \frac{L_1}{L_2}$ where L_1 is the longest span and L_2 is the shortest span of the column. This recommendation provides a relatively small brace stiffness, $k_{br} = k_{req}$, to ensure that the flexible brace can still restrict the column failure and at the same time keep the brace force as small as 1%. When a rigid brace is designed, even the smallest shapes among those available in the industry provide brace stiffness as high as $50k_{req}$ (some variation are possible depending on the brace length). Later in this work, the flexible brace will be assumed to have stiffness equal to k_{req} and rigid brace will have stiffness $50k_{req}$.

When comparing plots at Figures 2 and 3, it is not surprising that the rigid brace provides a better lateral resistance compared to the flexible brace. Nevertheless, it would be still beneficial to estimate the difference between those cases. Table 1 contains values which demonstrate the difference between the failure load and the estimated critical load when a brace is attached at 0.4L.

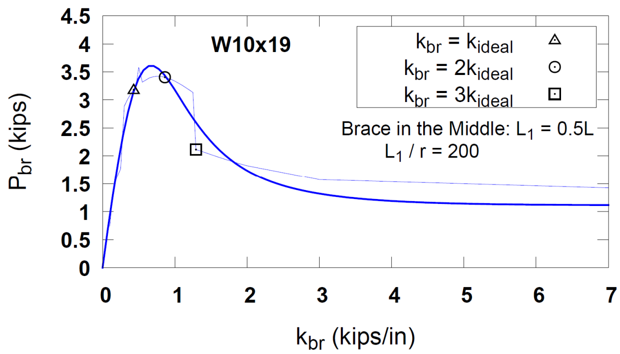
It is remarkable that a designer can have up to 50% higher column capacity with the rigid brace located at the center of the column or slightly shifted from it. A perfectly centered brace provides an even higher capacity but it is conservative to assume that it may be slightly shifted and so fail through the asymmetric mode.

Table 1. Difference Between Real Failure Load and Estimated Critical Load for W10x19 Column When Brace is Attached at 0.4L

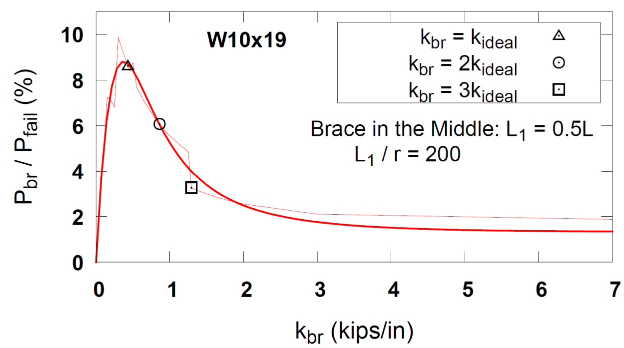
Flexible Brace, $k_{br} = k_{req}$			**Rigid Brace, $k_{br} = 50k_{req}$**		
$\lambda = \frac{1}{\pi}\frac{L_1}{r}\sqrt{\frac{F_y}{E}}$	$\frac{L_1}{r}$	$\frac{P_{fail}}{P_{cr}}$ 100%	$\lambda = \frac{1}{\pi}\frac{L_1}{r}\sqrt{\frac{F_y}{E}}$	$\frac{L_1}{r}$	$\frac{P_{fail}}{P_{cr}}$ 100%
0.54	41.2	11.2%	0.54	41.2	13.7%
0.82	61.8	22.4%	0.82	61.8	30.3%
1.18	89.2	36.9%	1.18	89.2	57.5%
1.54	116.7	35.2%	1.54	116.7	51.3%
2.00	151.0	27.8%	2.00	151.0	39.6%
2.45	185.4	28.5%	2.45	185.4	39.5%

BRACE FORCE

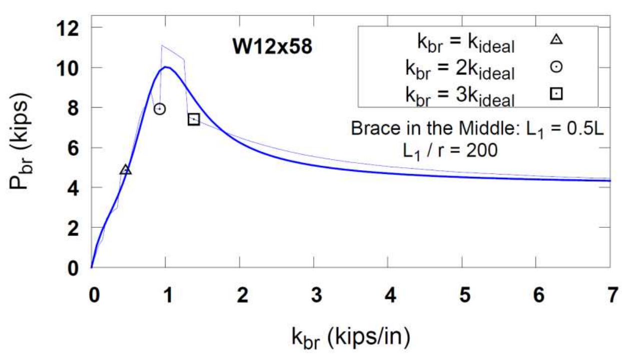
Since a designer can design a structure for the maximum axial load the column-brace system can resist, the brace force value should be analyzed at column's failure when the axial load and the brace force reach their maximum. According to Stanway et al (1992) and Yang (1996), the brace force highly depends on the brace location, column's slenderness ratio and brace stiffness. The parametric study implemented for several typical compact shapes, such as W10x19, W12x58, W14x145, helps to analyze brace behavior. The next four plots in Figure 6 demonstrate that brace force can be associated with its percentage of the failure load since there is strong correlation between these values. It is also interesting to note that brace force doesn't always decrease with the increasing of the brace stiffness. Using Winter's approach for the elastic columns and axial forces lower than Euler load, a designer can be confident that increasing of the brace stiffness from k_{ideal} to $2k_{ideal}$ will always decrease the brace load. However, this is not true in the



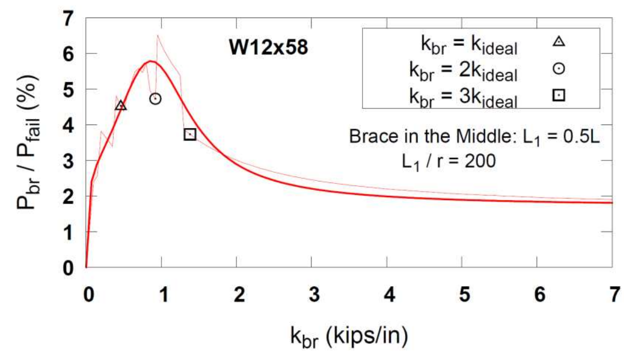
(a) Brace Force at Failure, W10x19



(b) Brace Force as Percentage of Failure Load



(c) Brace Force at Failure, W12x58



(d) Brace Force as Percentage of Fail Load

Figure 6. Brace Force vs Brace Stiffness when Brace is in the Middle and $L_1 / r = 200$.

case with higher axial loads even when the column approaches the fully elastic behavior through the increasing of the slenderness ratio L_1/r.

The brace force can increase with decreasing of the brace stiffness. Figure 6 contains plots that describe cases when the brace is attached in the middle of the column, $L_1 = 0.5L$, and the slenderness ratio is relatively high, $L_1/r = 200$, i.e. the column behavior is close to the elastic buckling. Plot (b) built for W10x19 is an example of the situation when the brace force percentage at $k_{br} = 2k_{ideal}$ is lower than the brace force percentage at $k_{br} = k_{ideal}$. Plot (d) built for W12x58 illustrates an opposite situation: $k_{br} = 2k_{ideal}$ cannot guarantee a smaller brace force compared to the force when $k_{br} = k_{ideal} = \frac{2P_e}{L_{unbr}}$.

It would be interesting to compare brace force profiles depending on brace position and column's slenderness ratio. Figure 7 demonstrates how the brace force profile changes with the change of slenderness ratio. The brace is not attached at the middle anymore. The longest unbraced span, L1, consists of 0.6 of the column length for all 3 profiles. But the length of the column is changing so the slenderness ratio is changing too. The shapes of the profiles differ from each other. The bottom plot describing the span with the smallest slenderness ratio does not have a peak or a significant bump along the line. This is a useful fact for a designer since brace force stays relatively low for any brace stiffness. However, in this situation again the brace force at $k_{br} = 2k_{ideal}$ is slightly higher than at $k_{br} = k_{ideal}$. In this case, ideal stiffness was estimated using critical load for the classically supported inelastic span instead of the Euler load. At the higher values of brace stiffness, when the brace can be called rigid, the brace force became almost constant and the values of the brace force percentage for all three plots became closer to each other. This means that variation in the slenderness ratio affects the brace load for rigid braces very little compared to the flexible ones.

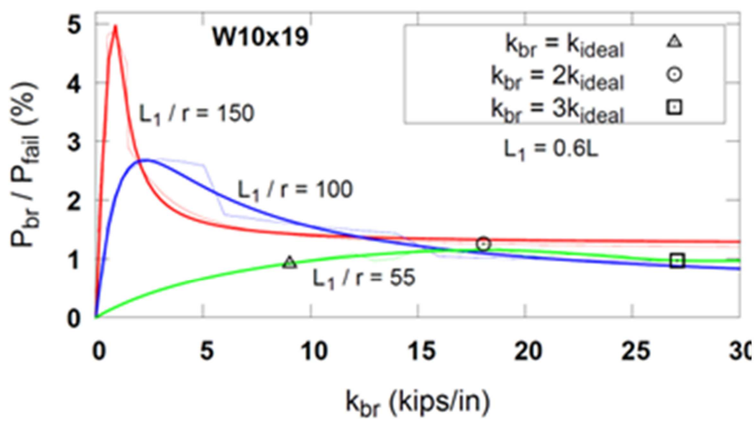


Figure 7. Brace Force Profile Change Depending on Various Slenderness Ratios by Example of W10x19 Column.

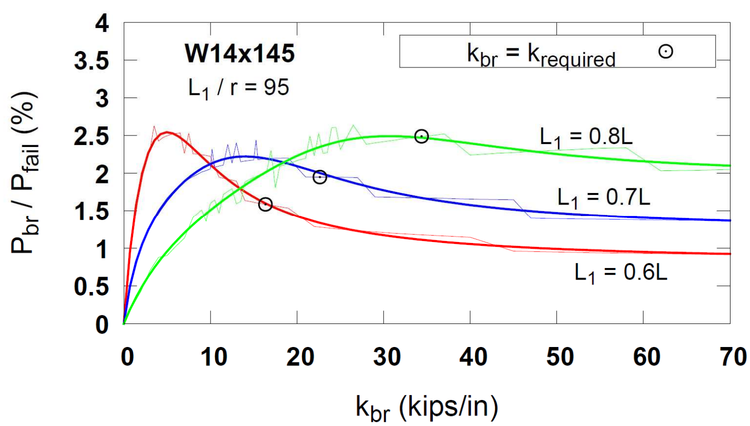


Figure 8. Brace Force Profile Change Depending on Brace Position by Example of W14x145 Column.

The next figure demonstrates brace force behavior depending on the brace position. The plots in Figure 8 were constructed for W14x145 columns such that the longest unbraced span varies from 0.6L to 0.8L but the slenderness ratio stays constant for all cases. The slenderness ratio was chosen to be 95. The maximum values of the brace force percentages are very similar for all three plots proving that the brace position does not significantly affect the flexible brace load. However, the profiles are not close to each other when brace stiffness is large. Braces attached closer to the center of the column take less percentage of the column's axial load compared to the braces significantly shifted form the column's mid-height. Thus, the rigid brace located at 0.8L, where L is the length of the whole column, is supposed to restrict a two times larger percentage of the failure axial load in comparison with the brace attached at 0.6L. For braces shifted from the center of the column there is a more complex formula defining the required brace stiffness. The required stiffness is not associated with the $2k_{ideal}$ anymore; it can be calculated according to (2). While the factored axial load is not known, it can be approximated by the estimated critical load for inelastic span. This required stiffness is marked at all profiles of Figure 8 and it is remarkable that recommended stiffness is adequate in all cases: the brace force does not grow for $k_{br} > k_{req}$. It means that the maximum brace force can be expected at stiffness equaled to k_{req} and the percentage of the axial failure load which brace takes at this stiffness varies from 1.6% to 2.5% that collides to the same requirements (2) where brace force is expected to be lower than 1% of the axial load.

HOW BRACE FORCE SATISFIES THE STRENGTH REQUIREMENTS

As mentioned above, the brace load transferred from the column's axial load can exceed 1% of it while another requirement is satisfactory, i.e. brace stiffness is greater than or equal to the required stiffness. It is important at this stage to examine when the brace force becomes large enough to surpass the 1% limit. Figures 9 and 10 show how percentage of the brace force at failure depends on the column slenderness ratio and the brace position. These results are obtained using an Abaqus™ model applied for the columns of various lengths and various cross-section sizes: W10x19, W12x58 and W14x145. Brace stiffness is set to k_{req} for Figure 9 and to $50k_{req}$ for Figure 10 or, in other words, the brace is chosen to be flexible for Figure 9 and rigid for Figure 10. Initial crookedness is assumed to have a half sine wave shape with the maximum amplitude equal to L/1000. Comparing plots at Figure 9 and 10, it can be concluded that brace force percentage grows faster and reaches higher values in case of the flexible brace. Nevertheless, both figures demonstrate that the brace force can exceed 1% of the column's failure load even within the recommended range of the slenderness ratio. This range, recommended for 50-ksi steel, was obtained from Ziemian [9] (Fig. 3.18 p.47). This analysis shows that the column-brace structure is at highest risk when the column has larger slenderness ratio, L_1/r, the brace has the minimum possible stiffness, $k_{br} = k_{req}$, and the brace locates farther from the column's mid-height. These parameters define the situation when the brace load can exceed 1%, 2% or even 3% of the column's axial load. The charts at Figure 9 and 10 can be the good tools for estimation of the expecting brace force at failure. However, the plots are built as an approximation of the data received for three different cross-sectional shapes: W10x19, W12x58 and W14x145. Therefore, the plots illustrate the average brace force behavior while its absolute values can be slightly higher or lower.

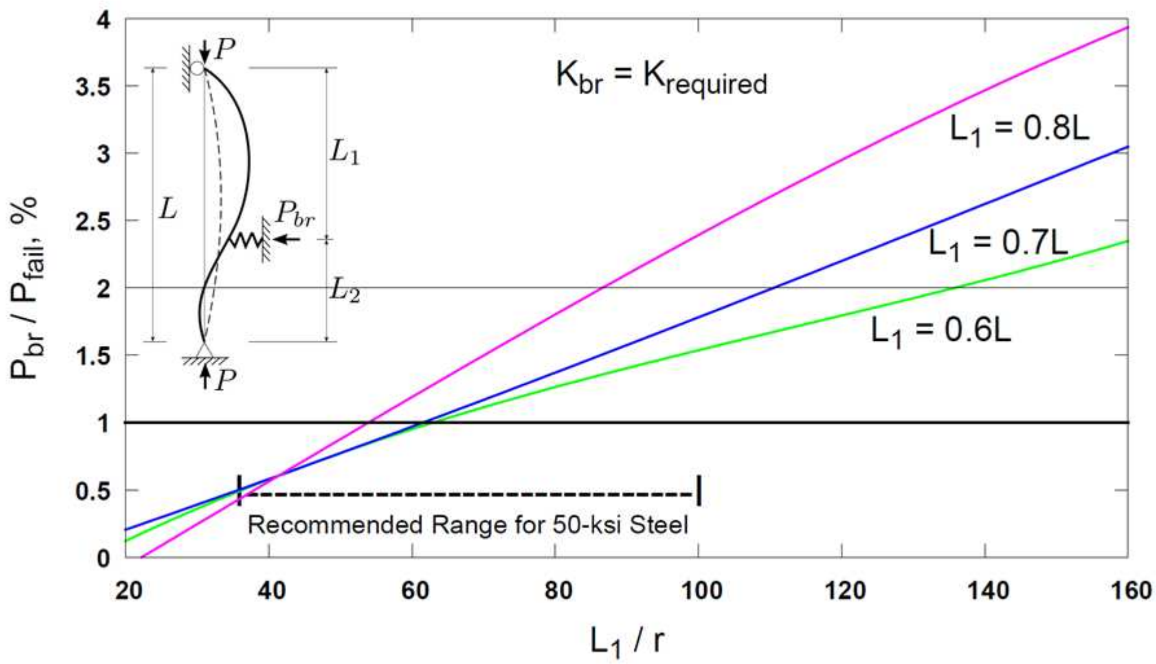


Figure 9. Percentage of Column's Axial Load the Flexible Brace is Supposed to Resist at Failure

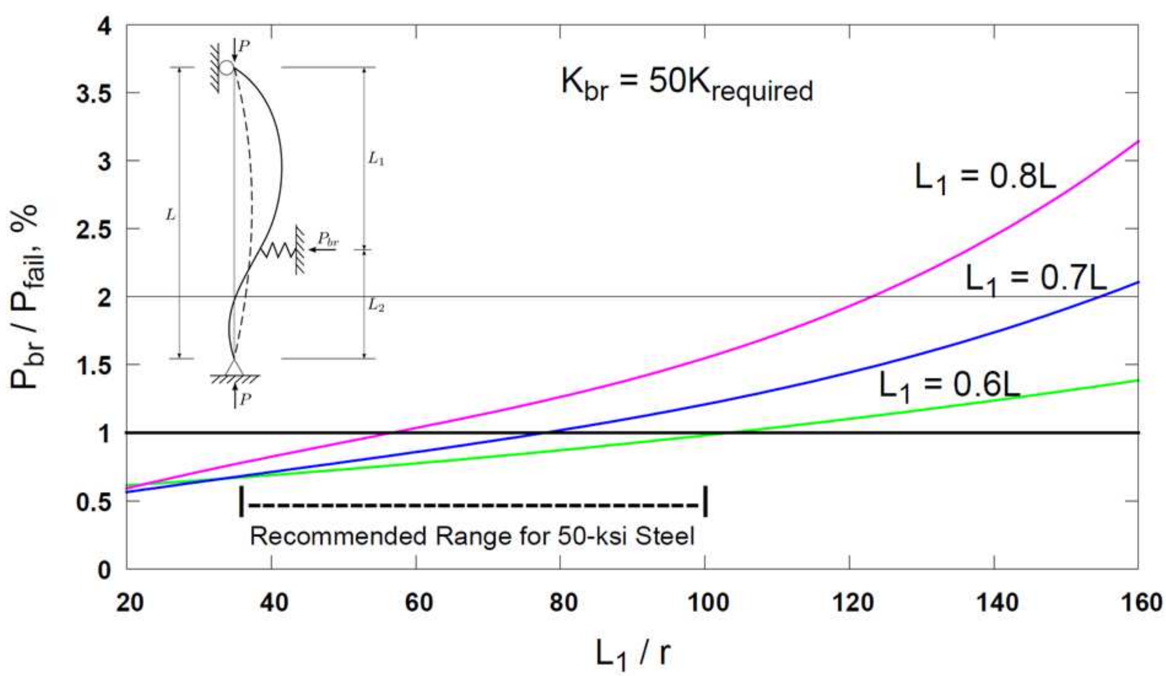


Figure 10. Percentage of Column's Axial Load the Stiff Brace is Supposed to Resist before Failure

According to Yang [8] A36 columns behave similar in terms of producing brace load. Plots in Figure 11 demonstrate that the brace load can exceed 1% limit when the brace stiffness $k_{br} = k_{full}$ where k_{full} represents the stiffness provided fully bracing according to the statistical analysis of Group II, W10 columns:

$$k_{full} = (1.054\lambda + 4.863 - \frac{0.3065}{\lambda})\frac{P_{fail}}{L_2}$$

where $\lambda = \frac{1}{\pi}\frac{L_1}{r}\sqrt{\frac{F_y}{E}}$, L_1 is the longest unbraced span, L_2 is the shortest unbraced span, r is radius of gyration of the column's weak axis, F_y is column yield stress (equaled to 36 ksi in this particular case only), E - Young's Modulus of steel and P_{fail} is column's axial load at failure.

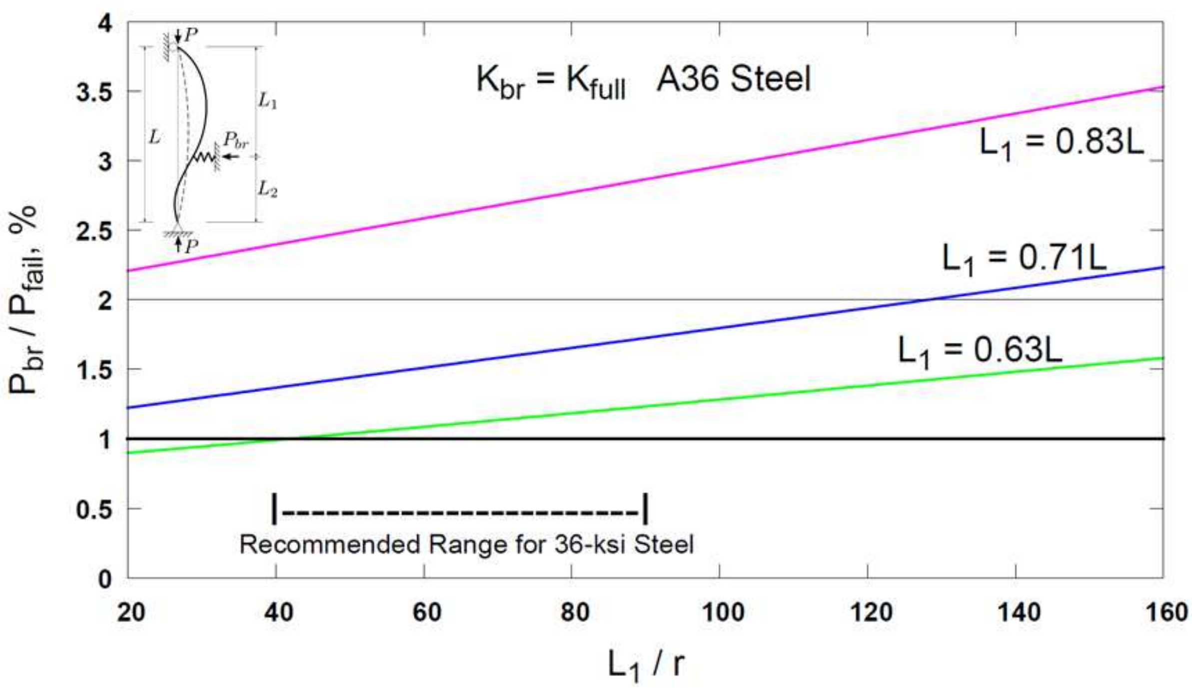


Figure 11. Percent of Column's Axial Load the Brace is Supposed to Resist for ASTM A36 Steel

Since the failure load is typically not available to the designer, it would be more useful to compare the brace force with the critical load which can be easily estimated using the current code provision. Thus, Figures 12 and 13 illustrate what percent of estimated critical load the brace can take at failure. If a designer wants to be conservative regarding the brace load, he should design the brace according to the

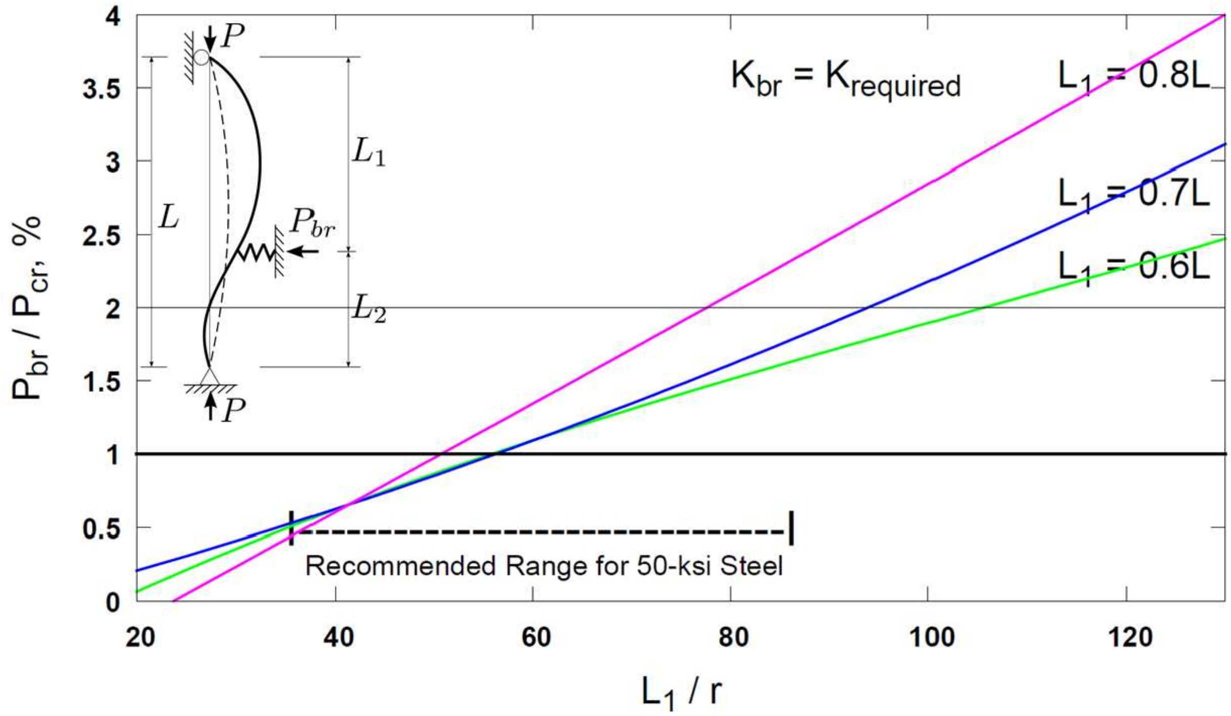


Figure 12. Percentage of Column's Critical Load Transferred to the Brace at Failure, $k_{br} = k_{req}$

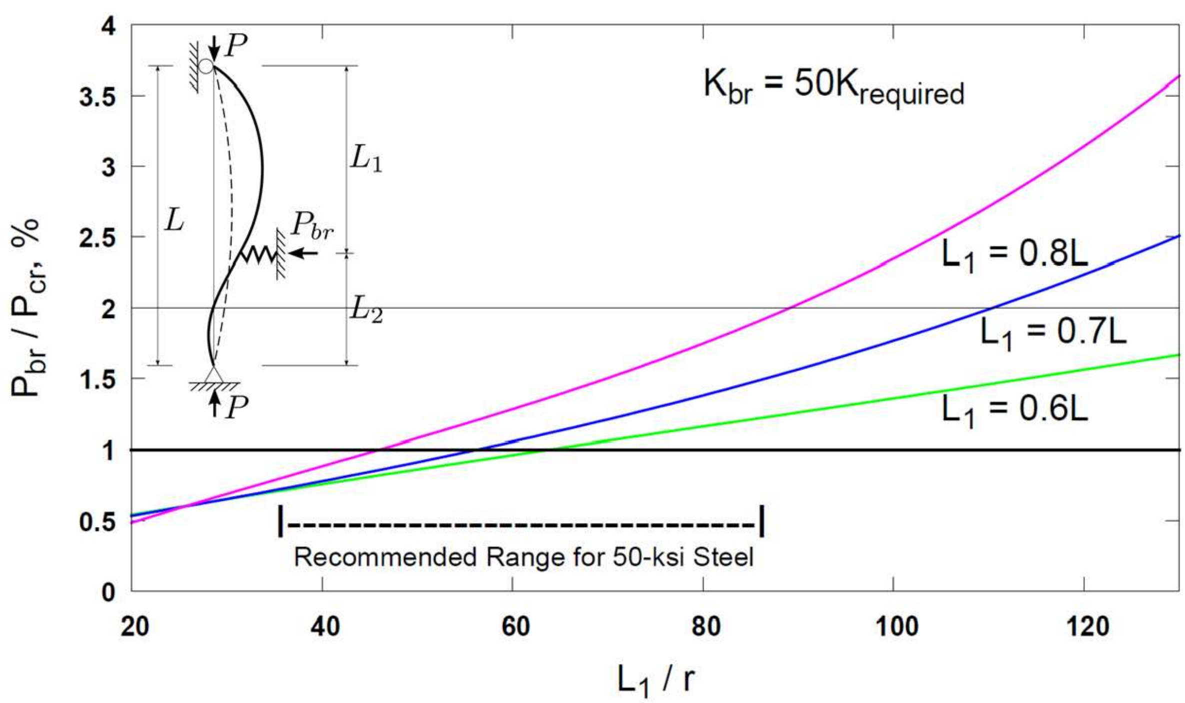


Figure 13. Percentage of Column's Critical Load Transferred to the Brace at Failure, $k_{br} = 50k_{req}$

charts of Figures 12 and 13. This approach will guarantee that the brace failure cannot appear before the column's collapse.

Using the numerical approach of Abaqus™, one more important fact was established: 1% limit can be broken when the column's axial load is not as high as the failure load but equal to the critical load for inelastic span, P, described at (1). This is the force designers typically use to estimate the load which the longest unbraced span, L_1, can sustain. Inefficiency of the current requirements (2) even in case of relatively small axial load can be illustrated through the example.

Consider 50-ksi steel W14x145 column classically supported and restrained by a flexible brace in such a way that $L_1 = 0.7L$. The brace length, L_{br}, is assumed to be 150 inches. The column's length, L, is 680 inches and it is axially loaded by a factored load, P, equal to 745 kips. Choose an appropriate brace size to satisfy stiffness and strength requirement (2).

Radius of gyration of the column's weak axes, r, equals 3.98 inches so slenderness ratio is

$$\frac{L_1}{r} = \frac{0.7L}{r} = \frac{0.7*680}{3.98} = 119.6$$

For this slenderness ratio, equations (1) give the estimated critical load $P_{cr} = 749.3$ kips that is still higher than the chosen factored load P = 745 kips so the bracing is adequate in terms of column's lateral deformation. The column should not fail if the brace is designed properly.

First, the brace should possess the required stiffness. According to (2)

$$k_{req} = (1 + \frac{0.7L}{0.3L})\frac{2*745}{0.75*(0.7*680)} = 13.0\,\frac{kip}{in}$$

Brace force creates a concentrated load at the center of the brace, Figure 5(a). Assuming that the brace is classically supported, the maximum brace deflection is

$$\delta = \frac{P\,(L_{br})^3}{48EI}$$

or

$$I_{req} = \frac{P}{\delta}\,\frac{(L_{br})^3}{48E} = \frac{k_{br}(L_{br})^3}{48E} = \frac{13.0(150)^3}{48*29000} = 31.53$$

The smallest C-Shape which has a moment of inertia larger than I_{req} is C8x11.5. It has

$$I_x = 32.5 \quad \text{and} \quad S_x = 8.14$$

Now the second requirement for the brace strength should be checked. According to (2)

$$P_{br} = 0.01P = 0.01*745 = 7.45 \text{ kips}$$

C-Shapes are made of A36 steel and so the brace stress should not exceed 36 ksi:

$$\sigma_{max} = \frac{M_{max}}{S_x} = \frac{P_{br}L_{br}}{4S_x} = \frac{7.45*150}{4*8.14} = 34.32 \text{ ksi}$$

σ_{max} is less than 36 ksi so the second requirement is also satisfied. However, unlike the recommended brace force presented above, the numerical solution for the continuous column gives a higher value for this brace force. Under the given conditions and the maximum initial imperfection equal to L/1000, Abaqus™ gives brace force, P_{br} = 7.90 kips. It also means that the percentage of the axial load is higher than 1%:

$$\frac{P_{br}}{P} 100\% = \frac{7.90}{745} 100\% = 1.06\ \%$$

In this case

$$\sigma_{max} = \frac{7.90*150}{4*8.14} = 36.38 \text{ ksi} > 36 \text{ ksi}$$

This illustrates that it is possible to have the brace overstressed. Therefore C8x11.5 should not be assumed as a conservative choice for the brace design in the stated conditions.

If a designer wants the brace to be appropriately designed regardless of the column's axial load, then the chart at Figure 12 should be used. For slenderness ratio $\frac{L_1}{r} = 119.6$ and $L_1 = 0.7L$ the possible brace load can reach 2.75% of the column's estimated critical load so the brace should be designed to be able to resist this load

$$P_{br} = 0.0275*749.3 = 20.61 \text{ kips}$$

For example, C12x25 with $S_x = 24.0\ in^3$ will be a satisfactory choice:

$$\sigma_{max} = \frac{20.61*150}{4*24.0} = 32.20 \text{ ksi} < 36 \text{ ksi}$$

This is a conservative approach which guarantees that the brace never fails before the column failure.

CONCLUSION

Based on the study of classically supported 50-ksi and 36-ksi W-Shaped columns restricted against lateral deformation by flexible braces, it can be concluded that brace force, P_{br}, can be larger than 1% of the column axial load at failure. In some cases it can exceed 2% as well. If the axial load is less than the failure load, P_{fail}, then the brace force is smaller too but still can exceed 1%. We recommend reconsidering the bracing requirement using the results of the current research.

APPENDIX A. DESCRIPTION OF ABAQUS NUMERICAL MODEL

The numerical model of the axially loaded and laterally braced column was built using the Standard Mode of Abaqus™ Student Edition 6.12-2. This powerful software was chosen to accommodate elastic-plastic properties of the column, large deformation of the structure that causes extra nonlinearity of the model and initial imperfection or crookedness of the column. These goals were reached using a three-dimensional beam element, B31, for the column approximation. The brace was modeled by a linear elastic spring element, SPRINGA. The numerical algorithm included two steps: Linear Perturbation Analysis (to obtain crooked shape of the column) and Riks Method which allowed capturing the unstable behavior of the model.

Below there is a description of how this model can be built using Abaqus/CAE graphical interface. According to the Abaqus rules, its units should be chosen in the beginning and kept the same during the whole process. Inches (in) and kilo pounds (kip) were picked for this example.

1. Create the First Part - It is Going to Be a Column with Length of 400 in.
   The Module should be set on Part: choose the Create Part icon or double click on the Parts title in the list at the left side of the screen.
   Fill in the Create Part dialog according to Figure 14. Using the Create Lines option, sketch the straight vertical line for a wire beam from -200 to 200 or assign its length equal to 400. See Figure 15.

2. Create the New Material - A50 Steel.
   Choose Property for Module and hit the Create Material icon or double click on the Material title in the Model Tree on the left.

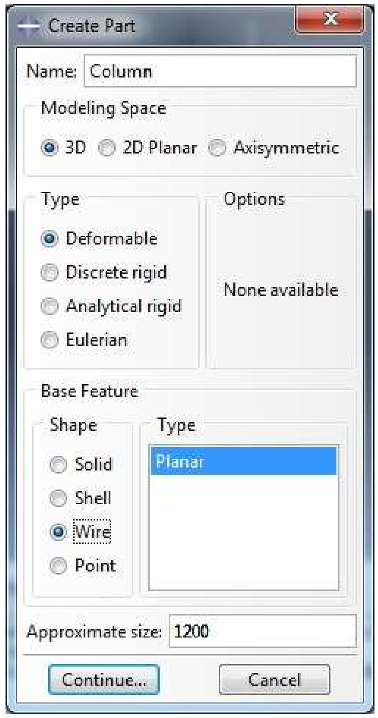


Figure 14. The Wire Beam Creation

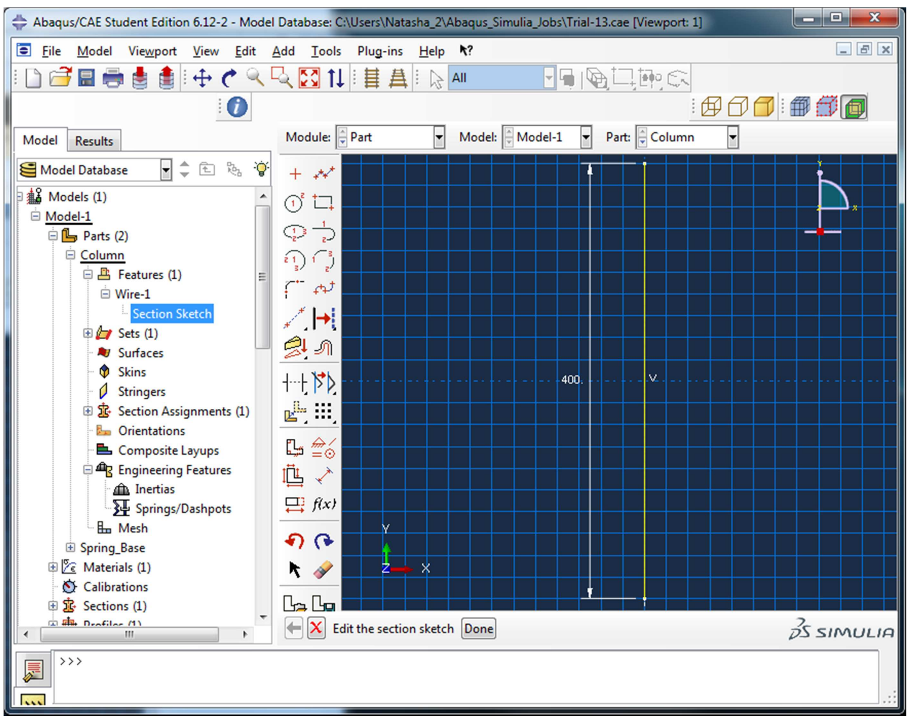


Figure 15. The Column Length Arrangement

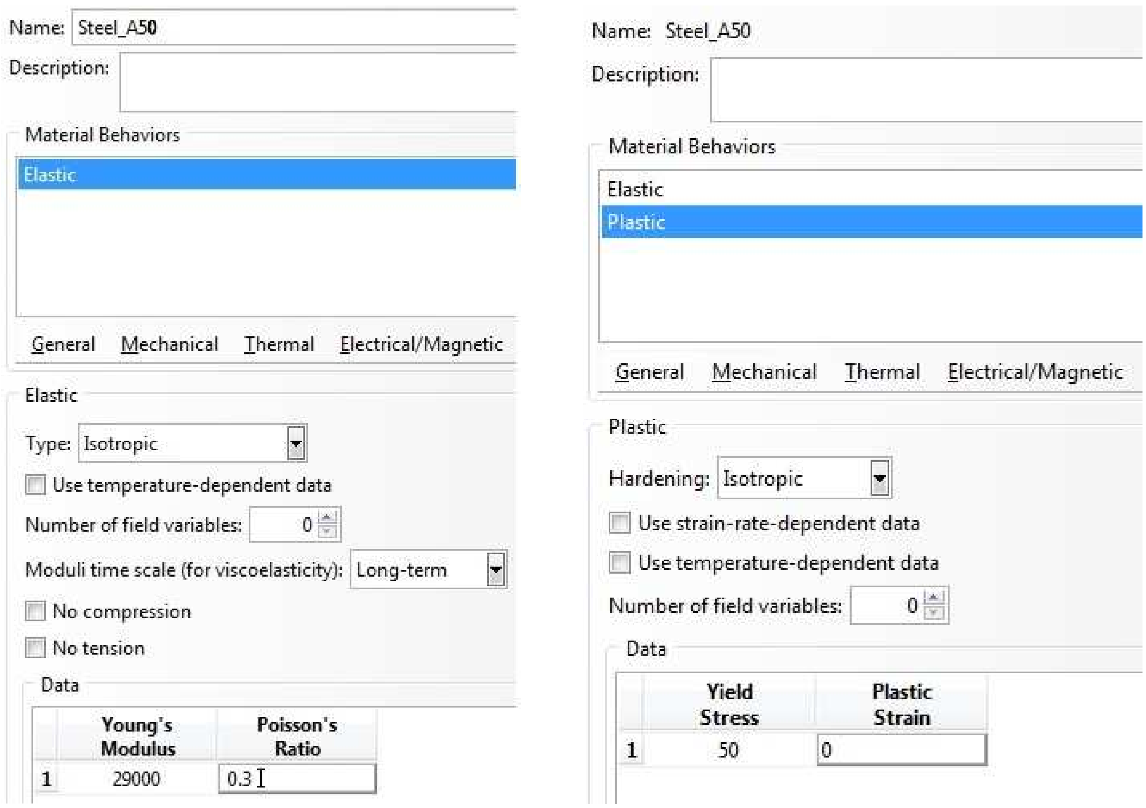


Figure 16. Material Properties Assignment

Name the material and go to Mechanical Properties. First, choose Elasticity and Elastic; fill in the correct constants for steel. Then choose Plasticity and Plastic; use an appropriate constant for yielding stress. Check with Figure 16.

3. Create Cross-Section Geometry - Standard W14x145 Shape.
   Choose the Create Profile icon or double click the Profiles title. Name the profile according to the cross-section geometry and select the I shape to continue. Fill in the shape dimensions according to Figure 17.
   Now it is necessary to create a section that has properties of steel and geometry of the recently created W-shape. This section will be assigned to the wire beam to complete column constructing. Using the appropriate icon or the Sections title create a section for the beam. The needed parameters should already be preselected. Check with Figure 18.
   Click the icon Assign Section or choose Assign at the top menu panel and then pick the Section tag. To select the region for the section assignment click on the vertical line that represents the

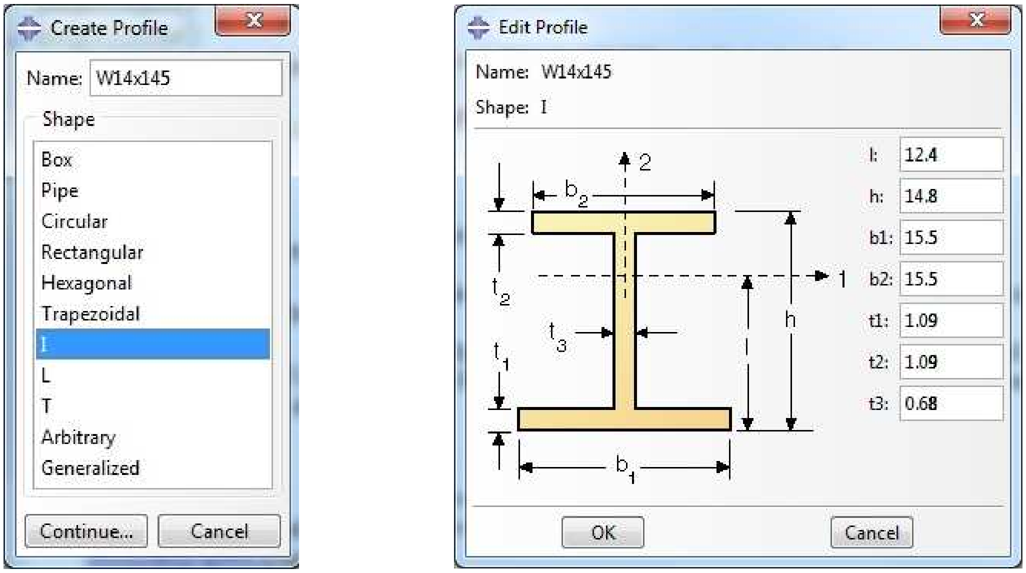


Figure 17. Cross-Section Geometry Assignment

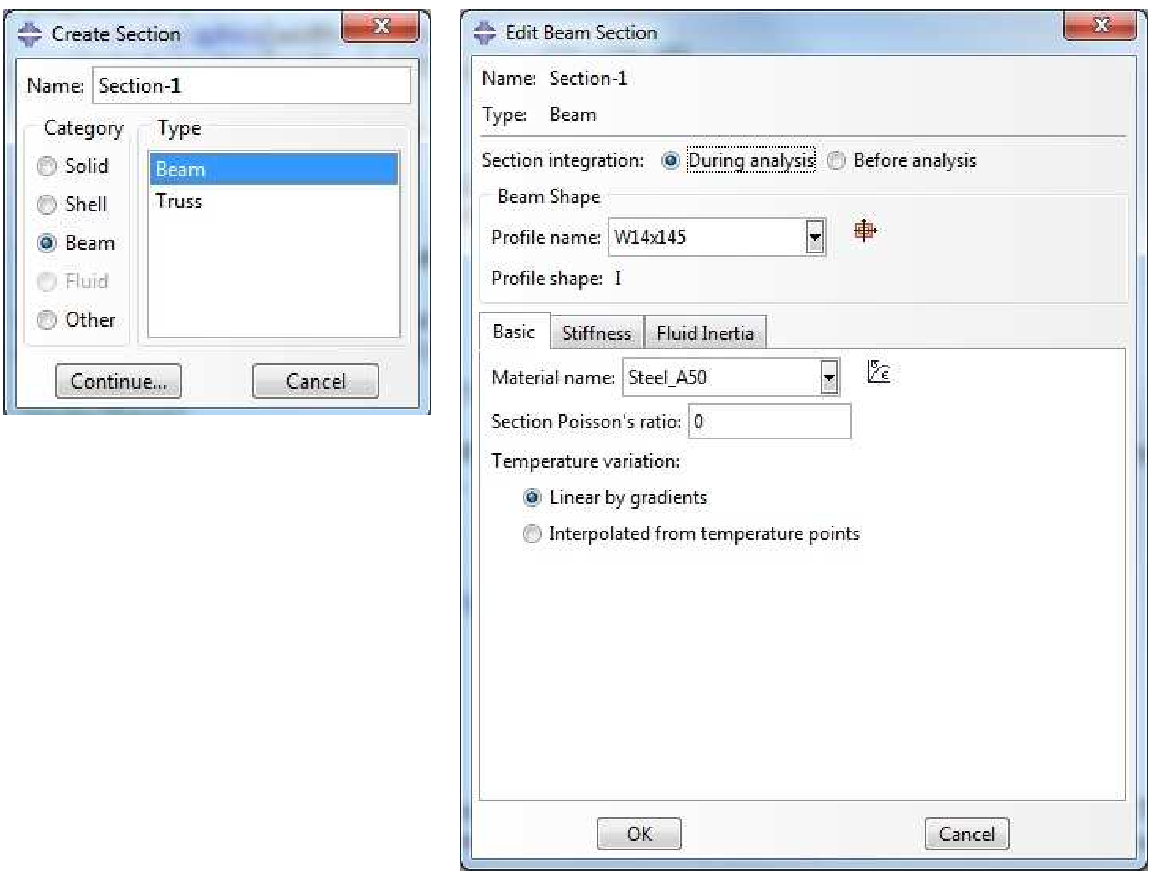


Figure 18. Section Creating

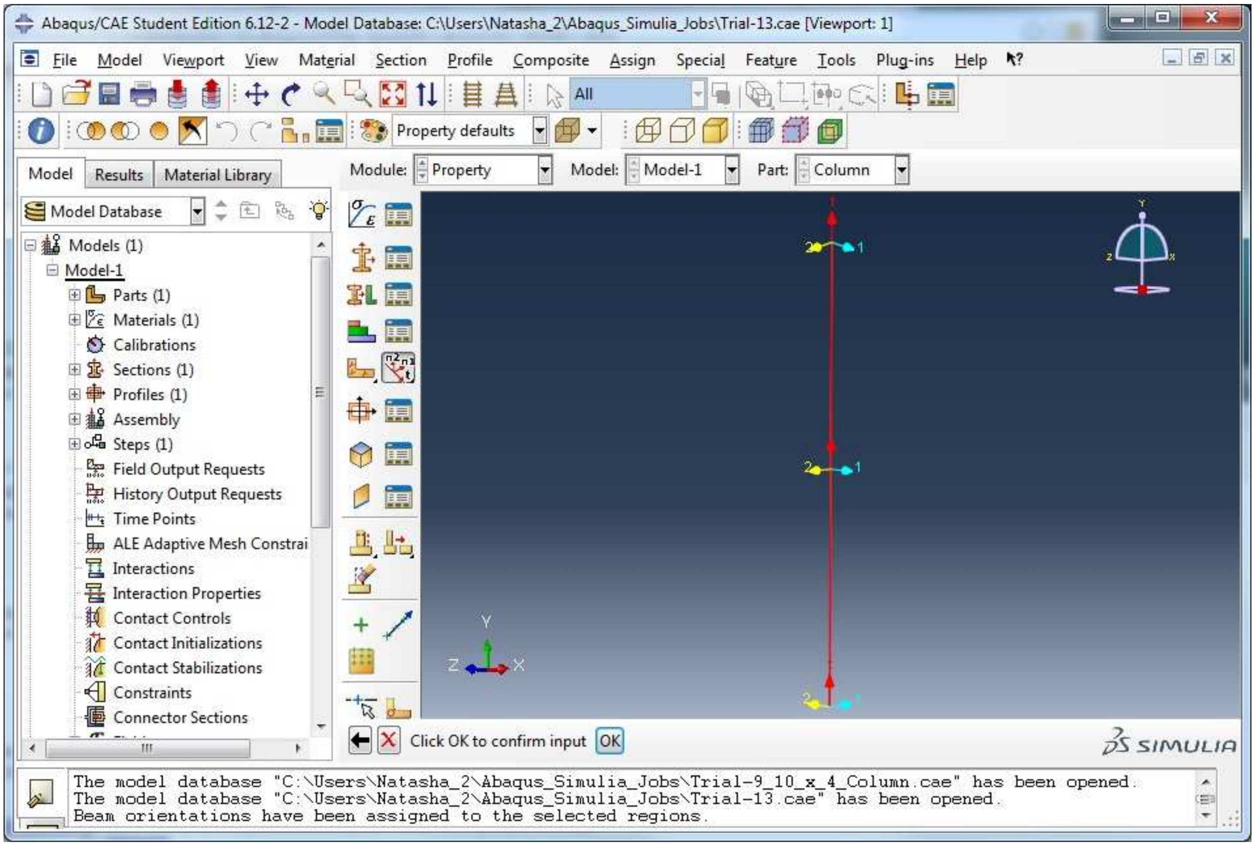


Figure 19. Orientation Assignment

column. Then hit the Done button and the assignment will be complete. The column should change its color.

It is also important to assign the orientation of the wire beam alias the column. Click the Assign Beam Orientation icon or go through the Assign tag at the top menu. Select the whole beam and then leave default value (0; 0; -1) for n1 vector. This value should be appropriate since the z-axis lays in the horizontal plane, perpendicular to the beam axis. And n1 belongs to the cross-section plane (see Figure 17) and so n1 is supposed to be set perpendicular to the beam axis. According to the Abaqus default settings, n1 represents the strong axis for the W-Shape cross-section and n2 represents the weak axis correspondingly. Since the column buckles first around its weak axis, the buckling will take place around the n2 axis which happens to be the x-axis in this case.

4. Assign the Mesh - Computational Network along the Column.

The mesh size can be determined according to the cross-sectional dimensions. The smaller

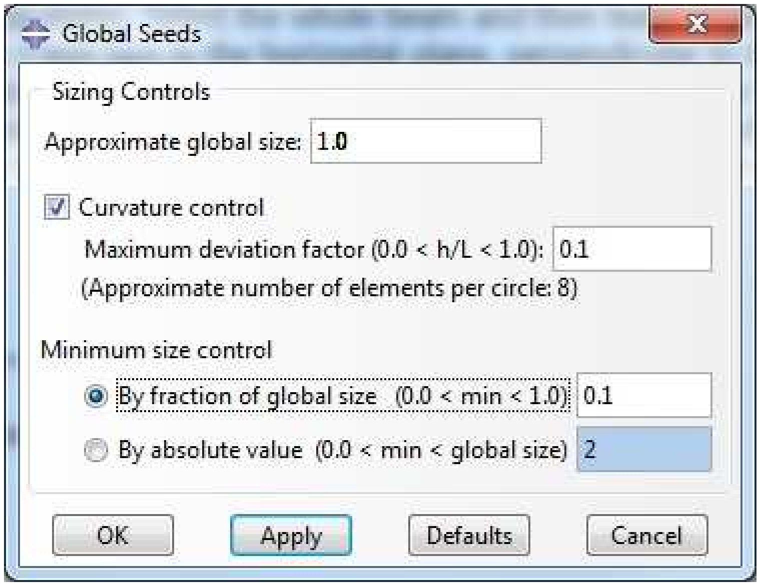


Figure 20. Mesh Size Assignment

dimension is the web width; it equals 0.68 in. For the beginning, the size of the mesh element should not be twice larger than this smaller dimension so it should not exceed 1.36 in. For example, the mesh size can be chosen as big as 1 in.

Set the Mesh option for the Module window. Click on the Seed Part icon or pick Seed and then Part at the top menu. Fill in the dialog placing 1.0 for an Approximate global size, see Figure 20. Then click on the Mesh Part icon or go through the Mesh tag at the top menu panel. The part meshing is complete.

5. Part 2 Should Be Created at the Next Step.

   At the Part module choose the Create Part option and then follow Figure 21 details. Enter the coordinates of the point as (0; 100; 20) - this is going to be the rigid point constraining the future brace or spring movement. The y-coordinate of the rigid point, RP, depends on the brace position. If the brace alias the spring is attached to the center of the column, then the y-coordinate should be set to zero. If, for example, it is attached to the middle of the top half, the RP coordinates can be (0; 100; 20) where 100 is at 3/4 of the column length because the column is located from -200 to 200 along y-axis. The x-coordinate must be zero while the z-coordinate can be chosen arbitrary.

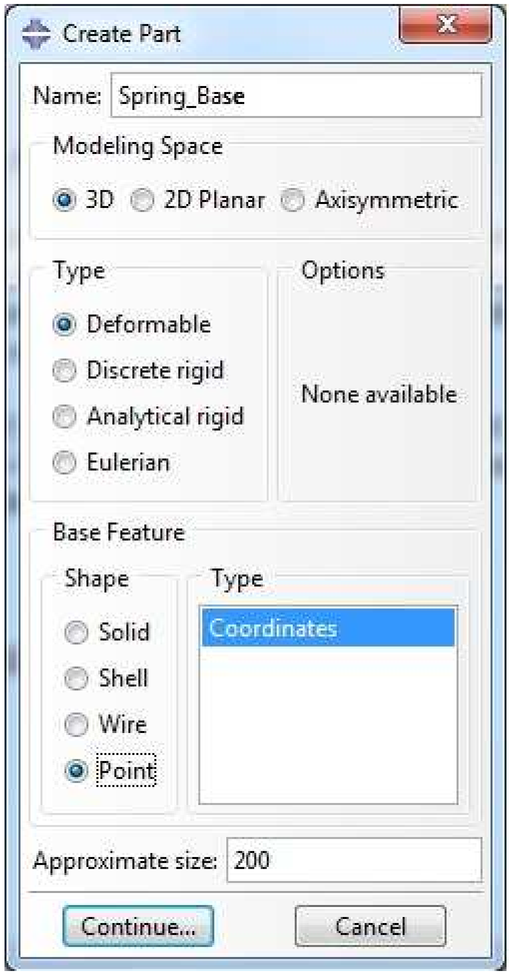


Figure 21. Spring Rigid Base Creation.

6. Combine the Parts into Assembly and Create Connecting Spring between the Parts.
   Set Assembly at the Module window. Hit Instance and then Create at the top panel menu or open Assembly branch in the tree menu at the left part of the screen and then double click on the Instances title. In the appearing dialog choose both parts and hit OK, Figure 22. The two parts Assembly has been created.
   The next step is connecting of the Column and the RP (Spring Base). The picture can be rotated to place the RP to the screen plane and to make the z-axis visible, Figure 24.
   Also it is necessary to mark a point on the column where the spring will be attached so a one point set should be created through the assigning a Node Set. In the Assembly branch at the left part menu double click on the Sets title. Name the Set and choose the Node type, Figure 23.

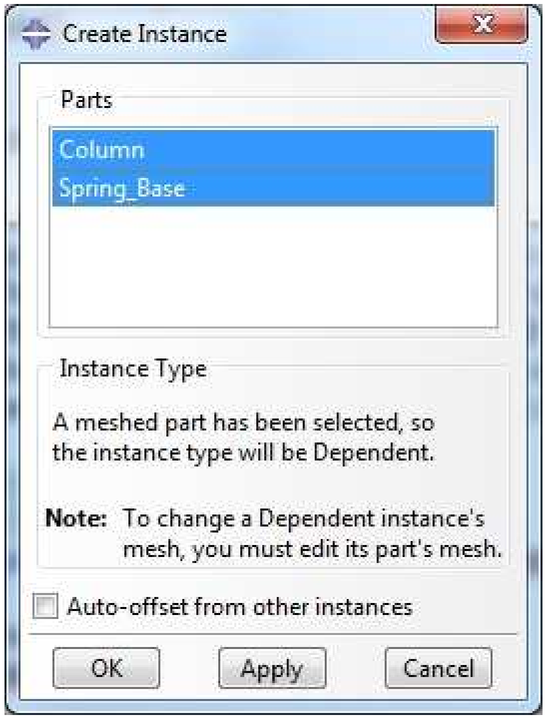


Figure 22. Assembly Creation

Then select any node closed to RP; later the node number can be corrected in the executing input file.

Finally, create the spring connections. Open the Engineering Features branch in the tree menu and double click on the Springs/Dashpots title. Name the connection and choose the Connect Two Points option, Figure 23. Then select RP and hit the Sets button to select Column Node 3/4 point. One more dialog will appear where the spring stiffness should be assigned. For this example, spring stiffness equal to 50 will represent a steel brace sufficient to support the column, Figure 23.

7. Create Steps to Apply Calculating Techniques.

   Before the problem is finalized the calculation methods should be determined. The main approach for the buckling analysis is developed through the Riks Method which should be chosen as a second step of this calculation process. However, it is necessary to use Linear Perturbation Analysis as the first step. It allows receiving the node positions of the crooked column since the column obtains a half of a sine wave shape as result of the Linear Perturbation first mode analysis. This step should be implemented as the preliminary analysis without brace

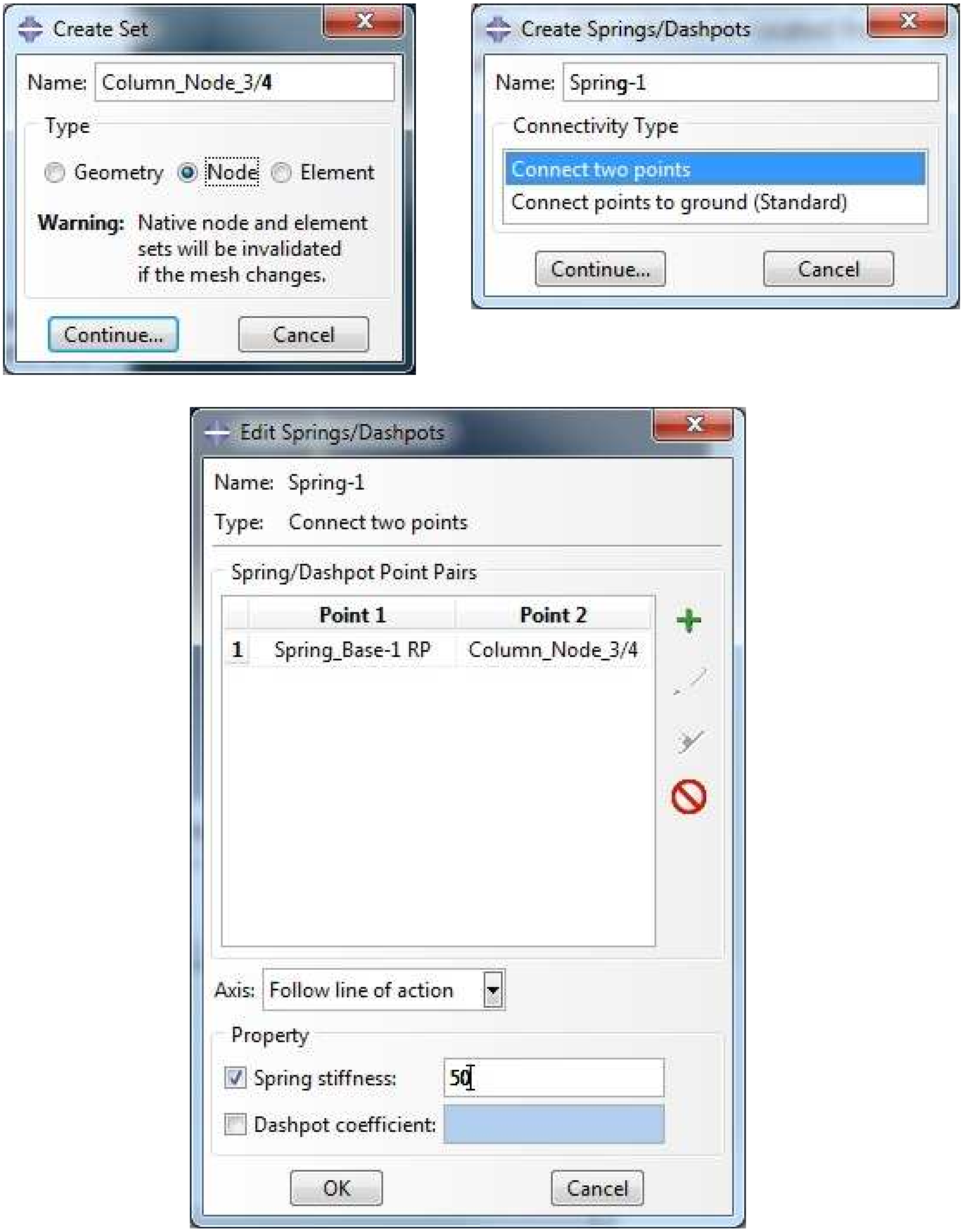


Figure 23. Node Set and Spring Assignment

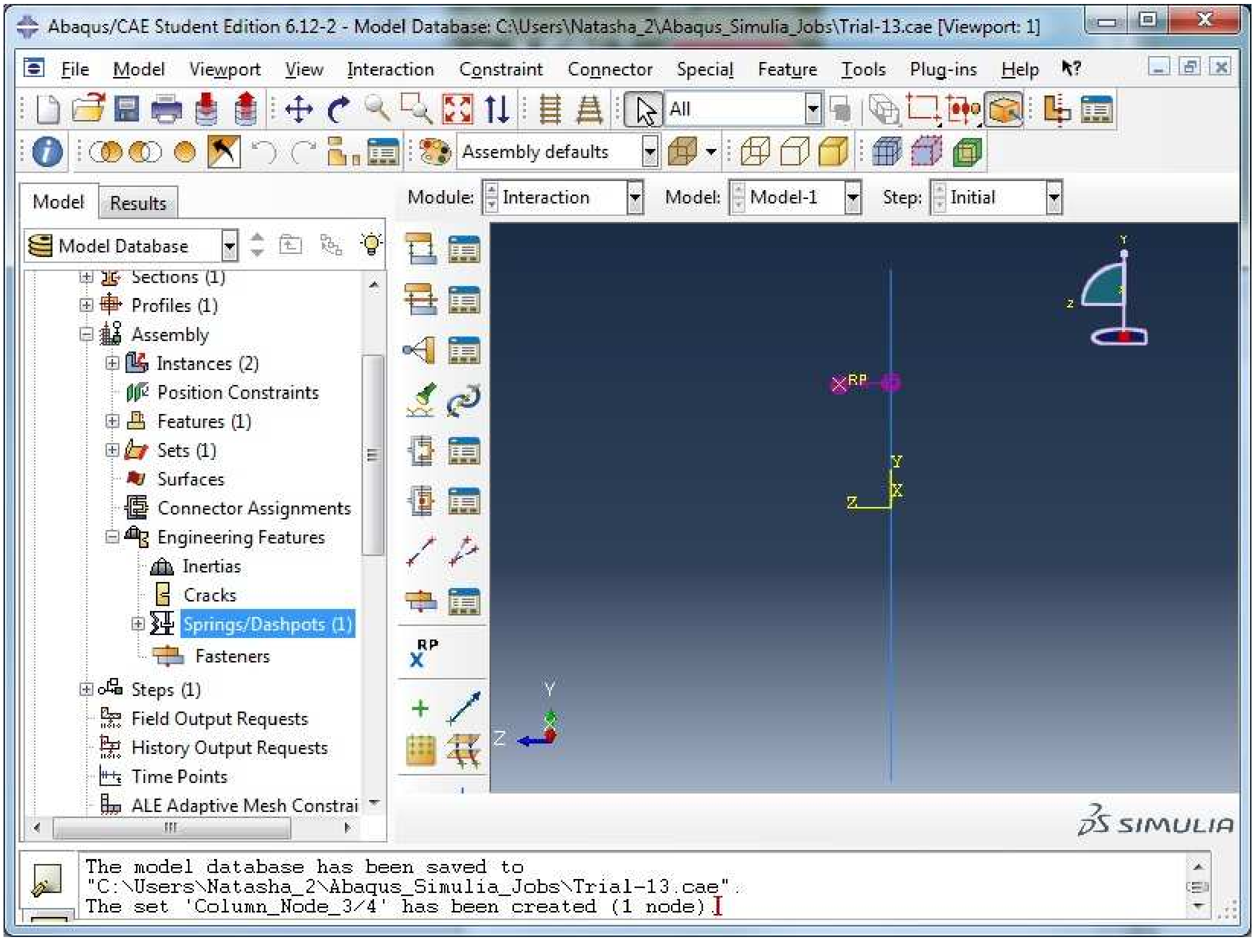


Figure 24. Final Assembly

restriction or with the spring stiffness equals to zero. Later, during the main Riks Analysis, node positions of the crooked column will be used as initial imperfection if multiplied by a reducing factor.

Choose the Step option at the Module window. Then click the Create Step icon or select the same actions through the top or left side menus. Follow Figure 25 details. The number of requested eigenvalues can be equal to or larger than 1.

The next step should be completed according to the Figure 26. It is necessary to turn on Non-Linear Geometry because a large displacement is expected for the buckling analysis. The number of Increments can also be changed from 100 to 50 if the Incrementation tag is chosen; see the Edit Step dialog.

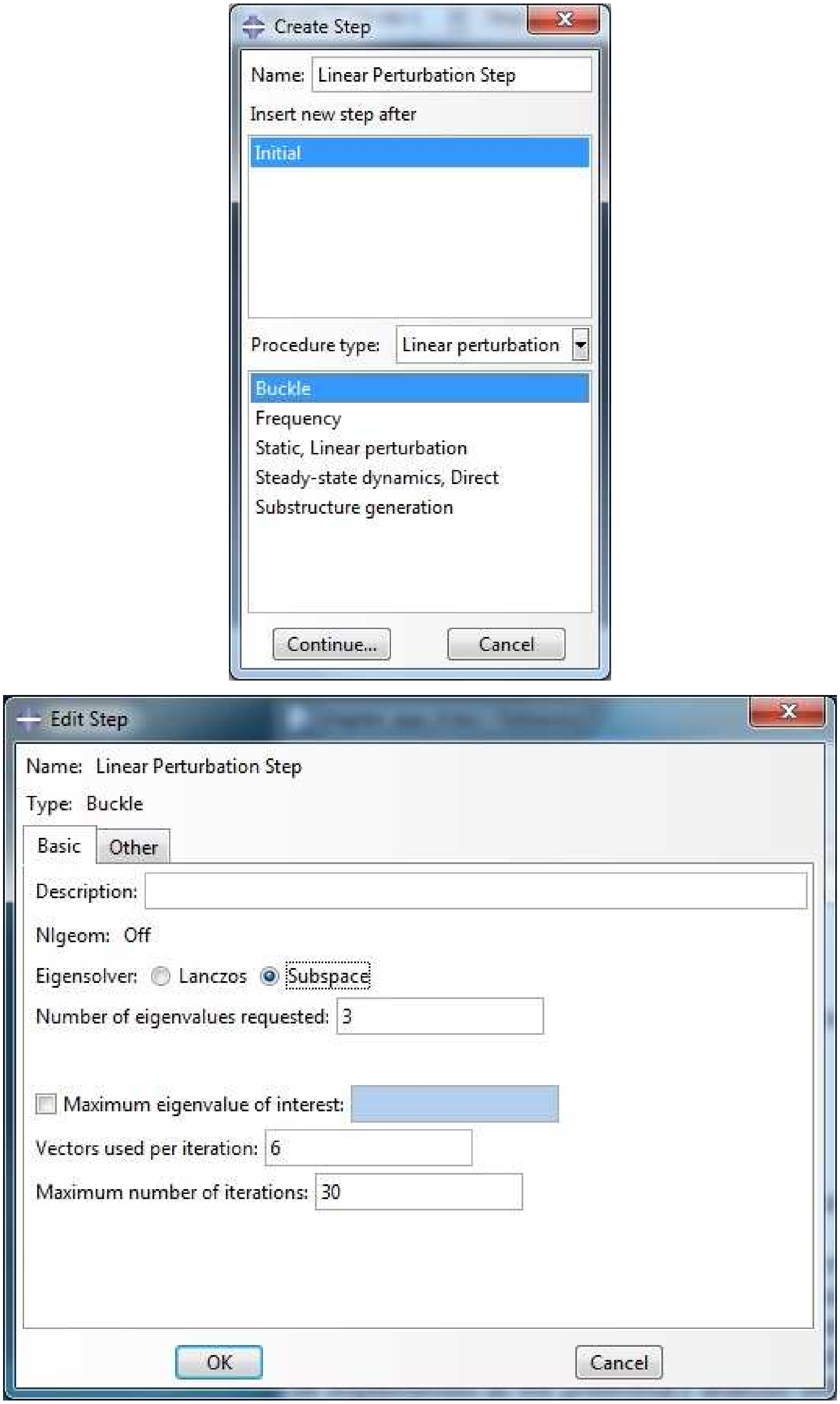


Figure 25. Linear Perturbation Step Assignment

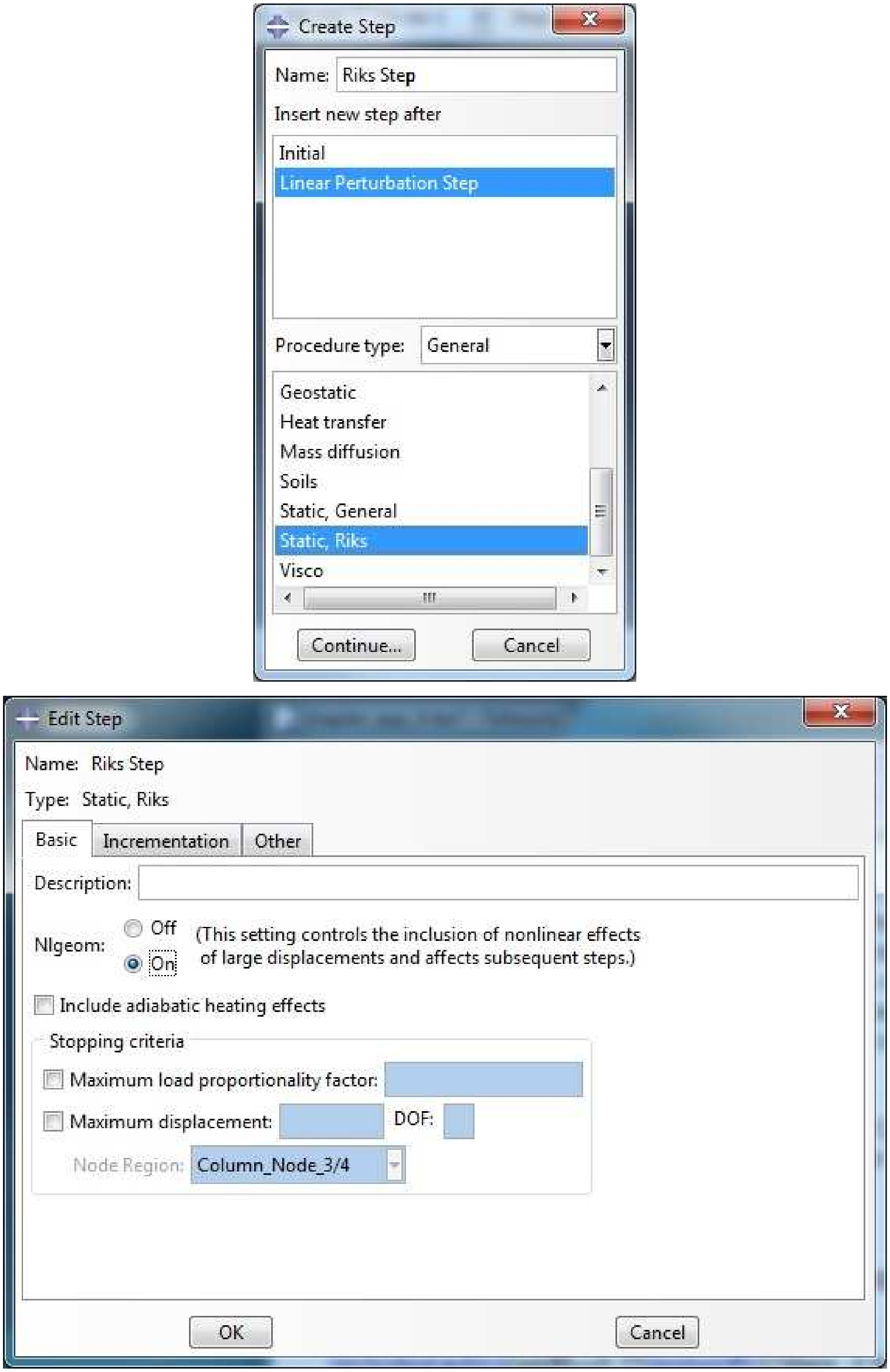


Figure 26. Riks Step Assignment

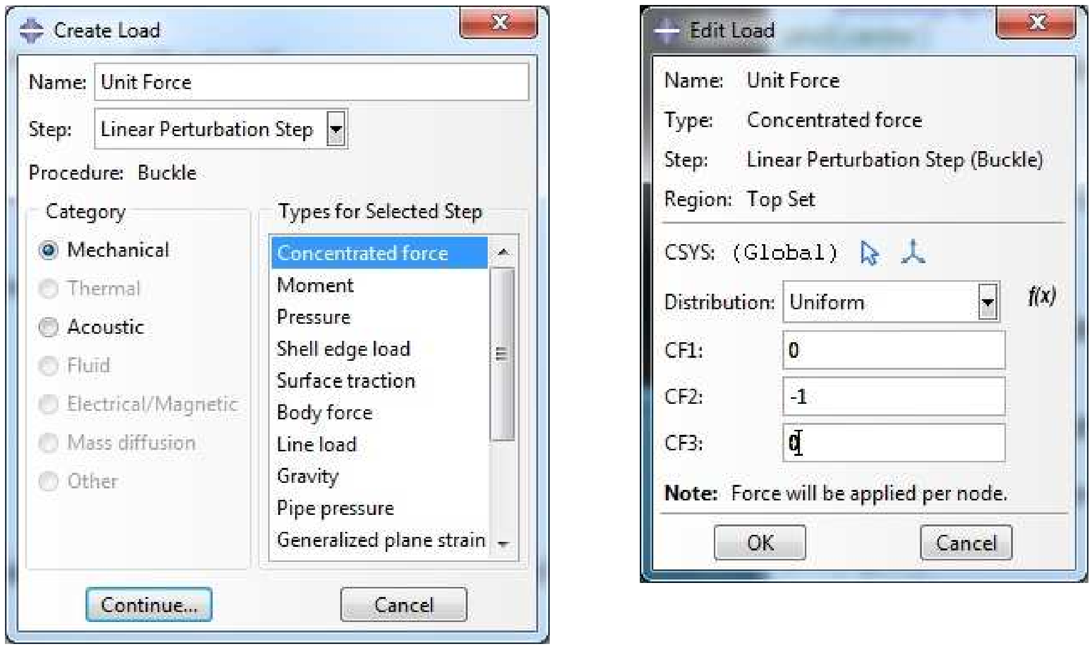


Figure 27. Vertical Unit Load Assignment

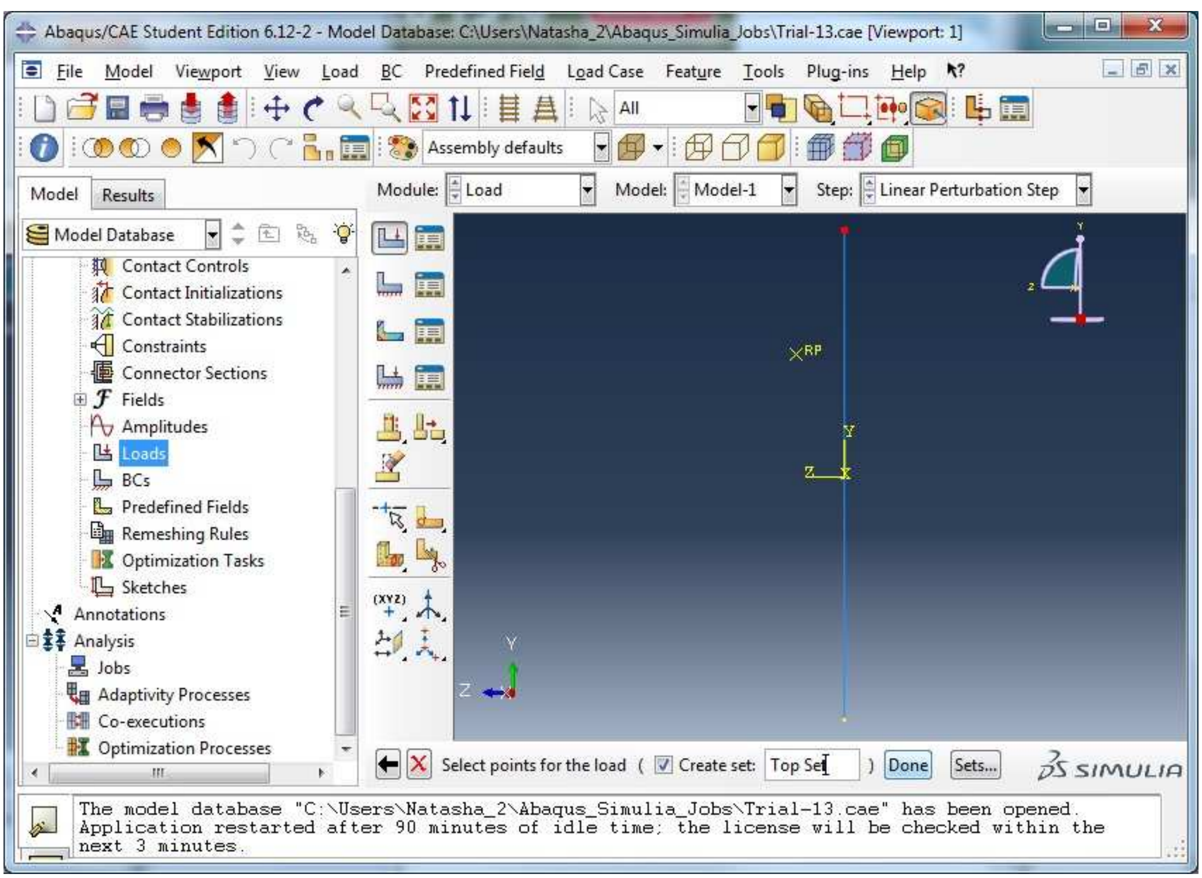


Figure 28 Load Applied at the Top Set

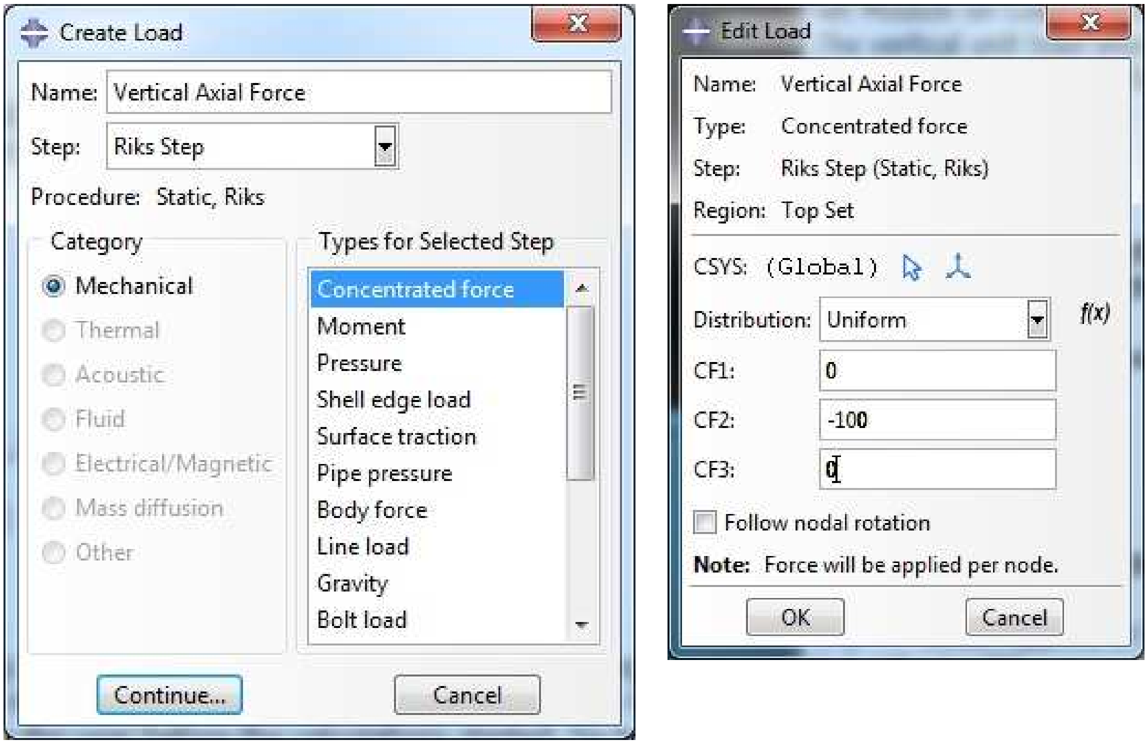


Figure 29. Load Assigned for the Riks Step

8. Assign Load and Boundary Conditions.

   Set Module on Load and then click the Create Load icon. The same dialog can be received through the top or left side menus. The vertical unit load should be assigned for the Linear Perturbation Step. Select the very top point (marked red) to apply the force. At the same time Top Point Set can be created. Check with the Figures 27 and 28.

   A relatively high load should be applied for the Riks Step. It can be near expected critical buckling load but not necessarily. Abaqus will find the load - deflection response of the model where the load increases from zero to the max critical value which can be larger or smaller than the load assigned initially. So begin creating the next load; follow Figure 29 details.

   Boundary conditions can be created if any of the BC tags are clicked. Choose the pinned condition for the bottom point of the column, Figure 31. The Bottom Point Set can be created simultaneously. Select the roller condition for the top of the column, Figure 30. And finally create pinned condition for the Reference Point, RP, which is the spring base.

   It is necessary also to prevent torsion during the Linear Perturbation Step. Normally lateral

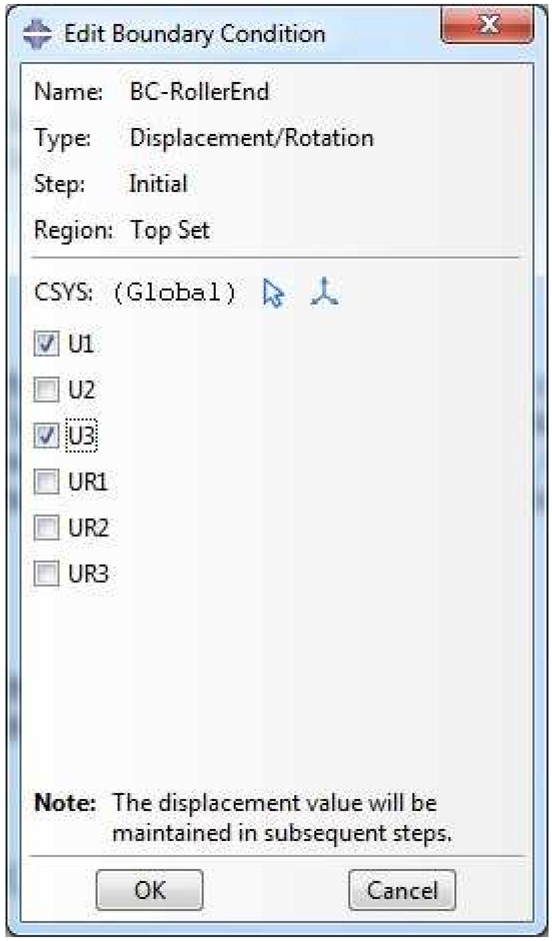


Figure 30. Boundary Conditions at the Top of the Column

buckling phenomena appear before torsion but this is not true for the Linear Elastic Theory. For short span columns critical buckling load can be very high so torsion can happen earlier and the shape of the deformed column will not be related to a half sine wave in this case. So one more boundary condition applied to the whole beam should be assigned for the Linear Perturbation Step, Figure 32. The same boundary condition can be propagated to the Riks Step if only lateral buckling is under the control for this study. The total list of the boundary conditions can be checked after clicking on BC Manager. It should look like the list at Figure 33.

9. Create the Job to Begin Calculation.
   Select the Job option at the Module window or double click on Jobs title at the left side menu tree. Name the job, hit the Continue button and then hit Done at the next dialog, Figure 34. Then the created job should be submitted for executing. When the process is finished (completed or

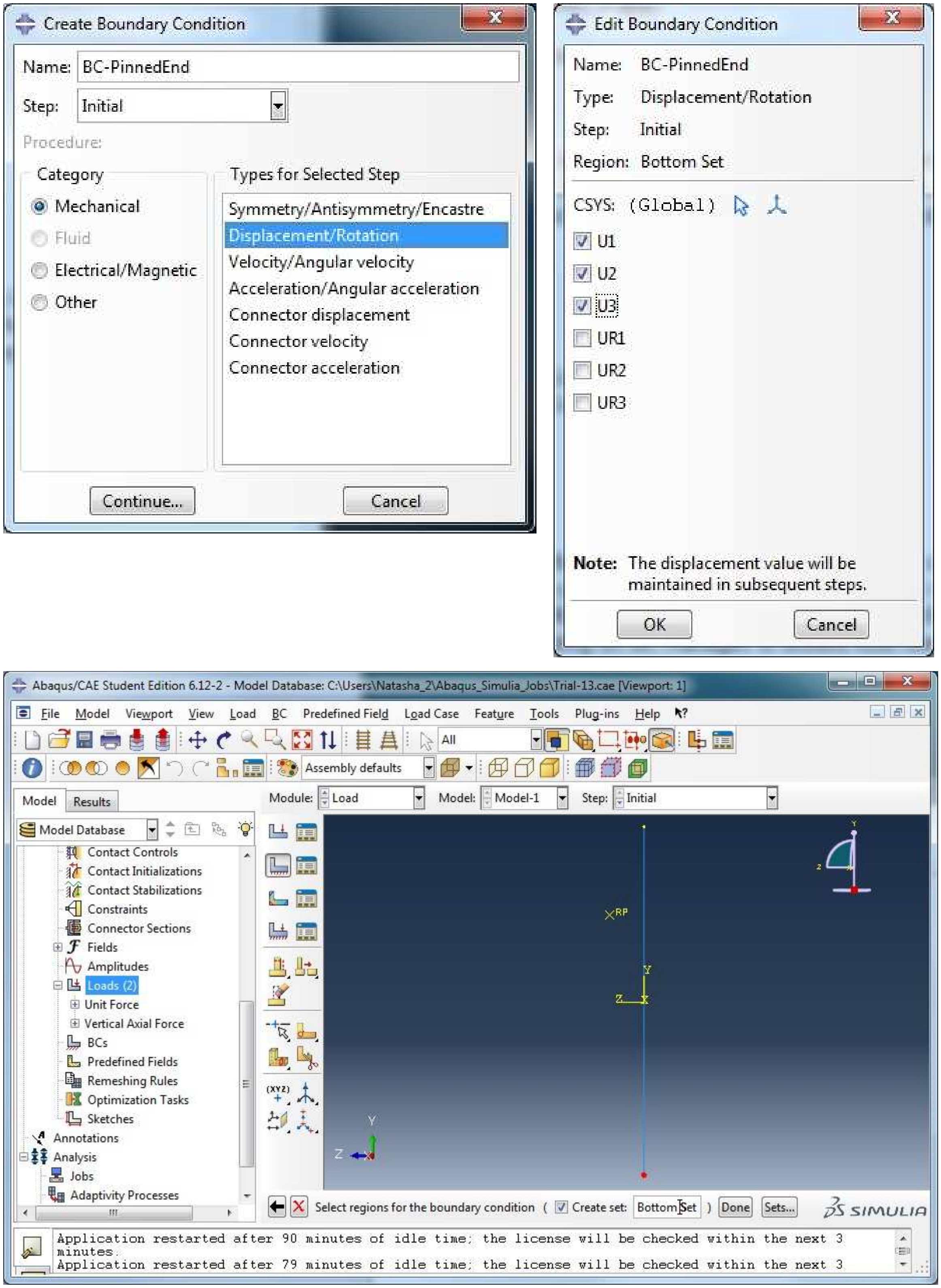


Figure 31. Boundary Conditions at the Bottom of the Column.

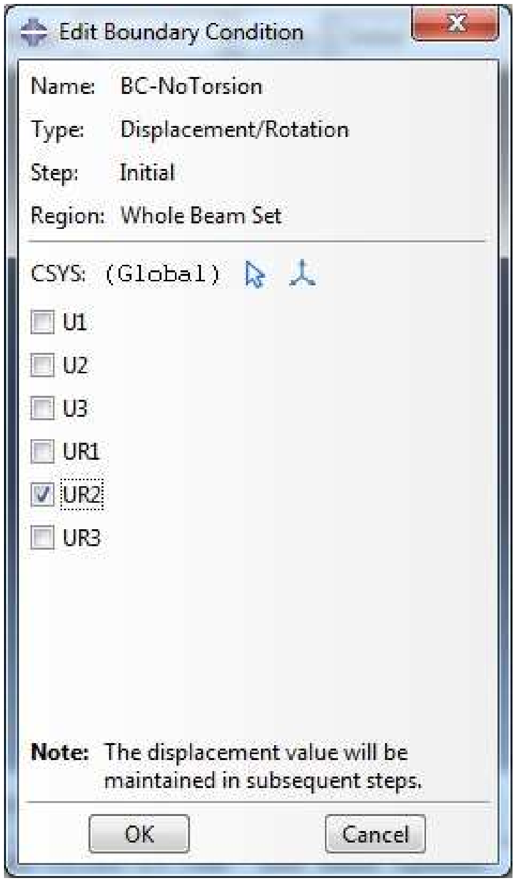


Figure 32. Boundary Conditions Preventing Column Torsion during Linear Preliminary Step

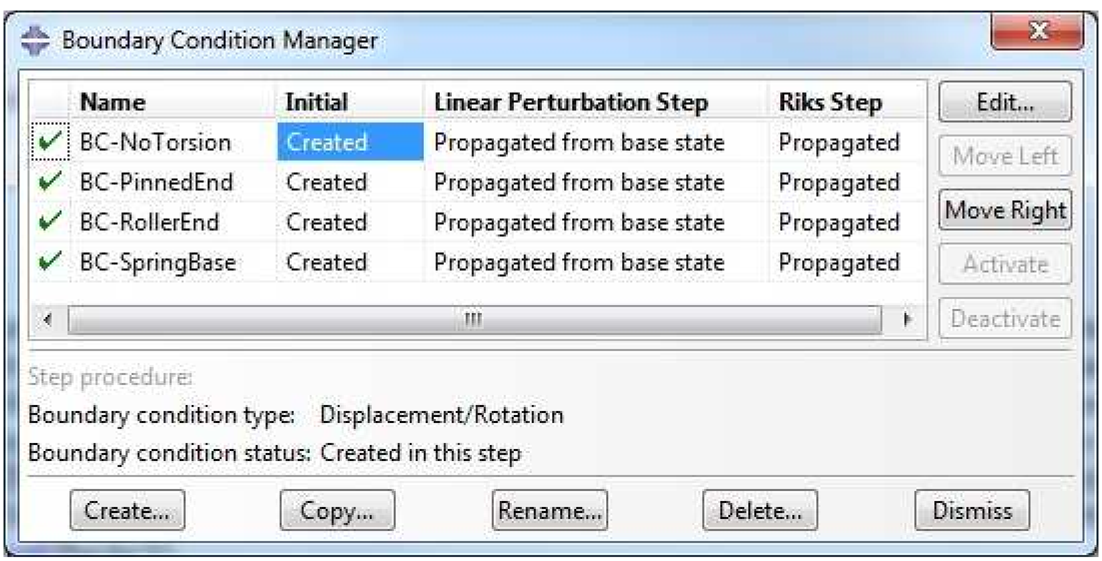


Figure 33. All Boundary Conditions Applied

aborted due to errors), the results can be seen on the screen: select Job and then Results at the top menu panel, for example. To check deflection and curved shape of the column choose U and U3 for Primary variables at the left-top corner of the screen. To switch between calculation modes select Result and then the Step/Frame option at the top menu panel.

Abaqus/CAE module does not have enough tools yet for models improvement. So some changes to the model can be done by editing its executing file: Job-1-Preliminary.inp. Open this file using a text editor and make some corrections. First, determine which node the spring is attached at. If it is not 301, it should be corrected to Column-1.301. The node number can be seen at the following lines of the input file:

```
*Element, type=SpringA, elset=Spring-1-spring
1, Spring_Base-1.1, Column-1.301
```

Also the spring stiffness should be neglected for the next Preliminary trial. If there is no spring, the brace does not influence the buckling process and so the shape of the crooked column will be perfectly corresponded to half of a sine wave. Make sure that the spring stiffness is set to zero:

```
*Spring, elset=Spring-1-spring
0.0
```

It is necessary to save the node positions when the column is deformed and has the half sine wave shape. That is why Output Request for the Linear Perturbation Step should be modified the following way:

```
** OUTPUT REQUESTS
*NODE FILE
U
*Restart, write, frequency=0
```

Later node positions will be used to create initial imperfection of the column but now the new trial has to be run. Save the corrected file as Job-2-Preliminary.inp and create the new Job based on this file instead of the model; choose the Input File option in the Source window for this purpose. Submit this new Job and run it. Riks Step may have conversion problems because

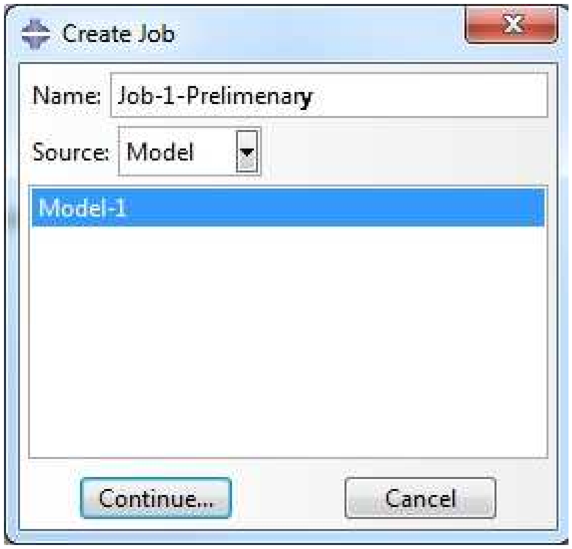


Figure 34. Creating a New Job Based on the Model

a perfectly straight column will be compressed but hardly can lose its stability. A little imperfection is required to provoke the lateral buckling.

Add Imperfection to the Column and Receive Buckling Results.
Now the input file should be modified again. It will use the nodes positions requested before. So these new lines should be inserted after Material Properties:

```
** MATERIALS
*Material, name=Steel
*Elastic
29000., 0.3
*Plastic
50.,0.
*IMPERFECTION, FILE=Job-2-Preliminary, STEP=1
1, 0.4
```

Node Request can be deleted or ignored now. But spring stiffness is a very important parameter for further analysis and it should be set back to 50. Save the corrected file as Job-3-Main-

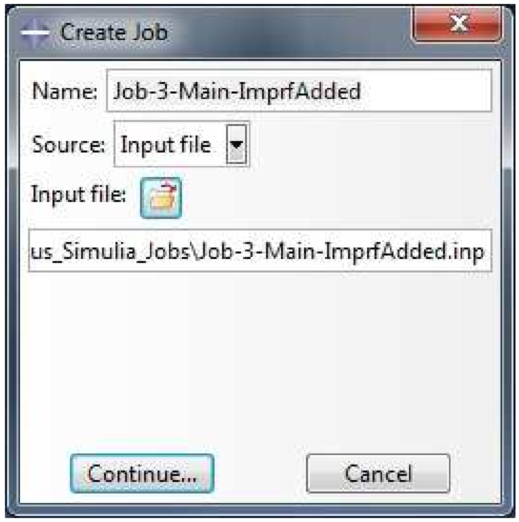


Figure 35. Creating the Main Job

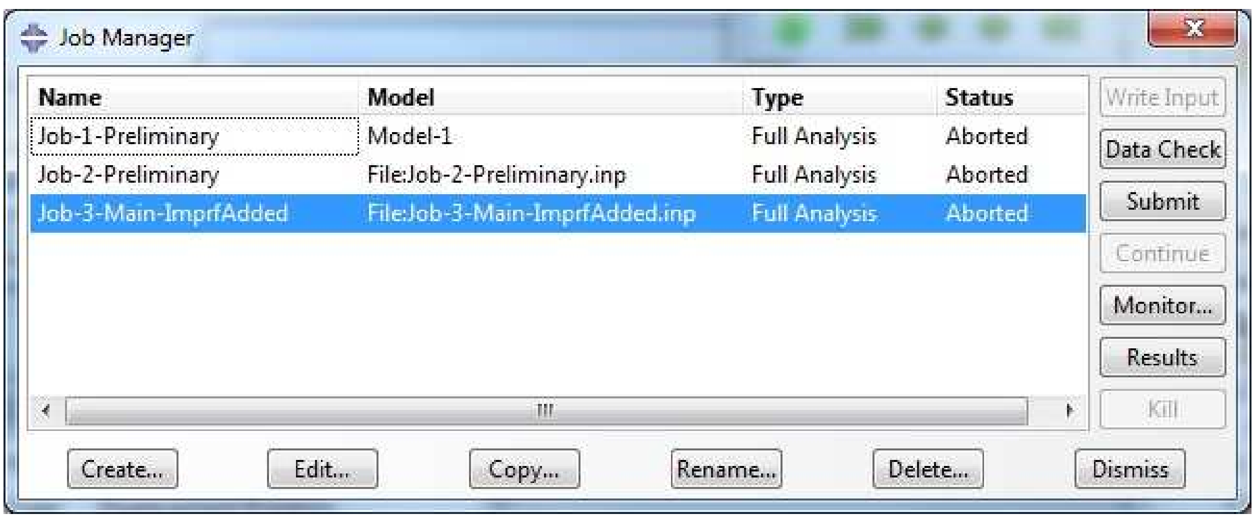
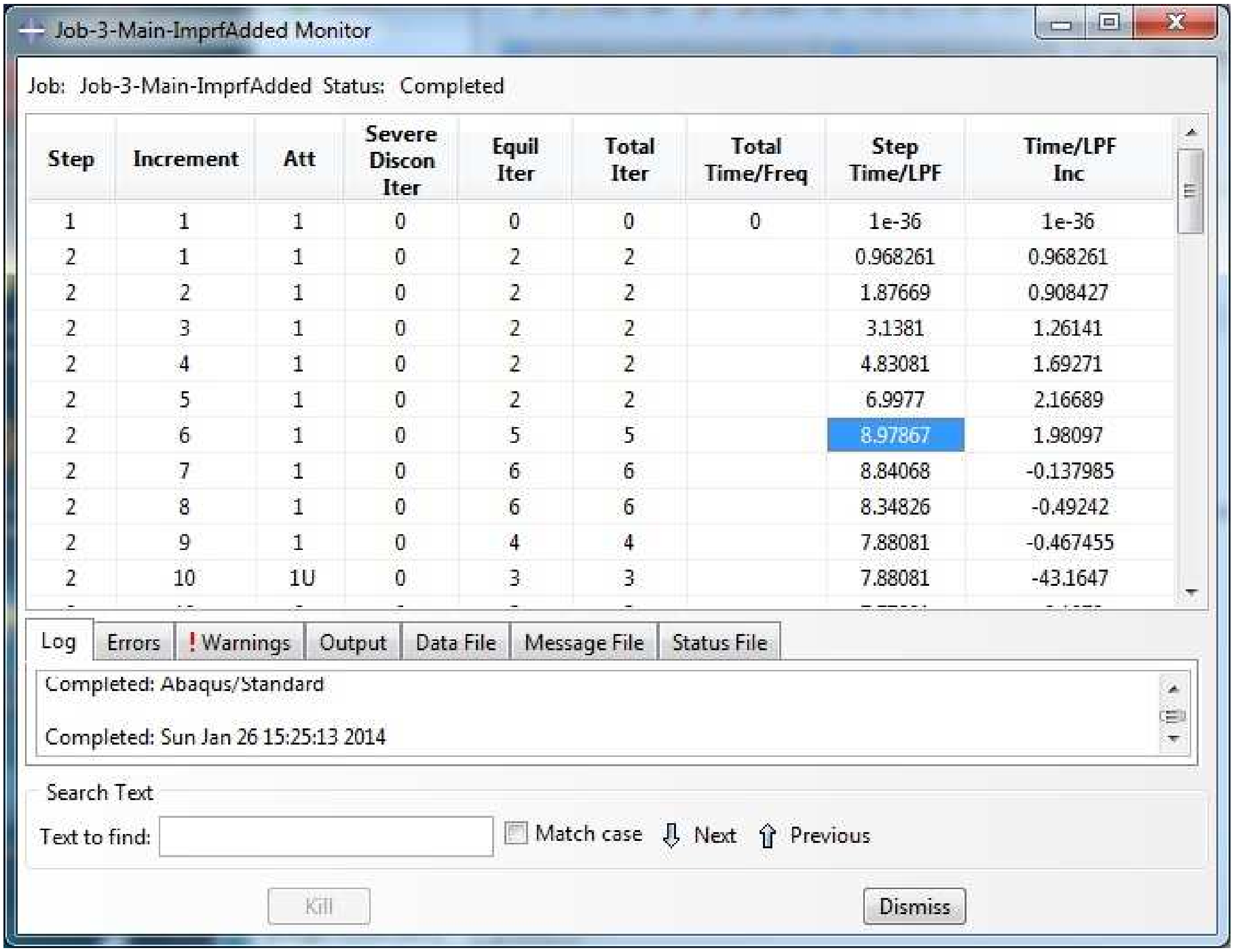


Figure 36. Job Manager

ImprfAdded.inp and create the new job based on this file, Figure 35. Submit the Job and check the results.

The results can be seen in two ways. First, the critical buckling load that is determined by the Riks Method should be checked. Open Job Manager, Figure 36, and click Monitor on the right panel to view the calculation process step by step. At Figure 37 two the last columns represent absolute values of Force incrementation. The column second from the right column, Step Time/LPF, shows the portion of the initially assigned load taken at the each step. For example, in



Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
1	1	1	0	0	0	0	1e-36	1e-36
2	1	1	0	2	2		0.968261	0.968261
2	2	1	0	2	2		1.87669	0.908427
2	3	1	0	2	2		3.1381	1.26141
2	4	1	0	2	2		4.83081	1.69271
2	5	1	0	2	2		6.9977	2.16689
2	6	1	0	5	5		8.97867	1.98097
2	7	1	0	6	6		8.84068	-0.137985
2	8	1	0	6	6		8.34826	-0.49242
2	9	1	0	4	4		7.88081	-0.467455
2	10	1U	0	3	3		7.88081	-43.1647

Figure 37. Critical Buckling Load Check

our case the load equaled to 100 lb was assigned for the Riks Analysis. According to the Monitor Table at the Increment 2 the load taken for the analysis is 0.968261 x 100 = 96.8261 lb. Then the load keeps increasing until the Increment 6, where the load reaches its max: 897.867 lb. This value is indeed a critical buckling load because at the next increment the arc length of the load-displacement diagram is increasing so the displacement value should grow while the load value will decrease as it can be seen in the table of Figure 37.

The second way how the absolute values of lateral displacement can be seen is using the visualization module. At the Job Manager, Figure 36, click the Results button. Then select U and U3 for the Primary variables at the top left corner. Now the final displacement of each column point can be seen on the screen. To switch between analysis steps or between increments values use the arrows above the screen or the Frame Selector located at the same panel.

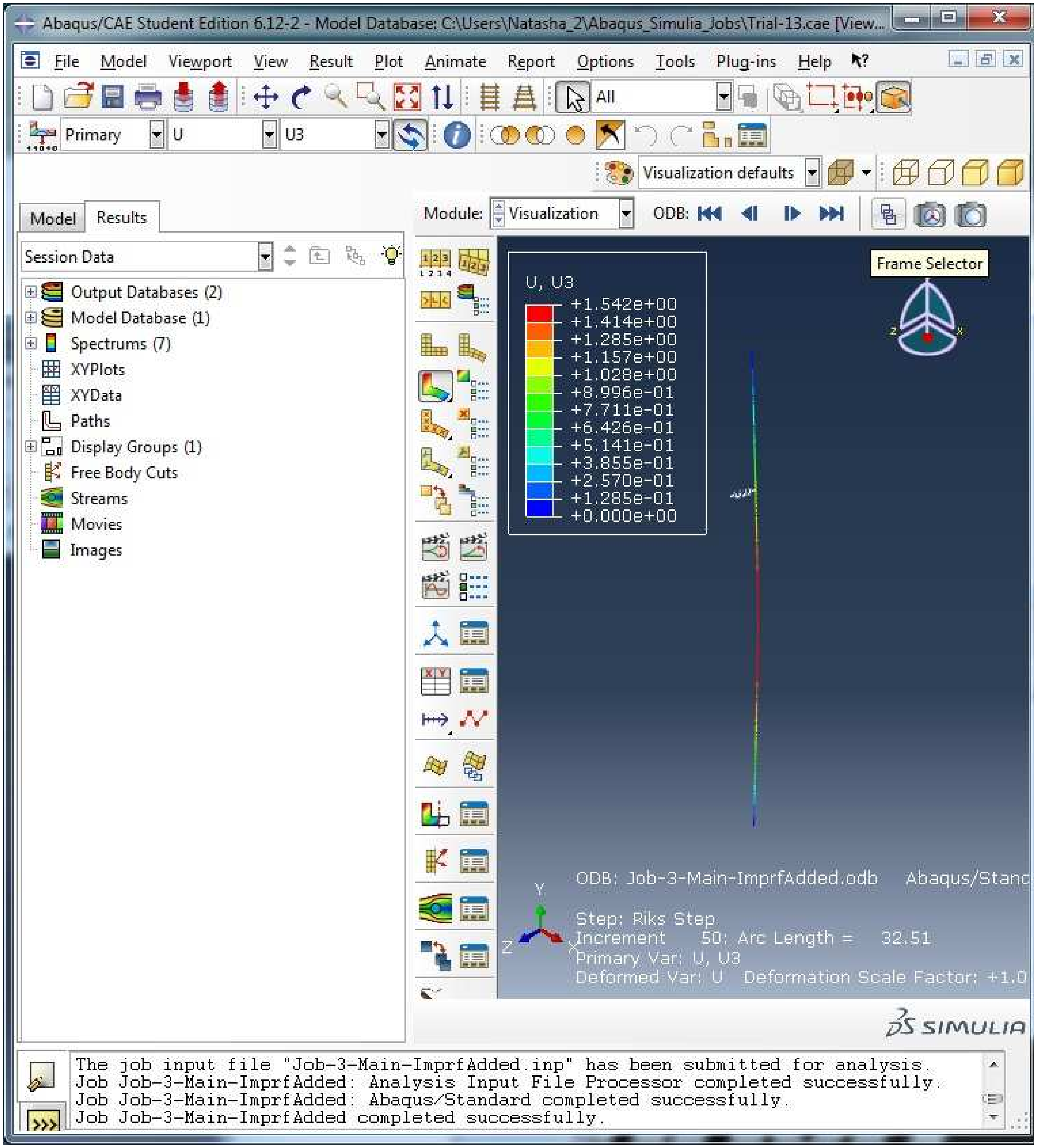


Figure 38. Final Displacement of the Points along the Column

The description above can help a person with any level of Abaqus experience to build the desired model. The model then might be optimized by input file parameterization. An example of such parameterization is displayed below as well as one of the subroutines the author would like to share with the Abaqus users who desire to implement a parametric study of the model.

EXAMPLE OF THE PARAMETRIZATION OF THE ABAQUS INPUT FILE

```
**********************************************************************
*Heading
** Job name: Job-73-Trial-9_640Clmn_Sprs Model name: Model-640-Springs
** Generated by: Abaqus/CAE Student Edition 6.12-2
*Preprint, echo=NO, model=NO, history=NO, contact=NO
**
**********************************************************************
**
*PARAMETER
Length = 640.                        # column length
mesh_size = 4
no_of_elem = int(Length/mesh_size)
no_of_nodes = int(Length/mesh_size) + 1
imperf_amplitude = Length/1000
position = 0.3                      # brace position along the column
L2 = int(position*Length)
L1 = Length - L2
attach_node = int(L1/mesh_size) + 1
F_y = 50.                           # yielding stress
k_br = 1900.                        # brace stiffness
**********************************************************************
**
```

```
** PARTS
**
*Part, name=Part-1
*Node
            1,           0.,        <Length>,           0.
<no_of_nodes>,           0.,         0.,           0.
*NGEN
1, <no_of_nodes>, 1
**
*Element, type=B31
  1,   1,   2
*ELGEN
1, <no_of_elem>
**
*Nset, nset=Set-Whole_Beam, generate
   1,  <no_of_nodes>,    1
*Elset, elset=Set-Whole_Beam, generate
   1,  <no_of_elem>,    1
**
*PARAMETER
W = 12.58
*PARAMETER DEPENDENCE, TABLE=sect_geom, NUMBER VALUES=8
4.0, 8.0, 8.0, 8.0, 0.435, 0.435, 0.285, 8.31
5.1, 10.2, 4.02, 4.02, 0.4, 0.4, 0.25, 10.19
5.05, 10.1, 8.02, 8.02, 0.62, 0.62, 0.35, 10.45
6.1, 12.2, 10.0, 10.0, 0.64, 0.64, 0.36, 12.58
7.4, 14.8, 15.5, 15.5, 1.09, 1.09, 0.68, 14.145
```

```
*PARAMETER, TABLE=sect_geom, DEPENDENT=(center, h, b1, b2, t1, t2,
t3), INDEPENDENT=(W)

**

*Beam Section, elset=Set-Whole_Beam, material=Steel,
temperature=GRADIENTS, section=I

<center>, <h>, <b1>, <b2>, <t1>, <t2>, <t3>

0.,0.,-1.

*End Part

**

*Part, name=Part-2

*Node

      1,            0.,          <L2>,           15.

*Nset, nset=Part-2-RefPt_, internal

1,

*End Part

**

** ASSEMBLY

*Assembly, name=Assembly

**

*Instance, name=Part-1-1, part=Part-1

*End Instance

**

*Instance, name=Part-2-1, part=Part-2

*End Instance

**

*Nset, nset=Set-Top, instance=Part-1-1

 1,
```

```
*Nset, nset=Set-Bottom, instance=Part-1-1
 <no_of_nodes>,
*Nset, nset=Set-MostDefl, instance=Part-1-1
 9,
*Nset, nset=Set_RigidPt, internal, instance=Part-2-1
 1,
*Spring, elset=Springs/Dashpots-1-spring

<k_br>
*Element, type=SpringA, elset=Springs/Dashpots-1-spring
1, Part-2-1.1, Part-1-1.<attach_node>
*End Assembly
**
** MATERIALS
**
*Material, name=Steel
*Elastic
29000., 0.3
*Plastic
<F_y>, 0.
**
*IMPERFECTION, FILE=W14_L640_MshSz2_A50, STEP=1
1, <imperf_amplitude>
**
** BOUNDARY CONDITIONS
**
** Name: BC-Bottom Type: Displacement/Rotation
```

```
*Boundary
Set-Bottom, 1, 1
Set-Bottom, 2, 2
Set-Bottom, 3, 3
Set-Bottom, 5, 5
** Name: BC-RP-Fixed Type: Displacement/Rotation
*Boundary
Set_RigidPt, 1, 1
Set_RigidPt, 2, 2
Set_RigidPt, 3, 3
** Name: BC-Top Type: Displacement/Rotation
*Boundary
Set-Top, 1, 1
Set-Top, 3, 3
Set-Top, 5, 5
** Name: BC-WholeBeam Type: Displacement/Rotation
*Boundary
Part-1-1.Set-Whole_Beam, 5, 5
** ----------------------------------------------------------------
**
** STEP: Linear Perturbn
**
*Step, name="Linear Perturbn", perturbation
*Buckle
4, , 20, 30
**
** BOUNDARY CONDITIONS
```

```
**
** Name: BC-Bottom Type: Displacement/Rotation
*Boundary, op=NEW, load case=1
Set-Bottom, 1, 1
Set-Bottom, 2, 2
Set-Bottom, 3, 3
Set-Bottom, 5, 5
*Boundary, op=NEW, load case=2
Set-Bottom, 1, 1
Set-Bottom, 2, 2
Set-Bottom, 3, 3
Set-Bottom, 5, 5
** Name: BC-RP-Fixed Type: Displacement/Rotation
*Boundary, op=NEW, load case=1
Set_RigidPt, 1, 1
Set_RigidPt, 2, 2
Set_RigidPt, 3, 3
*Boundary, op=NEW, load case=2
Set_RigidPt, 1, 1
Set_RigidPt, 2, 2
Set_RigidPt, 3, 3
** Name: BC-Top Type: Displacement/Rotation
*Boundary, op=NEW, load case=1
Set-Top, 1, 1
Set-Top, 3, 3
Set-Top, 5, 5
*Boundary, op=NEW, load case=2
```

```
Set-Top, 1, 1
Set-Top, 3, 3
Set-Top, 5, 5
** Name: BC-WholeBeam Type: Displacement/Rotation
*Boundary, op=NEW, load case=1
Part-1-1.Set-Whole_Beam, 5, 5
*Boundary, op=NEW, load case=2
Part-1-1.Set-Whole_Beam, 5, 5
**
** LOADS
**
** Name: Vertical_Unit_Load   Type: Concentrated force
*Cload, op=NEW
Set-Top, 2, -1.
**
** OUTPUT REQUESTS
**
*NODE FILE
U
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field, variable=PRESELECT
*End Step
** STEP: Riks
```

```
**
*Step, name=Riks, nlgeom=YES, inc=20
*Static, riks
0.1, 1., 1e-05, 2., ,
**
** LOADS
**
** Name: Critical Load   Type: Concentrated force
*Cload, op=NEW
Set-Top, 2, -400.
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-2
**
*Output, field, variable=PRESELECT
**
** HISTORY OUTPUT: H-Output-1
**Output, history, frequency=1
**node output, nset=Set-MostDefl
** CF3, U3
**
*Output, history, variable=PRESELECT
*End Step
```

EXAMPLE OF THE SUBROUTINE FOR THE PARAMETRIC STUDY

```
######################################################################

import sys, glob
import math
import os.path
from odbAccess import*
from abaqusConstants import*

#List of PARAMETERS
Length = 560.                        # column length
mesh_size = 2
no_of_elem = int(Length/mesh_size)
no_of_nodes = int(Length/mesh_size) + 1
imperf_amplitude = Length/1000
position = 0.3                     # brace position along the column
                                     should be less or equal to 0.5
L2 = position*Length
L1 = Length - L2
attach_node = int(L1/mesh_size) + 1

F_y = 50.                            # yielding stress
W = 14.145                           # cross-section type
r = 3.98
A = 42.7
Load = 400.

lambd = math.sqrt(F_y/29000)*L1/r/math.pi
```

```
if (lambd <= 1.0):
     P_fail = F_y*(1.035 - 0.202*lambd - 0.222*lambd*lambd)*A
elif (lambd <= 2.):
     P_fail = F_y*(-0.111 + 0.636/lambd + 0.087/lambd/lambd)*A
else: P_fail = F_y*(0.009 + 0.877/lambd/lambd)*A

k_br_req = (1+L1/L2)*2*P_fail/0.75/L1
k_br_ideal = 2*P_fail/L1
P_euler = math.pow(math.pi*r/L1, 2)*29000*A
k_br_ideal_euler = 2*P_euler/L1

#k_br = 1900.
k_br_initial = 50*k_br_req
k_br_final = 50*k_br_req
k_br_interval = 1

# CREATE THE STUDY

######################################################################
studyName = '50Kreq_05'
######################################################################
study1 = ParStudy( par=('k_br', 'Length', 'mesh_size', 'position',
                 'imperf_amplitude', 'F_y', 'W'), name=studyName )

study1.define('CONTINUOUS', par='k_br', domain=(k_br_initial,
                k_br_final))
study1.sample('INTERVAL', par='k_br', interval=k_br_interval)
```

```
study1.define('CONTINUOUS', par='Length')

study1.sample('VALUES', par='Length', values=Length)

study1.define('CONTINUOUS', par='mesh_size')

study1.sample('VALUES', par='mesh_size', values=mesh_size)

study1.define('CONTINUOUS', par='position')

study1.sample('VALUES', par='position', values=position)

study1.define('CONTINUOUS', par='imperf_amplitude')

study1.sample('VALUES', par='imperf_amplitude',
values=imperf_amplitude)

study1.define('CONTINUOUS', par='F_y')

study1.sample('VALUES', par='F_y', values=F_y)

study1.define('CONTINUOUS', par='W')

study1.sample('VALUES', par='W', values=W)

study1.combine('MESH')

# CHOOSE TEMPLATE AND EXECUTE THE STUDY

########################################################################

templateName = '05_29'

########################################################################

study1.generate(template=templateName)


study1.execute('ALL')


#CREATE FILE FOR RESULTS COLLECTION

out_filename = templateName + studyName + '_results.txt'

# sys.stdout.write( 'Creating file %s\n' % out_filename )

out_file = open( out_filename, 'a' )
```

```
# FIND MAX LPF, its INDEX and corresponding INC VALUE


filenames = glob.glob( templateName + '_' + studyName + '*.sta' )

j = 0


for filename in filenames:

      k_br = k_br_initial + k_br_interval*j

      sys.stdout.write( '-------------------------\n' )

      sys.stdout.write( 'Working on %s\n' % filename )


      # read the file

      lines = [ line.strip() for line in open( filename ).readlines()
][6:-2]

      #for line in lines: print( line )


      # parsing string

      lpf = []

      inc = []

      for line in lines:

            list_line = line.split()

            inc.append( int(list_line[1]) )

            lpf.append( float(list_line[6]) )

      #print( inc )

      #print( lpf )


      # finding max

      lpf_max = 0.0
```

```
    lpf_imax = 0

    lpf_secondmax = 0.0

    for i,e in enumerate( lpf ):

         if e > lpf_max:

              lpf_max = e

              lpf_imax = i

         elif e == lpf_max:

              lpf_secondmax = e


    if ( lpf_secondmax == lpf_max ):

         sys.stderr.write( 'Warning: two or more maxima in the
file\n' )


    sys.stdout.write( 'Max LPF element:% 5.3f\n' % ( lpf_max ) )

    sys.stdout.write( 'INC value is: %d\n' % inc[lpf_imax] )


    # OUTPUT ANALYSIS


    filename2 = os.path.splitext( os.path.basename(filename) )[0] +
                                   '.odb'

    myODB = openOdb(path='/users/Natasha_2/Abaqus_Simulia_Jobs/%s' %
                     (filename2))

    failFrame = myODB.steps['Riks'].frames[inc[lpf_imax]]

    failDeflection = failFrame.fieldOutputs['U'].values


    for v in failDeflection:

        if (v.nodeLabel == attach_node):

              attach_node_defl = v.data[2]
```

```
# print('U3 deflection at the attachment point:%
           5.3f\n' % (attach_node_defl))

# print('Node where the brace is attached: %d\n' %
           (v.nodeLabel))


# EXTRA CALCULATION


P_fail = lpf_max*Load

P_br = k_br*attach_node_defl

Percentage = P_br/P_fail*100

L1_over_r = L1/r

treshold = 4.71*math.sqrt(29000/F_y)

F_e = math.pow(math.pi/L1_over_r, 2)*29000

if (L1_over_r < treshold):

     P_cr_theor_RigidBr = math.pow(0.658,
                                  F_y/F_e)*F_y*A

else: P_cr_theor_RigidBr = 0.877*F_e*A

F_e = math.pow(math.pi*r/Length, 2)*29000

if (Length/r < treshold):

     P_cr_theor_NoBr = math.pow(0.658, F_y/F_e)*F_y*A

else: P_cr_theor_NoBr = 0.877*F_e*A


# RESULTING DATA


out_file.write( '% 5.3f % 5.3f % 5.3f %d %d % 5.3f %
                5.3f % 5.3f % 5.3f % 5.3f % 5.3f %
                5.3f % 5.1f % 5.3f % 5.3f % 5.3f\n'

 % (k_br, lpf_max, P_fail, inc[lpf_imax], v.nodeLabel,

attach_node_defl, P_br, Percentage, Length, mesh_size,
```

```
                position, L1_over_r, F_y, W, P_cr_theor_RigidBr,
                P_cr_theor_NoBr) )


        j = j + 1


out_file.close()
```

APPENDIX B. VERIFICATION OF ABAQUS NUMERICAL SOLUTION

The verification of the Abaqus solution was started by comparing it to the theoretical closed form solution which can be built for an elastic model. This solution is mentioned in the literature and its detailed derivation can be found, for example, in Yang's paper [8]. The comparison of the Abaqus and analytical solutions can be done only within elastic domain so a large column's slenderness ratio was chosen: $\frac{L_1}{r}$ = 275. The graphs at Figures 14 and 15 represent the deformed shape of the column (situated along the x axis) in cases when the brace is in the center or shifted from it. The comparison was done using W10x14 shape.

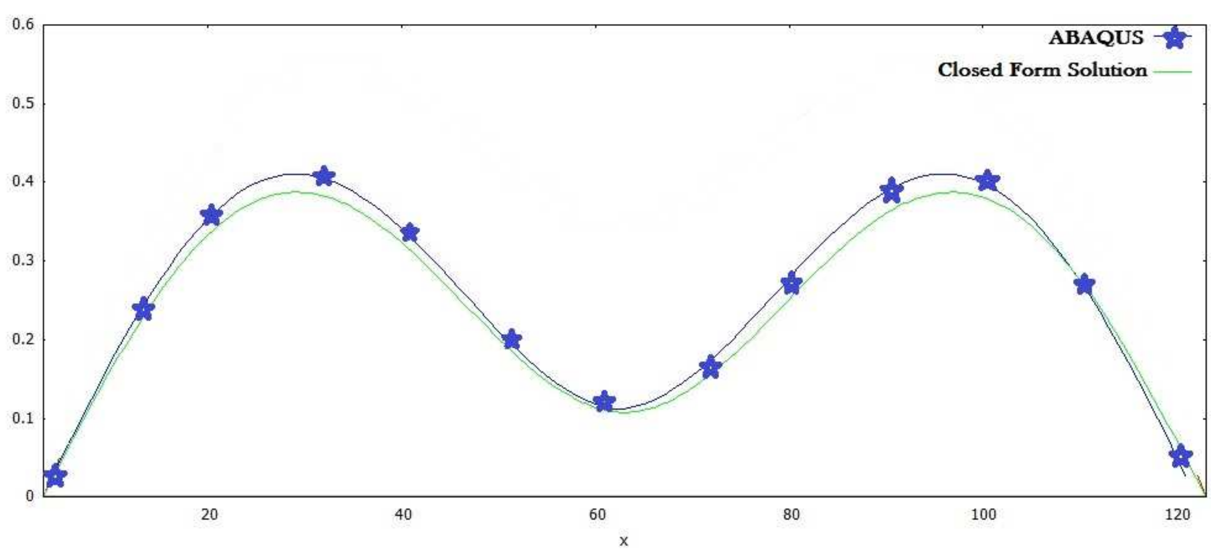


Figure 39. Deformed Column's Shape according to Abaqus and Closed Form Solutions when Brace is in the Middle

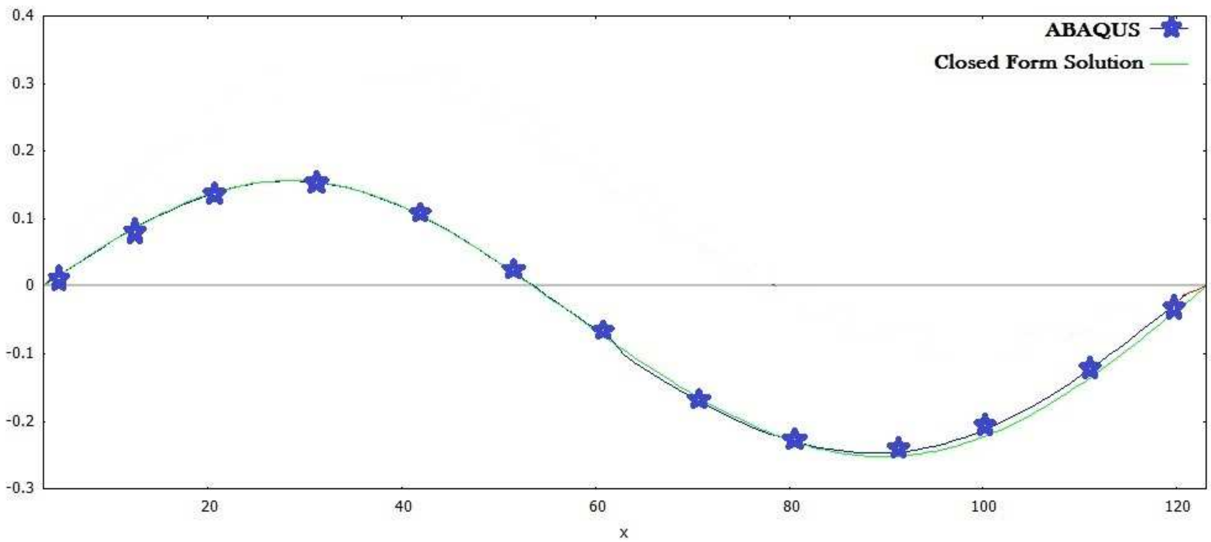


Figure 40. Deformed Column's Shape according to Abaqus and Closed Form Solutions when Brace is Shifted from the Center

The next step of the verification was an experimental curve for a critical buckling load of an unrestrained column that has its statistical approximation in the current AISC manual:

$$F_{cr} = \left(0.658^{\frac{F_y}{F_e}}\right) F_y \qquad \text{when} \quad \frac{KL}{r} \leq 4.71\sqrt{\frac{E}{F_y}} \qquad (3a)$$

$$F_{cr} = 0.877\, F_e \qquad \text{when} \quad \frac{KL}{r} > 4.71\sqrt{\frac{E}{F_y}} \qquad (3b)$$

The comparison of the critical stresses, Figure 16, was conducted for various slenderness ratios using W10x14 and W14x145 shapes. The difference between Abaqus results and AISC curve for short columns can be explained by influence of the residual stresses which typically reduce the critical stress and the Abaqus model does not take residual stresses into account (due to limitation of the student version of Abaqus™). Also the experiment data is only available for columns without any bracing so futher verification of Abaqus model is required.

Comparison with the results of the numerical solutions that other authors published was the last step in the process of Abaqus solution qualification. So the Abaqus results were analyzed against results obtained by Clark and Bridge [6] and O'Connor [3] for failure load (marked as P_u in their works). Figures 17 and 18 demonstrate that Abaqus solution, represented by isolated stars or pentagons, is close enough to the results of previous authors. The small difference, less than 7%, arisen in some cases can be again explained by the impact of the residual stresses that steel shapes have as a consequence of the manufacturing process.

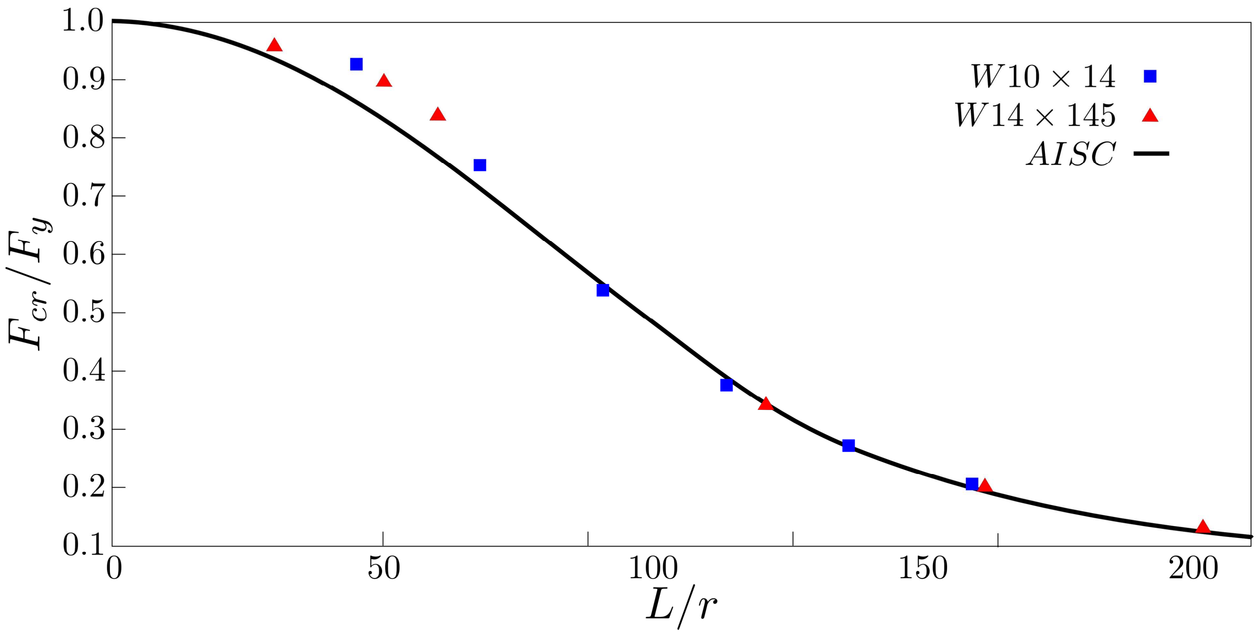


Figure 41. Comparison of the Critical Buckling Load Defined in AISC Manual and One Obtained with the Abaqus Solution.

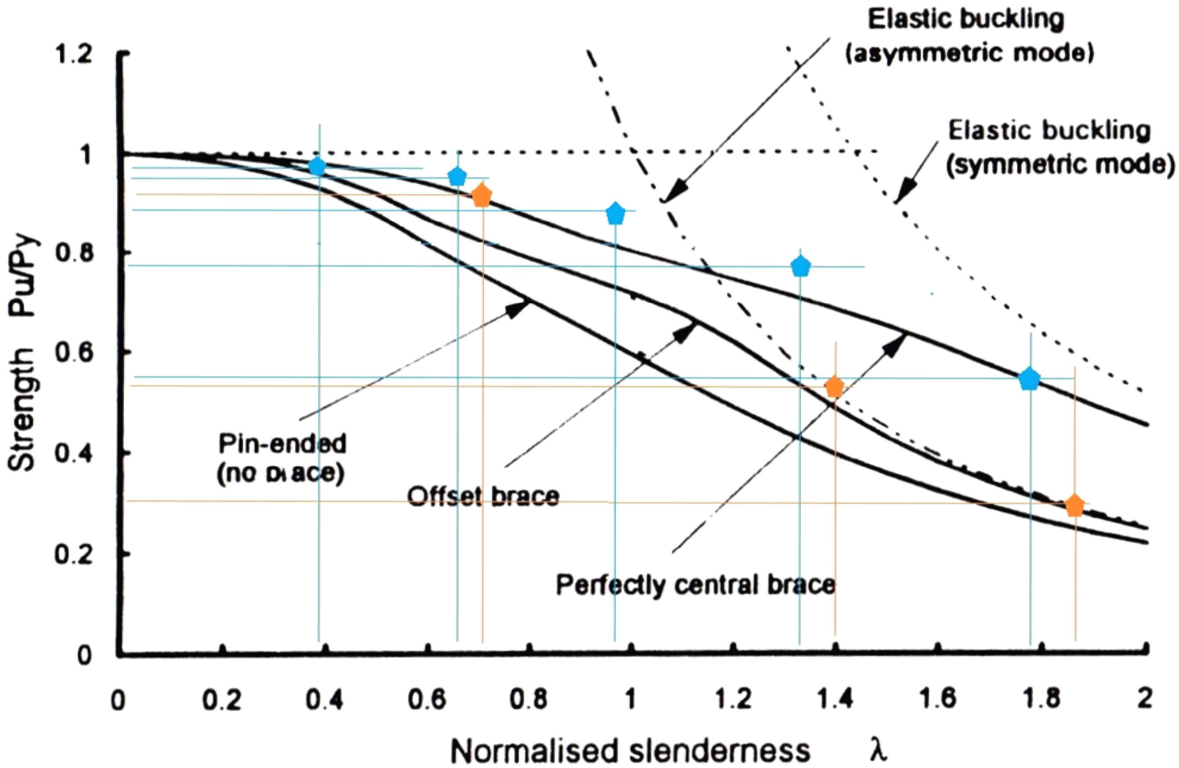
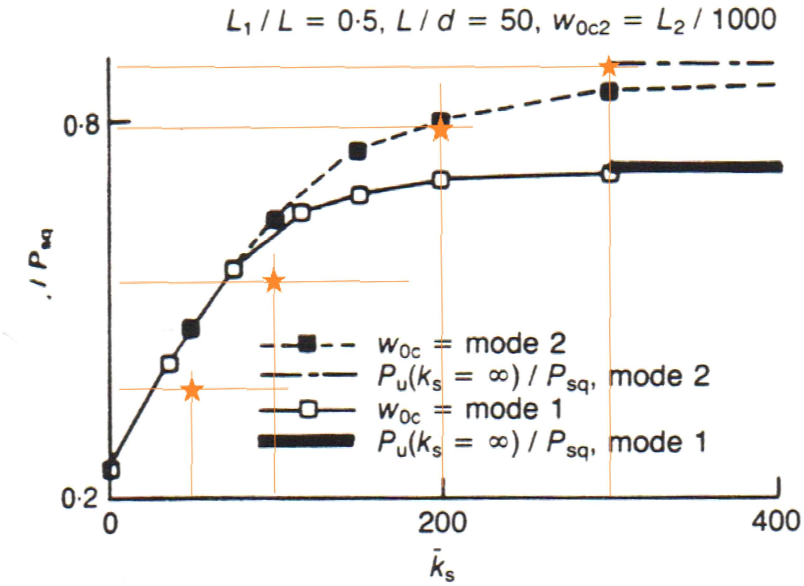


Figure 42. Comparison with Clark and Bridge Results



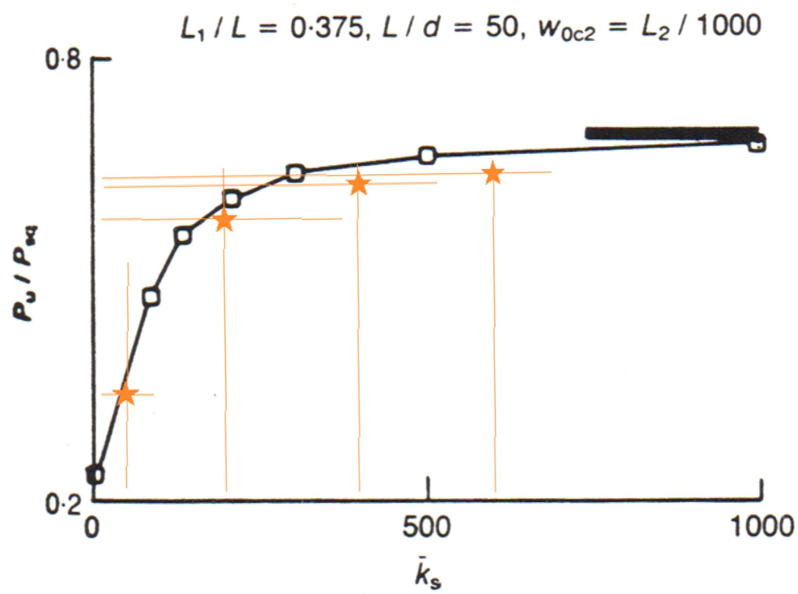


Figure 43. Comparison with O'Connor Results Obtained for Rectangular Beams

REFERENCES

[1] G. Winter, "Lateral Bracing of Columns and Beams," *Trans. ASCE,* vol. 125, no. Part 1, pp. 807 - 825, 1960.

[2] S. Timoshenko and J. Gere, Theory of Elastic Stability, New York: McGraw-Hill, 1936.

[3] C. O'Connor, "Imperfectly Braced Columns and Beams," *Institution of Engineers (Australia) Civ Eng Trans,* vol. CE21, no. 2, pp. 69-74, 1979.

[4] G. S. Stanway, J. C. Chapman and P. J. Dowling, "A Simply Supported Imperfect Column with a Transverse Elastic Restraint at Any Position. Part1: Behaviour," in *Instn Civ. Engrs Structs & Bldgs*, 1992.

[5] R. H. Plaut and J. G. Yang, "Lateral Bracing Forces in Columns with Two Unequal Spans," *Journal of Structural Engineering,* vol. 119, no. 10, pp. 2896-2912, 1992.

[6] M. J. Clarke and R. Q. Bridge, "Bracing force and stiffness requirements to develop the design strength of columns," in *Is Your Structure Suitably Braced*, Milwaukee, Wisconsin, 1993.

[7] J. A. Yura, "Winters Bracing Model Revisited," in *50th Anniv. Proc. SSRC*, Bethlehem, PA, 1994.

[8] L. Yang, "Behavior of the Column Restrained by a Flexible Brace. Master Thesis," University of Alaska, Fairbanks, Fairbanks, Alaska, 1996.

[9] R. D. Ziemian, Guide to stability design criteria for metal structures, John Wiley & Sons, 2010.