PERFORMANCE ANALYSIS SOFTWARE FOR REINFORCED CONCRETE BEAM-COLUMNS UNDER VARIOUS LOAD AND DISPLACEMENT PATTERNS

by

SCHYLER ROHLEDER

B.S., Kansas State University, 2014

A THESIS

submitted in partial fulfillment of the requirements for the degree

MASTER OF SCIENCE

Department of Civil Engineering
College of Engineering

KANSAS STATE UNIVERSITY
Manhattan, Kansas

2017

Approved by:

Major Professor

Dr. Asad Esmaeily

Copyright

SCHYLER ROHLEDER

2017

Abstract

Performance-based building design is a necessity in geographic locations where buildings are susceptible to large earthquakes and high winds. This design method requires an analysis of the performance of the structural system with loadings and deflections caused by earthquakes and wind. Current design codes include the load intensity in analysis procedures, but do not consider the effect of load pattern in the performance analysis of reinforced concrete columns. Because a thorough analysis must take into consideration load pattern and load intensity, computer software is ideal to analyze these systems.

A computer program was originally developed by Esmaeily (USC_RC), and was revised later to be renamed (KSU_RC) in order to make the analysis of concrete column performance accurate, yet simple for design purposes. This analytical tool used analytical methods and material models, verified against experimental data, to accurately predict the performance of reinforced concrete columns under various loading conditions, including any pattern in lateral direction and independently variable axial load. However, the program was limited to circular, rectangular, hollow circular and rectangular sections and uniaxial lateral curvature and displacement. The next generation of the program, KSU RC 2.0, was developed to overcome the aforesaid limitations.

Table of Contents

List of Figures ix
Chapter 1 - Introduction 1
Chapter 2 - Review of Analytical Models 1
Prior Work by Esmaeily 1
Esmaeily (2002) 1
Esmaeily (2004) 2
Esmaeily (2005) 2
Esmaeily (2006) 2
Esmaeily (2007) 2
Prior Work by Shirmohammadi 3
(Shirmohammadi & Esmaeily, 2015) 3
(Shirmohammadi, 2015) 3
(Shirmohammadi & Esmaeily, 2015) 4
(Shirmohammadi & Esmaeily, 2016) 5
Campione and Fossetti Model for Confined Concrete 5
Chapter 3 - Introduction to KSU-RC 2.0 6
Program Interface 7
Graphical Interface 8
Program Limitations 9
File Types Supported by KSU RC 2.0 9
Chapter 4 - How to Use KSU RC 2.0 11
Using the Menu Strip for Program Level Functions 11
File Menu 11
Tools 13
Help 13
Using the Tool Strip Menu for Project Level Functions 14
Section 14
Material Properties 15
Meshing 15

Analysis Options 16
Analysis Results List 17
Limitations 17
Opening Text File Results 19
Project Settings 20
Units 20
Data Points 21
Analysis Settings 22
Mesh Settings 23
Locking and Unlocking the Section 24
Pause/Stop Analysis 25
Basic Steps to Create a Project 26
Section Geometric Properties 27
Transverse Reinforcement 27
Longitudinal Reinforcement 28
Deleting a Section 29
Creating a Finite Mesh 29
Material Properties 31
User Defined Material Properties 31
Default Material Properties 31
Monotonic Material Models 33
Steel Monotonic Models 33
Elastic-Perfectly Plastic Model 34
Bi-Linear Model 35
Tri-Linear Model 36
Quadra-Linear Model 37
Esmaeily Model 38
Core Concrete Monotonic Models 39
Concrete Confined by Lateral Steel Reinforcement 39
Mander Model 39
Cusson and Paultre Model 41

Campione and Fossetti Model 42
Concrete Confined by FRP 42
Youssef Model 43
Samaan Model 43
Concrete Confined by FRP and Conventional Steel 44
Lee Model 44
Kawashima Model 45
Shirmohammadi Model 46
Custom Model 47
Note 47
Cover Concrete Monotonic Models 48
Plain or Unconfined Concrete 48
Mander 48
Cusson and Paultre 49
Concrete Confined by FRP 50
Youssef Model 50
Samaan Model 51
Custom Model 52
Note 52
FRP Monotonic Models 53
Cyclic Material Models 54
Concrete Cyclic Models 55
Linear Model 55
Mander Model 55
Esmaeily Model 56
Steel Cyclic Models 57
Linear Model 57
Esmaeily Model 57
Menegotto-Pinto Model 59
FRP Cyclic Models 60
Linear Model 60

Graphic Test of Cyclic Models 60
Analysis Methods 63
Axial Force-Bending Moment Interaction 63
ACI Interaction Curves 63
Method 1 63
Method 2 64
User Input 65
Results 66
Interaction Using Material Models 67
User Input 68
Results 69
Interaction Analysis Considering Moment Curvature 69
User Input 70
Results 71
Interaction Analysis Considering Load Pattern 71
User Input 72
Results 73
Moment Curvature Analysis Methods 74
ACI Code-Based Moment-Curvature Analysis 74
User Input 75
Results 76
Moment Curvature Based on Material Models 77
User Input 78
Results 79
Force-Deflection Analysis Methods 82
Priestley and Park 83
Priestley and Park (modified by Esmaeily) 83
Esmaeily's First Method 83
Esmaeily's Second Model 84
User Input 86
Fixed Axial Force 87

Non-Proportionally Variable Axial Force 88
Results 89
Viewing and Manipulating Result Graphs 91
Chapter 5 - Conclusion and Recommendations 93
Chapter 6 - Bibliography 95

List of Figures

Figure 1. Main Interface for KSU RC 2.0 7
Figure 2. Menu Strip 8
Figure 3. Tool Strip 8
Figure 4. CharAsad Graphical Interface 9
Figure 5. Sample Image Output for Capture Section Image Function 13
Figure 6. Result List Window 18
Figure 7. Plot Options Window 18
Figure 8. List of Available Forms in Plot Options Form 18
Figure 9. Sample Comparison of Variable Axial Force Using the Result List 19
Figure 10. Units Tab in Project Settings Window 21
Figure 11. Data Points Tab in Project Settings Window 22
Figure 12. Analysis Settings Tab in Project Settings Window 23
Figure 13. Mesh Settings Tab in Project Settings Window 24
Figure 14. RC Section Properties Window 26
Figure 15. Triangulation or Meshing of a Hollow Oval Section 30
Figure 16. Basic Material Properties Window 32
Figure 17. Steel Monotonic Models Tab in Monotonic Material Models Window 34
Figure 18. Elastic-Perfectly Plastic Stress-Strain Curve 35
Figure 19. Bi-Linear Stress-Strain Curve 36
Figure 20. Tri-Linear Stress-Strain Curve 36
Figure 21. Quadra-Linear Stress-Strain Curve 37
Figure 22. Mander Steel Model Stress Strain Curve 38
Figure 23. Esmaeily Stress-Strain Curve 39
Figure 24. Mander Model for Confined Concrete 40
Figure 25. Cusson and Paultre Model for Confined Concrete 41
Figure 26. Campione and Fossetti Model for Confined Concrete 42
Figure 27. Youssef Model for Confined Concrete 43
Figure 28. Samaan Model for Confined Concrete 44
Figure 29. Lee Model for Confined Concrete 45

Figure 30. Kawashima Model for Confined Concrete 46
Figure 31. Shirmohammadi Model for Confined Concrete 47
Figure 32. Mander Model for Unconfined Concrete 49
Figure 33. Cusson and Paultre Model for Unconfined Concrete 50
Figure 34. Youssef Model for Confined Concrete 51
Figure 35. Samaan Model for Confined Concrete 52
Figure 36. FRP Tab in the Monotonic Material Models Window 53
Figure 37. Cyclic Material Model Window 54
Figure 38. Linear Concrete Cyclic Curve 55
Figure 39. Mander Concrete Cyclic Curve 56
Figure 40. Esmaeily Concrete Cyclic Curve: Unloading and Loading Segments 57
Figure 41. Esmaeily Steel Cyclic Model 59
Figure 42. Menegotto-Pinto Steel Cyclic Model 60
Figure 43. Test of Steel Cyclic Model 61
Figure 44. Test of Core Concrete Cyclic Model 62
Figure 45. Test of Cover Concrete Cyclic Model 62
Figure 46. Curvature Change in Method 1 64
Figure 47. Curvature Change in Method 2 65
Figure 48. ACI Interaction User Input Window 66
Figure 49. ACI Interaction Curve Produced by KSU-RC 2.0 67
Figure 50. Interaction Analysis Input Window 68
Figure 51. Interaction Curve Considering Material Models Produced by KSU-RC 2.0 69
Figure 52. Interaction User Input Window 70
Figure 53. Interaction Curve Considering Moment-Curvature Produced by KSU-RC 2.0 71
Figure 54. Interaction Curve Considering Axial Load Pattern 74
Figure 55. ACI Approximate Moment-Curvature Nomenclature 75
Figure 56. ACI Moment-Curvature User Input Window 76
Figure 57. ACI Moment-Curvature Curve Produced by KSU-RC 2.0 77
Figure 58. Moment-Curvature User Input Window 79
Figure 59. Monotonic Moment-Curvature Results Produced by KSU-RC 2.0 80
Figure 60. Variable Axial Load Moment-Curvature Results Produced by KSU-RC 2.0 80

Figure 61. Cyclic Moment-Curvature Results Produced by KSU-RC 2.0 81
Figure 62. Curvature Distributing along Column Height as Assumed in (a) Priestley and Park and Priestley and Park revised by Esmaeily method, (b) Esmaeily's First method, and (c) Esmaeily's Second Method..... 85
Figure 63. Force-Deflection Analysis Flowchart..... 85
Figure 64. Force-Deflection User Input Window 86
Figure 65. Monotonic Force-Deflection Results Produced by KSU-RC 2.0..... 89
Figure 66. Cyclic Force-Deflection Results Produced by KSU-RC 2.0..... 90
Figure 67. Single Axis Analysis Results Window..... 91
Figure 68. Dual Axis Analysis Results Window 92

Chapter 1 - Introduction

Performance-based building design is a necessity in geographic locations where buildings are susceptible to large earthquakes and high winds. These advanced design procedures surpass code requirements to improve the life-safety of buildings in critical seismic regions. Current design codes include the load intensity in analysis procedures, but do not consider the effect of load pattern in the performance analysis of reinforced concrete beam-columns. Researchers have taken to the task of better understanding the behavior of reinforced concrete structures under seismic loading. The works on this topic by two researchers in particular, Dr. Esmaeily, and Dr. Shirmohammadi, have been applied extensively to this thesis. Their research was to find accurate analytical models to represent the behavior of reinforced concrete columns under variable axial forces and displacements. The research done by these two individuals is reviewed in this paper as it applies to the creation of a computer program to analyze the performance of reinforced concrete beam-columns subjected to variable axial loads and displacements.

The aforementioned computer program was developed by Esmaeily as USC_RC in 2000, and was revised later to be renamed KSU_RC in order to make the analysis of concrete column performance accurate, yet simple for design purposes. This analytical tool used analytical methods and material models to accurately predict the performance of reinforced concrete columns under various loading conditions, including any pattern in lateral direction and independently variable axial load. However, the program was limited to circular, rectangular, hollow circular and rectangular sections and uniaxial curvature and lateral displacement. The next generation of the program, KSU RC 2.0, was developed to overcome the aforesaid limitations. This thesis reports the work done to improve the user interface of KSU RC 2.0 and

refine the analytical algorithms used in the computer program. All of the implemented features of the program are discussed in detail to provide a clear understanding of how to use the program. The material models and methods used in the program, as well as the performance of the program using these models were validated in previous research projects, that are reviewed in this paper.

Chapter 2 - Review of Analytical Models

Analytical material models are used in KSU RC 2.0 to model the behavior of the constituent materials in the reinforced concrete section. They have been developed by researchers to model the stress-strain relationship of steel, concrete, and FRP. Mathematical relationships are fit to the results of experimental data to represent the behavior of these materials. The following sections review the research that was used as the basis of the theoretical backbone of KSU RC 2.0, or the performance of reinforced concrete members. Esmaeily's work started in the late 1990's and Shirmohammadi's work continued that of Esmaeily into 2016.

Prior Work by Esmaeily

A large amount of research was done by Asad Esmaeily on the behavior of reinforced concrete columns under variable axial loads and displacements prior to the creation of KSU RC 2.0. This section reviews the work done by Dr. Esmaeily that was used in the creation of KSU RC 2.0. The work is presented in chronological order to show the progression of the work over time.

Esmaeily (2002)

Six large scale bridge piers were tested to find the effect of the axial force magnitude and loading pattern on the performance of the reinforced concrete section. The research found that variable axial loads had a significant impact on the performance of the section, compared to the same loading pattern under constant axial force. USC_RC was developed in this research as an analytical tool to compare analytical models to the experimental results of this research.

Esmaeily (2004)

This research was a continuation of Esmaeily (2002). Analytic models were compared with experimental results from large scale tests done on reinforced concrete bridge piers under variable axial loads and lateral displacements. The results from this research were used to modify and calibrate plastic hinge models that are used in KSU RC (Esmaeily, 2005).

Esmaeily (2005)

A computer program was developed to analyze the effects of axial load magnitude and pattern on the performance of reinforced concrete members. The research from Esmaeily (2004) was used to find that the computer program, and the analytical models used in the program, could predict the behavior of a reinforced concrete member under variable load patterns. Another point made by this research is that the analytical models used in to represent the material behaviors could be improved upon to ensure even more accurate behavior predictions.

Esmaeily (2006)

Esmaeily (2005) revealed that there was a need to improve the analytical stress-strain relationships representing material behavior. The computer program developed in Esmaeily (2005) was used to analyze the effect of axial load on the confinement effects of reinforced concrete.

Esmaeily (2007)

KSU_RC was the second version of the USC_RC program developed by Esmaeily to analyze the performance of reinforced concrete sections and structural members under any kind

of load and displacement path variation. While USC_RC was developed in late 90's and early 2000's, the KSU_RC was created around the earlier findings from Esmaeily (2004, 2005, and 2006). The functionality of the program was verified against experimental results to ensure that the predictions of member behavior by the program were accurate. This program has been used as a simple and free analytic tool by students, faculty, and engineers all over the world.

Prior Work by Shirmohammadi

Dr. Shirmohammadi supplemented the work of Dr. Esmaeily by continuing research of the performance of reinforced concrete columns. The work done by Dr. Shirmohammadi that was used in the creation of KSU RC 2.0 is reviewed in this section. Again, the work is presented in chronological order.

(Shirmohammadi & Esmaeily, 2015)

A stress-strain model was created in this research to predict the behavior of concrete confined by FRP wraps and conventional lateral steel. The accuracy of the constituent model was validated against experimental results from members under concentric axial loads. The model was also validated by using the new model in the KSU_RC to test the moment curvature response, which was then compared with experimental moment curvature results.

(Shirmohammadi, 2015)

Shirmohammadi (2015) wrote an extensive literature review of the material models used in KSU RC 2.0. The models included those for steel, confined concrete, cover concrete, and

FRP. In order to avoid repetition, the models included in the review from Shirmohammadi (2015) are not included in this paper. These models are:

- Steel
    - Elastic Perfectly Plastic Model
    - Multilinear Models
    - Mander Model
    - Esmaeily's Model
- Confined Concrete
    - Mander Model
    - Cussan and Paultre Model
    - Lee et al. Model
    - Samaan et al. Model
    - Youssef et al. Model
    - Kawashima et al. Model
    - Shirmohammadi's Model
- Unconfined Concrete
    - Mander Model
    - Cussan and Paultre Model
- FRP
    - Linear Model

(Shirmohammadi & Esmaeily, 2015)

The accuracy of some of the previously mentioned material models was analyzed against experimental data using moment curvature and force deflection analyses in a fiber based analytical procedure. These models were found to be accurate and were then used as to create an analytic method to analyze the cyclic performance of concrete columns under biaxial loads and displacements.

(Shirmohammadi & Esmaeily, 2016)

The second version of KSU_RC, KSU RC 2.0 was developed by (Shirmohammadi & Esmaeily, 2016) to assess the biaxial cyclic performance of reinforced concrete members under variable axial force and displacement patterns. Monotonic and cyclic material models are used with a fiber based section model to analyze the performance of a reinforced concrete member using Force-Deflection, Moment Curvature, and Interaction analysis methods. Any section geometry can be analyzed by using the fiber based modeling technique. The accuracy of the material models and methods were verified with analytical and experimental results from past research.

Campione and Fossetti Model for Confined Concrete

An additional model was added to the computer program that was not covered in Shirmohammadi (2015). Campione and Fossetti (2007) created a simplified model to determine the stress-strain relationship for elliptical shaped concrete sections with steel hoop confinement. Effective stresses in the hoops are evaluated to predict the maximum confinement and the maximum compressive strength of the confined concrete. The proposed stress-strain relationship follows a similar equation to that of Mander (1988), with the consideration of the geometry of the section in the calculation of effective confinement of the concrete from the steel hoops. This model can be used for circular columns, since a circle is an ellipse with equal dimensions in both height and width.

Chapter 3 - Introduction to KSU-RC 2.0

KSU-RC is an analytical tool that can be used to assess the performance of reinforced concrete columns. The very first version of the software was first created by Esmaeily as USC_RC (2002) and revised and renamed later by Esmaeily as KSU_RC (2007). This version, was capable of analysis the performance with any lateral load pattern and axial load variation, but was limited to circular, rectangular, and hollow circular or rectangular sections under uniaxial lateral curvature or displacement.

Shirmohammadi (2015) began updating the second version of the program, KSU RC 2.0, as part of her doctoral dissertation. The work done on KSU RC 2.0 described in this thesis is a continuation of that work, focusing on refining the user interface and the analytical algorithms used in the program. The culmination of the program has resulted in the current generation of KSU-RC that has the following functionality and options that have changed since the original version of the program.

Triangulation of the defined section allows for a wider range of cross-sectional geometries. The basis of the program is the use of analytic material behavior models under monotonic and cyclic loading to represent the section behavior under the specified loading conditions. Each element of the section (created by triangulation) follows the appropriate stress-strain behavior model based on the element type and deformation history. The element can be specified as confined concrete, unconfined concrete, longitudinal steel or FRP bars, or lateral/longitudinal FRP wraps. Biaxial lateral curvature, displacement and forces can be combined with any sequence of axial loads in the analysis of the user defined section. This allows for the analysis of a reinforced concrete column under any load or displacement path.

Another feature in the current version of KSU-RC is the use of FRP reinforcement in both the lateral and longitudinal directions.

Program Interface

The main window is shown below in Figure 1. From this window, the user can access program level functions using the Menu Strip (see Figure 2), project level functions using the Tool Strip (see Figure 3), and graphical functions using CharAsad (see Figure 4).

Figure 1. Main Interface for KSU RC 2.0

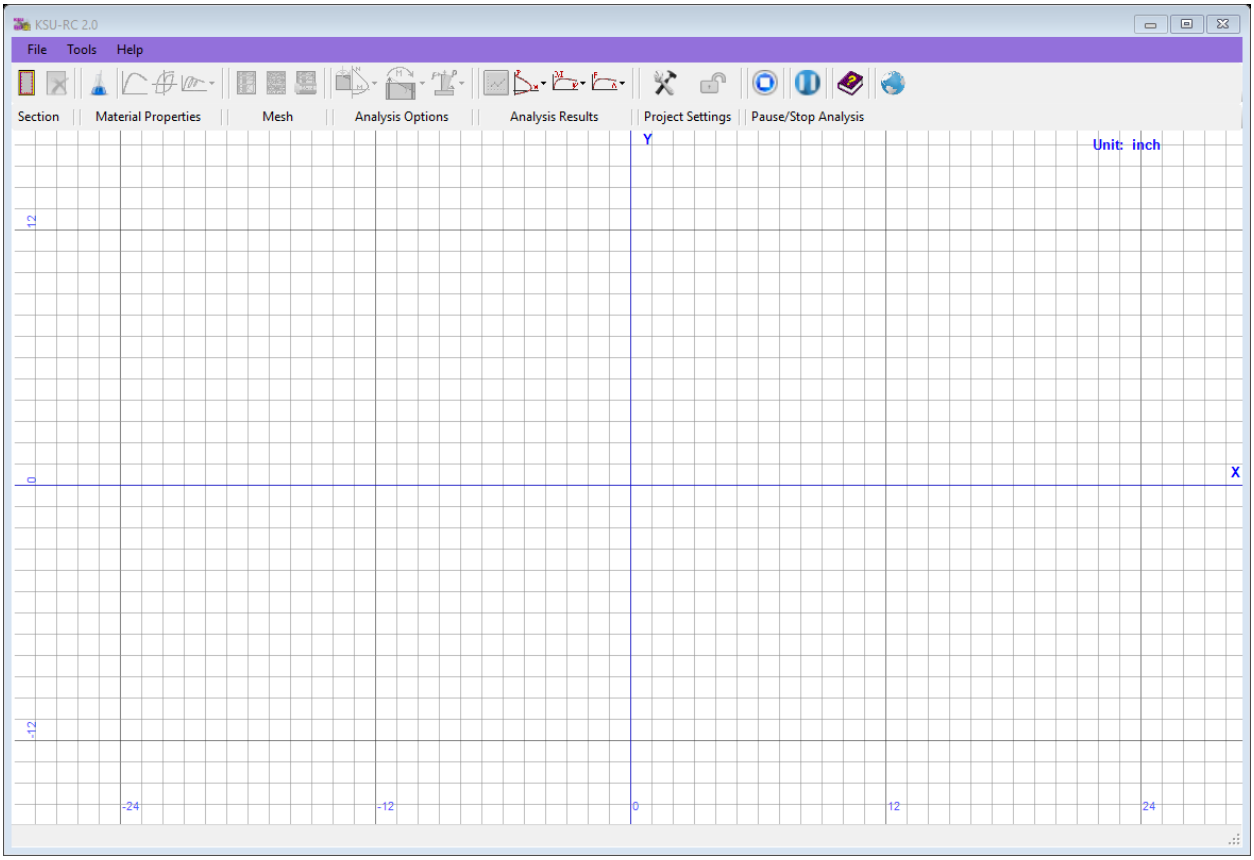


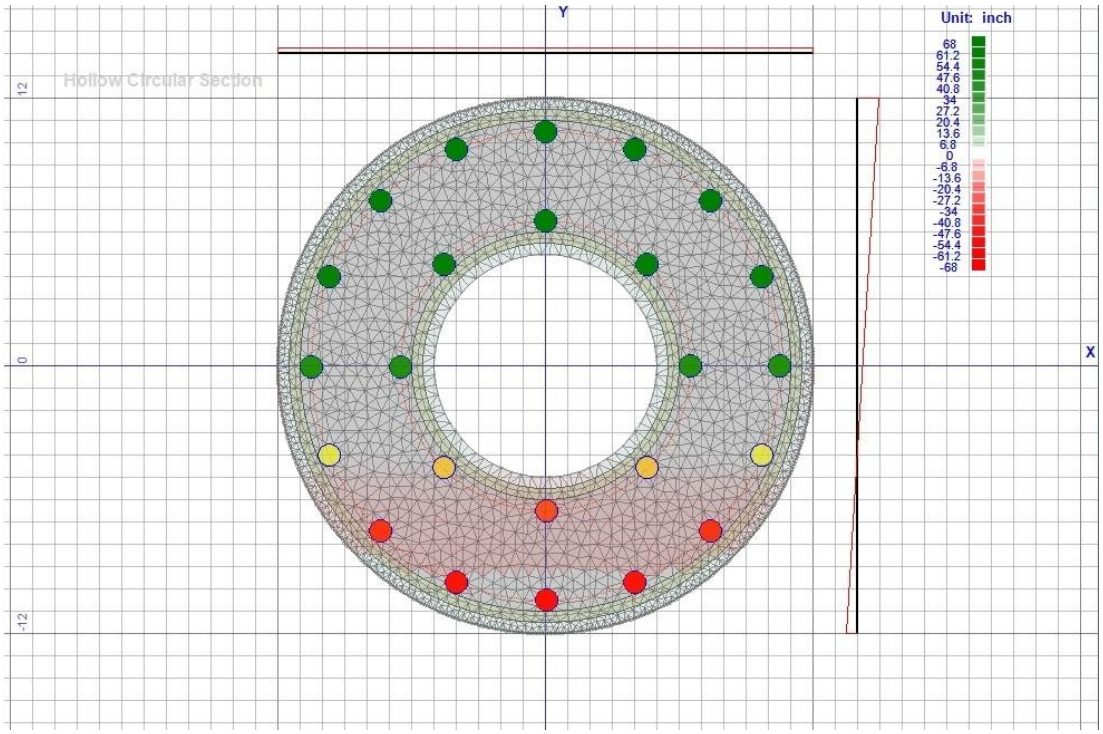
Figure 2. Menu Strip

Figure 3. Tool Strip

Graphical Interface

The CharAsad control was created as a tool to show the RC section and triangular mesh. The user can also input longitudinal reinforcement in the section using the CharAsad control. The size, position, and color of the longitudinal bars can be changed graphically. When an analysis is run in KSU-RC 2.0, the stress and strain conditions in the steel and concrete are shown in real time during an analysis in this control. Future versions of KSU-RC will allow the user to input the section geometry graphically.

Figure 4. CharAsad Graphical Interface



Program Limitations

10 results of each type of analysis can be stored in a project (50 total analyses). If the user runs more than 10 analysis of each type, the previous analysis results will be overwritten in the project.

File Types Supported by KSU RC 2.0

KSU RC 2.0 uses the .krc2 extension, a unique file type created for saving and opening projects using the new program. More functions are available within the program to save data

created by analyses in text file (.txt) form or to save result graphs in an image form (.png, .jpg, etc.). Results saved in text file form can be opened again in a graphical form.

Chapter 4 - How to Use KSU RC 2.0

The proceeding sections will detail the specifics of all of the user functions within KSU RC 2.0. Program functions are explained starting at high level program functions that control user specific settings stored in the program and other functions such as those to save and open new and existing program files. Next, project specific functions such as those related to defining a concrete cross-section, analyzing a section, and viewing graphical results are discussed.

Using the Menu Strip for Program Level Functions

The user can access program level functions using the Menu Strip. Program functions are used to open new and existing projects, save a project, change user specific settings within the program, and view help documentation. The following sections describe each Menu item in detail: the File menu, the Settings menu, and the Help Menu.

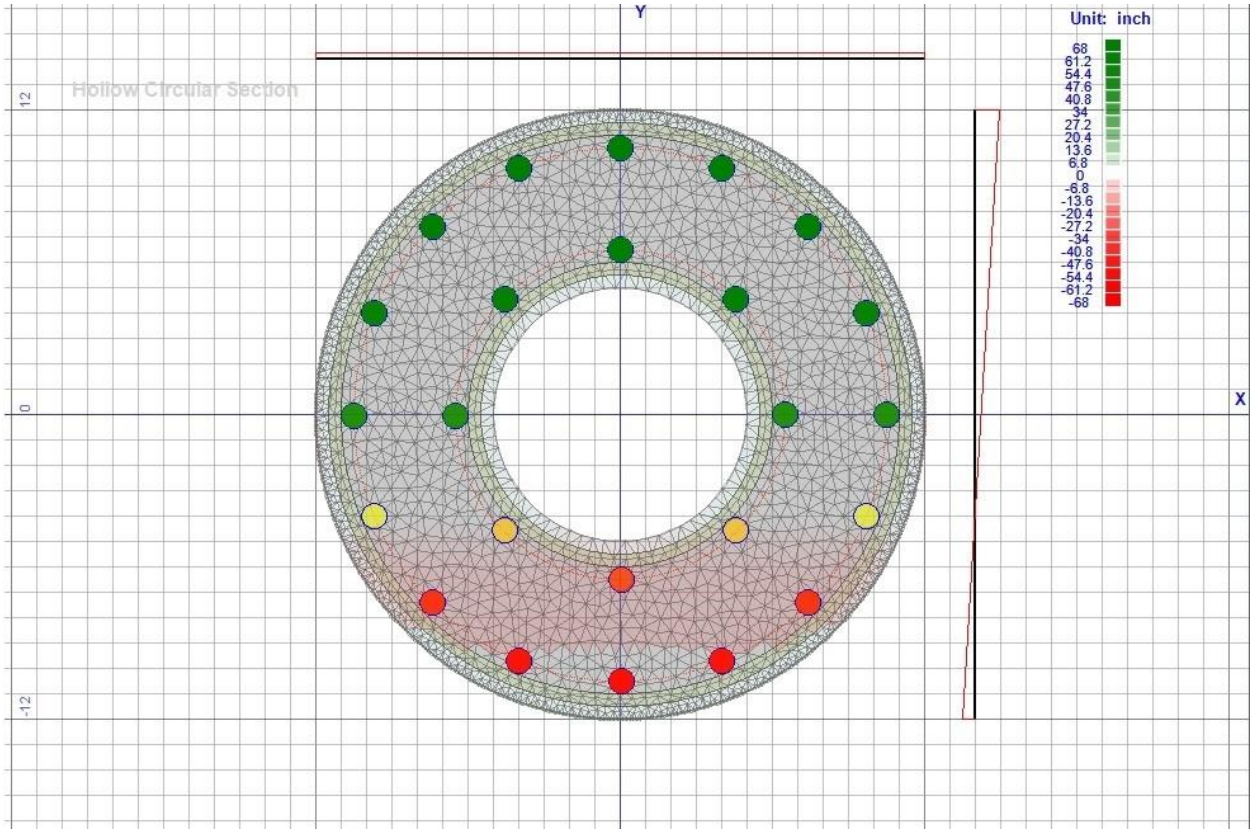
File Menu

- New: Opens another instance of KSU-RC and creates a new project in this new window.
- Open: Opens an existing '.krc2' project in the current project window.
- Save: Saves the current project state using a '.krc2' extension. Note that if the project has not been saved previously, the Save As prompt will open.
- Save As: Saves the project using a '.krc2' extension.
- Save RC Section: This function is a method of extracting data from the program in a text file (.txt). Project information such as the date and time that the text file was saved, the text file name, and the project unit system are listed at the top of the saved text file. The section properties listed below are saved using the Save RC Section function in the File Menu:

    - Section Properties
    - Section Type
    - Clear Cover
    - Dimensions (Height, Width, etc.)
    - Material Properties
    - Concrete Strength
    - Concrete Modulus
    - Longitudinal Steel Yield Strength
    - Longitudinal Steel Modulus
    - Transverse Steel Yield Strength
    - Transverse Steel Modulus
    - FRP Tensile Strength
    - FRP Tensile Modulus
    - FRP Compressive Strength
    - FRP Compressive Modulus
    - Longitudinal Steel Bar List
    - Bar No., Bar Size, Bar Diameter, Bar Area, Bar Location
    - Transverse Steel (when applicable)
    - Transverse Reinforcement Type
    - Transverse Bar Spacing
    - Bar Size
    - Bar Diameter
    - Bar Area
    - FRP (when applicable)
    - FRP Thickness
    - FRP Direction (lateral, “One”; or lateral and longitudinal, “Two”)
- Capture Section Image: Saves the current RC section graphic in image form. Supported Image Formats are: .bmp, .gif, .jpeg, .png, and .tiff. See Figure 5 for a sample of the output using this function.
- Close Project: Use this function to close the current project while keeping the program window open.

- Exit ---- Closes the application

Figure 5. Sample Image Output for Capture Section Image Function



Tools

- Program Settings: The user can change settings related to the visual graphics in the program. Here, stress and strain colors, mesh color, and mesh line thickness and transparency can be changed.

Help

- Help Topics: This link opens the help file in a separate window, opened to the contents section.
- About: This link opens a separate window that shows the program information
- Show Splash Form: The splash form is opened automatically when the program is first opened. The user can choose to show this form when the program starts up by checking

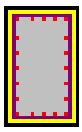
the *Show this form on startup* checkbox in the splash form. To close the splash form, click anywhere on the form.

- Show Credits: A window is opened, showing scrolling credits for the work done on this project.

Using the Tool Strip Menu for Project Level Functions

The user can access project level functions using the Tool Strip Menu. This is the main menu for a project in KSU-RC 2.0. From this menu, the user can create a reinforced concrete (RC) section, chose material properties and constitutive stress strain models, analyze the performance of the created section, and view and manipulate graphical results from saved analyses. The process for analyzing the performance of a section using the program closely follows the layout of the Tool Strip Menu as follows:

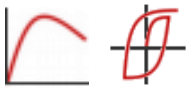
Section



The Section group of controls on the Tool Strip Menu allows the user to create and remove an RC section. Geometric properties, transverse reinforcement type, and longitudinal reinforcement placement are defined by the user in these functions. For more information on creating and deleting a section, see the Basic Steps to Create a Project section.

Material Properties

Material strength properties, analytical material monotonic and cyclic behavior models are implemented in this group of controls. The basic material properties can be changed by clicking the button shown above. To read more about changing material properties see the Material Properties section.

 Analytic models governing the stress-strain behavior of longitudinal bars, core concrete, cover concrete, and FRP wraps can be chosen by the user. Note that the default monotonic and cyclic models are chosen by the program based on the section type and transverse reinforcement type that are selected by the user. The user can chose to precede with the default material models, or select a model of their choice. Cyclic models can be tested graphically to confirm the model is used appropriately in the program. See the Monotonic Material Models and the Cyclic Material Models sections to read more about the available models and how to use them in a project.

Meshing

The defined section must be divided into discrete fibers in order to apply material stress-strain behavior models to those fibers during an analysis. To mesh the section, press the button shown above. To refine the mesh and increase the accuracy of an analysis press the button shown above. If a user wishes to delete the mesh, press the button

shown above. See the Creating a Finite Mesh section to read about how the mesh is applied to a section.

Analysis Options

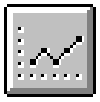
KSU-RC 2.0 has the capability of using 10 different methods to analyze the performance of the user defined RC section. These methods spur from 3 types of analysis:

- Axial Force-Bending Moment Interaction Analysis
- Moment-Curvature Analysis
- Force- Deflection Analysis

The Analysis Methods section contains more information about performing analyses with KSU RC 2.0. The 10 methods available to analyze the performance of a section are:

- ACI Based Interaction Analysis
- Interaction Analysis Using Material Models
- Interaction Considering Moment-Curvature Analysis
- Interaction Considering Load Pattern
- ACI Based Moment-Curvature Analysis
- Moment-Curvature Analysis Using Material Models
- Force-Deflection Analysis Using Esmaeily's First Method
- Force-Deflection Analysis Using Esmaeily's Second Method
- Force-Deflection Analysis Using Priestley and Park's Method
- Force-Deflection Analysis Using Priestley and Park Revised by Xiao and Esmaeily

Analysis Results List


The Result List window is used to access the results of analyses that have been performed in a project. The user can open analysis results in separate windows, or plot multiple result curves of the same type of analysis on the same window. When a result button is clicked in the Result List Window (Figure 6), the user will be prompted (Figure 7) to choose whether to open the result curve in a new window or open the result curve in an existing result window (Figure 8). Multiple curves of the same type of analysis (i.e. Moment-Curvature) can be plotted on one result window (Figure 9). This allows the user to compare the effect of changed material properties, transverse reinforcement properties, monotonic models, or cyclic models on the analysis results.

Limitations

10 results of each type of analysis can be stored in a project (50 total analyses). If the user runs more than 10 analysis of each type, the previous analysis results will be overwritten in the project.

Figure 6. Result List Window

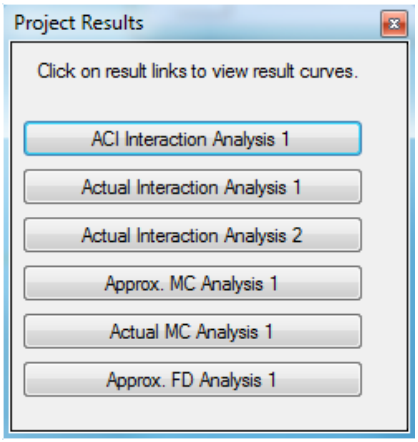


Figure 7. Plot Options Window

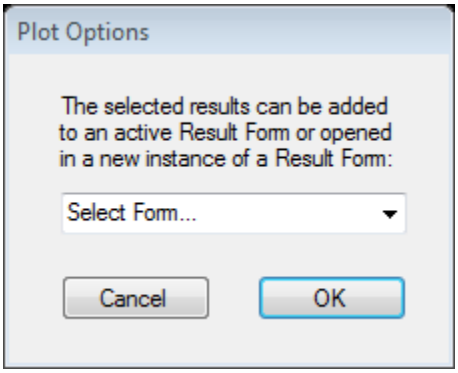


Figure 8. List of Available Forms in Plot Options Form

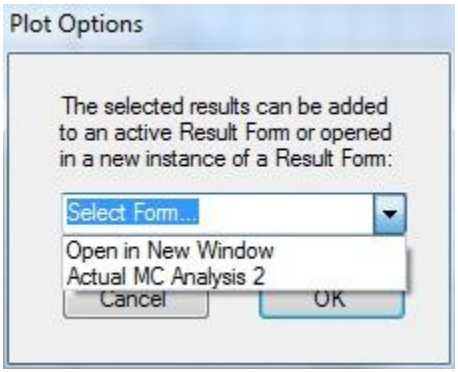
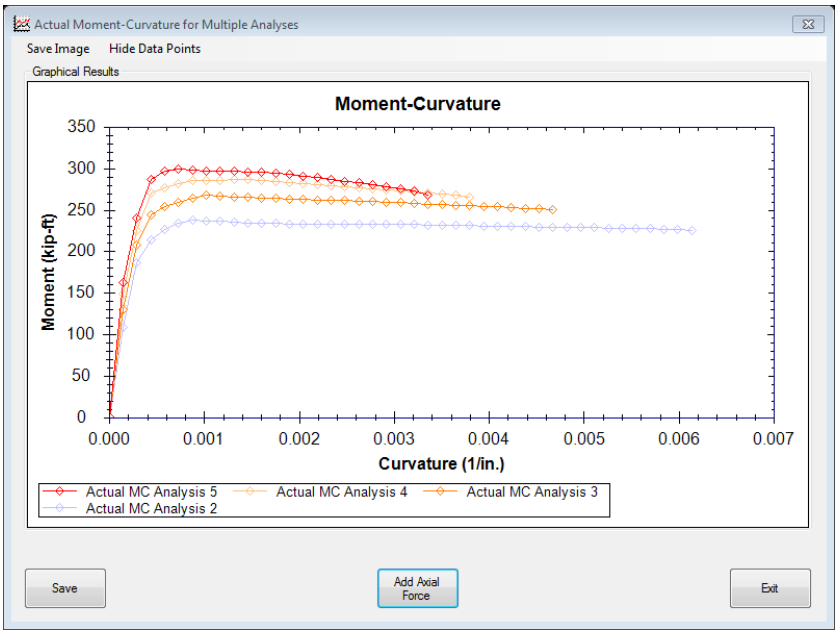


Figure 9. Sample Comparison of Variable Axial Force Using the Result List



Opening Text File Results

Results saved to a text file using KSU-RC 2.0 can be opened to view the result curve in a result window using the Analysis Results group of buttons on the Tool Strip Menu. The result type in the text file must match the result type chosen by the user in this group of buttons. Result types include:

- ACI Interaction Analysis
- Actual Interaction Analysis
- Approximate Moment-Curvature Analysis
- Actual Moment-Curvature Analysis
- Force Deflection Analysis

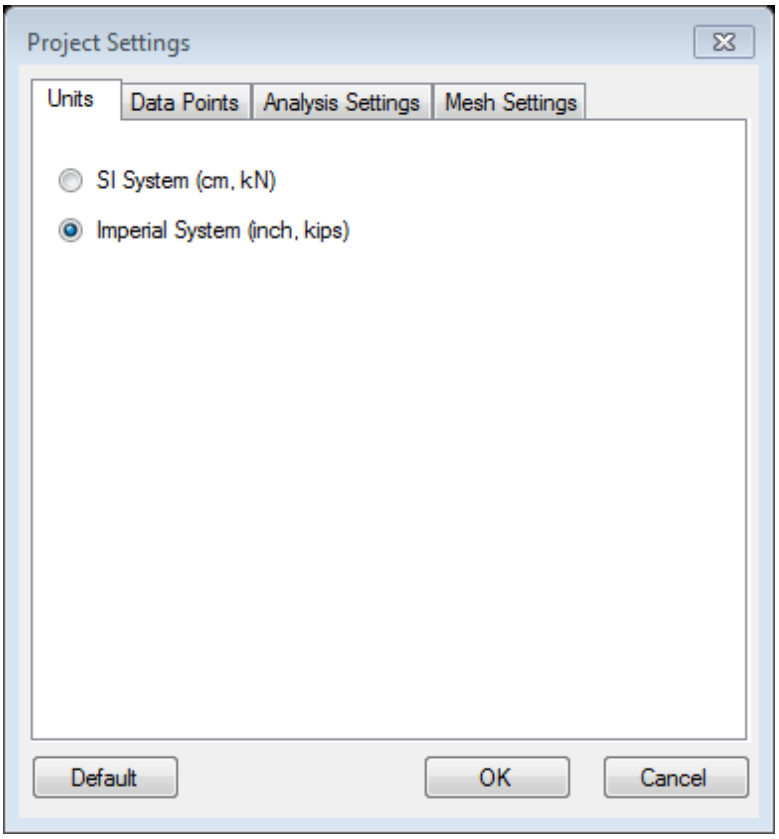
Project Settings

The user can choose the unit system for the project (SI or Imperial), specify the number of data points in each analysis, specify if an analysis should stop when steel or concrete reaches a strain limit (user specified), and change the mesh settings in the Project Settings Window.

Units

The unit system of the project can be defined under this tab. Imperial units or metric units are the two options for measurement units in KSU RC 2.0. If a section is defined, the unit system of a project cannot be changed. To change the unit system, the defined RC-section must be deleted and then the Units tab will be activated.

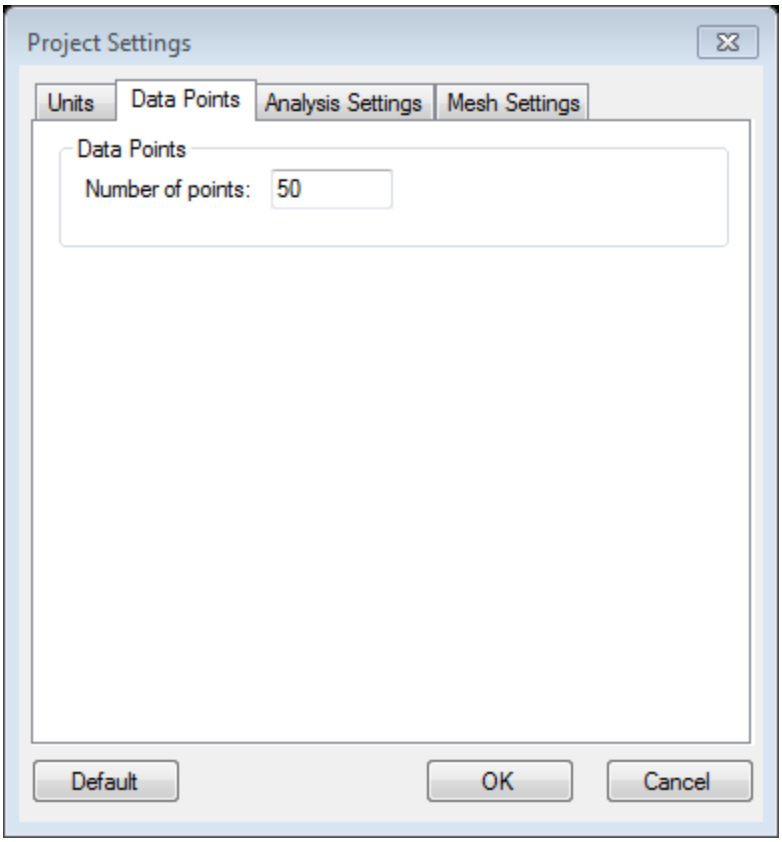
Figure 10. Units Tab in Project Settings Window



Data Points

The number of data points in each analysis can be defined using the “number of points” in the second tab of Project Settings window. The number of points is the number of points in each analysis graph. Increasing the “number of points” increases the analysis time since larger number of points should be calculated for each type of analysis.

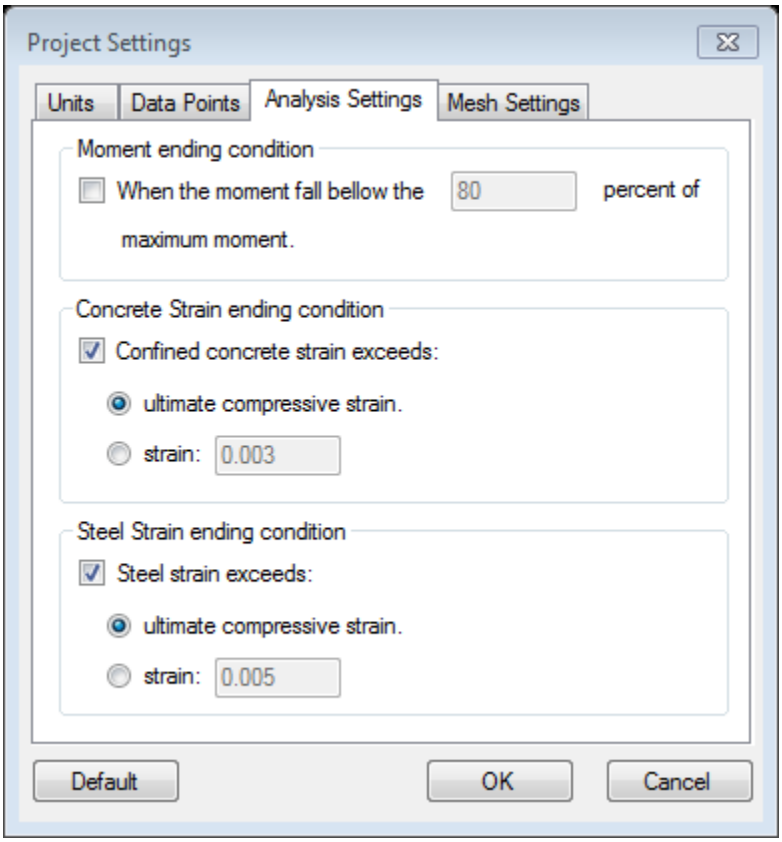
Figure 11. Data Points Tab in Project Settings Window



Analysis Settings

The analysis settings are defined in the Analysis Settings tab, the third tab of the Project Settings window. Three types of end conditions are defined in the software. The first end condition is when the moment falls below the specific percentage of the maximum moment. The second one is when the strain at one of the concrete fibers reaches a specific value or the ultimate compressive strain. The third condition is when the steel strain reaches a specific value of the ultimate tensile/compressive strain.

Figure 12. Analysis Settings Tab in Project Settings Window

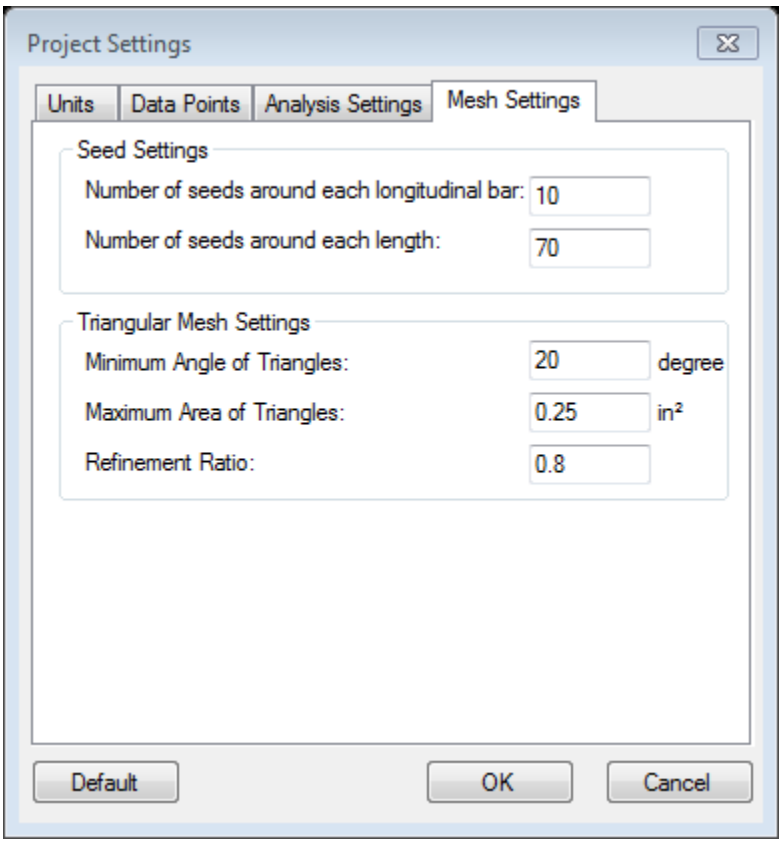


Mesh Settings

At the fourth and last tab of Project Settings window, the mesh settings can be defined. There are two parameters to control the seed settings and three parameters to control triangular mesh settings. In the Seed Settings group box, the first parameter is the number of seeds around each longitudinal reinforcement bar. To avoid any inaccuracy, the number of seeds around the longitudinal bars should be more than 4. The number of seeds in each length may vary based on the geometrical properties of the section. To set the properties of the triangular meshing function, there are three parameters that can be adjusted. The first parameter specifies the minimum angle of triangles. The second parameter is the maximum area of each triangle. And the last one is the

refinement ratio. When user clicks on the Refine Mesh button, the maximum area of triangles will decrease with the refinement ratio.

Figure 13. Mesh Settings Tab in Project Settings Window



Locking and Unlocking the Section

The RC section in CharAsad is locked against user changes when the section is meshed. The user can choose to unlock the section and delete the mesh to change the bar arrangement using the button shown above.

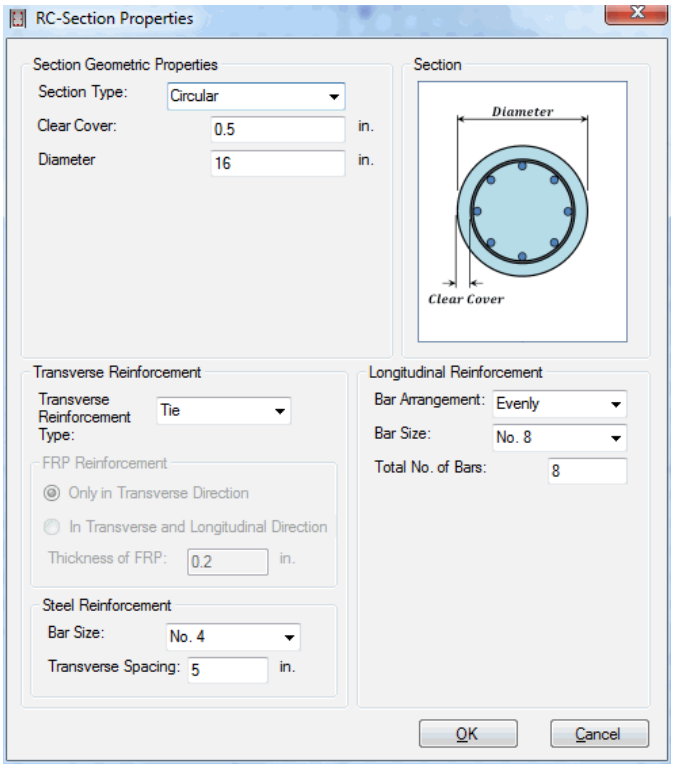
Pause/Stop Analysis

The user can pause or stop an analysis in progress. The analysis will be aborted if the user choses to push the stop analysis button. If the user choses to pause the analysis, the user can continue the analysis by clicking the continue button. This functionality is useful when observing the results as they are plotted during an analysis.

Basic Steps to Create a Project

A project is based around the definition of a reinforced concrete section by the user. The geometric and reinforcement properties of a section can be specified in the RC-Section Properties window (Figure 14). The composite section can be reinforced longitudinally by steel or FRP bars and laterally by conventional steel (tie or spiral), FRP wraps, or a combination of lateral steel and FRP wraps. The effect of lateral reinforcement and FRP wraps is indirectly taken into account in modeling because axial stress-strain behavior of the region enclosed by lateral reinforcement is affected by lateral reinforcement. FRP wraps can have stiffness in axial and lateral directions, depending on the orientation of FRP fibers.

Figure 14. RC Section Properties Window



Section Geometric Properties

9 options are currently available for the cross-sectional geometry of an RC section:

- Rectangular Section
- Hollow Rectangular Section
- Circular Section
- Hollow Circular Section
- Oval
- Hollow Oval
- Caltrans Section (Oval Section)
- T-Shape Section
- I-Shape Section

The clear cover dimensions from the edge of the section to the outer edge of the transverse reinforcement, and other necessary dimensions of the section, such as the diameter for a circular section, are defined by the user in Geometric Properties.

Transverse Reinforcement

The type of lateral reinforcement can be selected using the Transverse Reinforcement Type drop down button. The type of lateral reinforcement available for analysis is based on the cross section's shape. For example, circular and hollow circular sections can use one of the following types:

- Tie
- Spiral
- FRP
- FRP and Tie
- FRP and Spiral

Caltrans sections can use either ties or spiral reinforcement, and the other section types must use ties as transverse reinforcement. Future versions of KSU-RC will have the capability of using FRP reinforcement, with or without transverse steel, for rectangular and oval shapes. FRP reinforcement, steel reinforcement, or a combination of the two material property inputs is activated by the selection of the transverse reinforcement type. In the FRP reinforcement group box, the user can specify the thickness of FRP and specify if the FRP works in "Only Transverse Direction" which means it has lateral confining stiffness, or "In Transverse and Longitudinal Direction" which means FRP has the confinement effect for concrete as well as stiffness in longitudinal direction, similar to steel longitudinal bars.

Longitudinal Reinforcement

The quantity and size of longitudinal bars are defined in this group box. If the "Evenly Distributed" option is selected in the Bar Arrangement dropdown button, the program will arrange the location of bars automatically. Otherwise, the user should specify the size and location of bars graphically by clicking on the cross section using the CharAsad control.

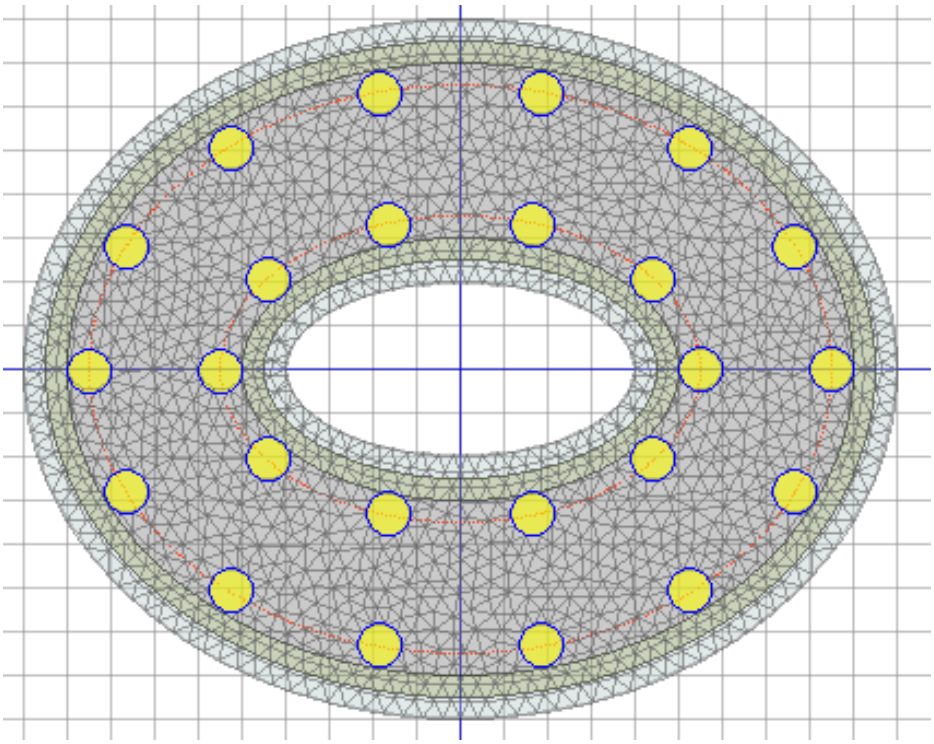
Deleting a Section

The defined reinforced concrete section and generated mesh are deleted using the button shown above. Note that all analysis results, material properties, and monotonic models will be removed and set to default values.

Creating a Finite Mesh

Before any analysis can be performed, the section should be divided into a number of fibers. The applicable material stress-strain behavior models are then used to model the behavior of each fiber during an analysis. KSU-RC 2.0 uses triangular cross section fibers. The size of the fibers in the section depends on user inputs for Mesh Settings. The Project Settings window contains options for the user to change triangulation settings, such as the number of seeds around each longitudinal bar in the section, the maximum area of the mesh triangles, and the refinement ratio. Meshing the section once is enough to run an analysis, but the mesh can be refined to make the size of the triangles smaller to make the results of the analysis methods more accurate. Once a mesh is applied to the section, the user can analyze the section using the default material properties and material models, or the user can input custom material properties and material models parameters. Material properties and material models options are discussed in the next section.

Figure 15. Triangulation or Meshing of a Hollow Oval Section



Material Properties

Default material properties are populated in the program upon creation of an RC section. An analysis can be run in the program once the section has been meshed, even if the user does not input material properties. Default and user defined material properties are discussed in this section.

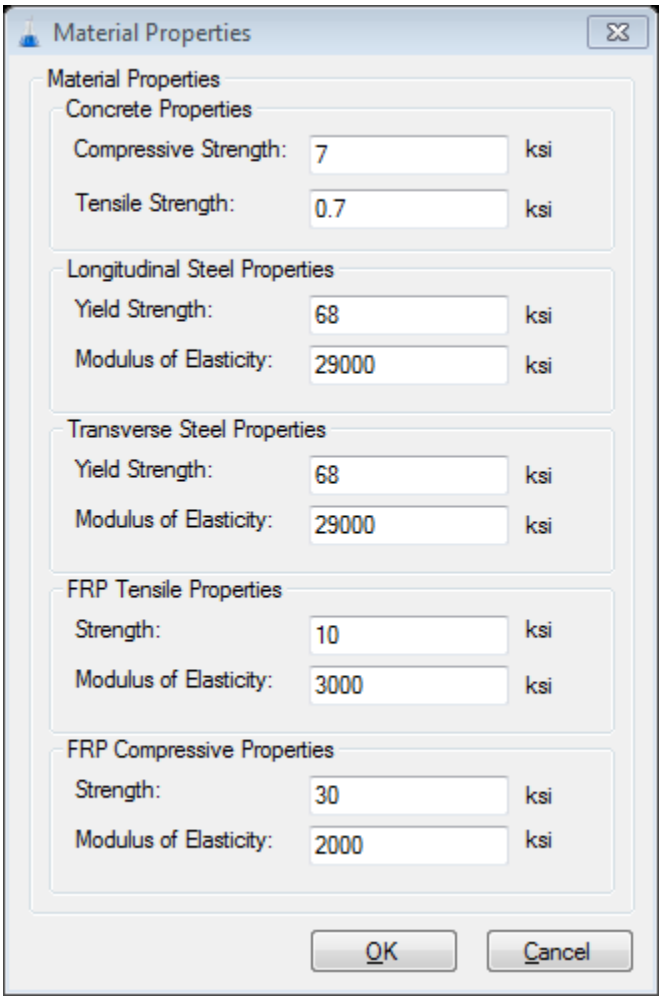
User Defined Material Properties

Mechanical properties of materials, including concrete, steel in longitudinal and lateral directions, and FRP are required inputs for the computer application. Concrete strength, as measured in the lab or desired for analysis, must be provided. For steel, yield strength and modulus of elasticity are provided as basic mechanical properties, assuming symmetric behavior for steel in tension and compression. The tensile and compressive strength and modulus of elasticity of FRP must be individually provided because the behavior of FRP wraps in tension and compression are generally not equal. These properties mentioned above can be changed in the Material Properties Window (Figure 16).

Default Material Properties

The default material properties for each material are shown in Figure 16.

Figure 16. Basic Material Properties Window



Monotonic Material Models

The behaviors of the constituent materials under monotonic stress-strain in an RC section are represented by numerical algorithms. These models were chosen based on research done by Esmaeily and Shirmohammadi, which was discussed in the Prior Work by Esmaeily, and the Prior Work by Shirmohammadi sections.

Steel Monotonic Models

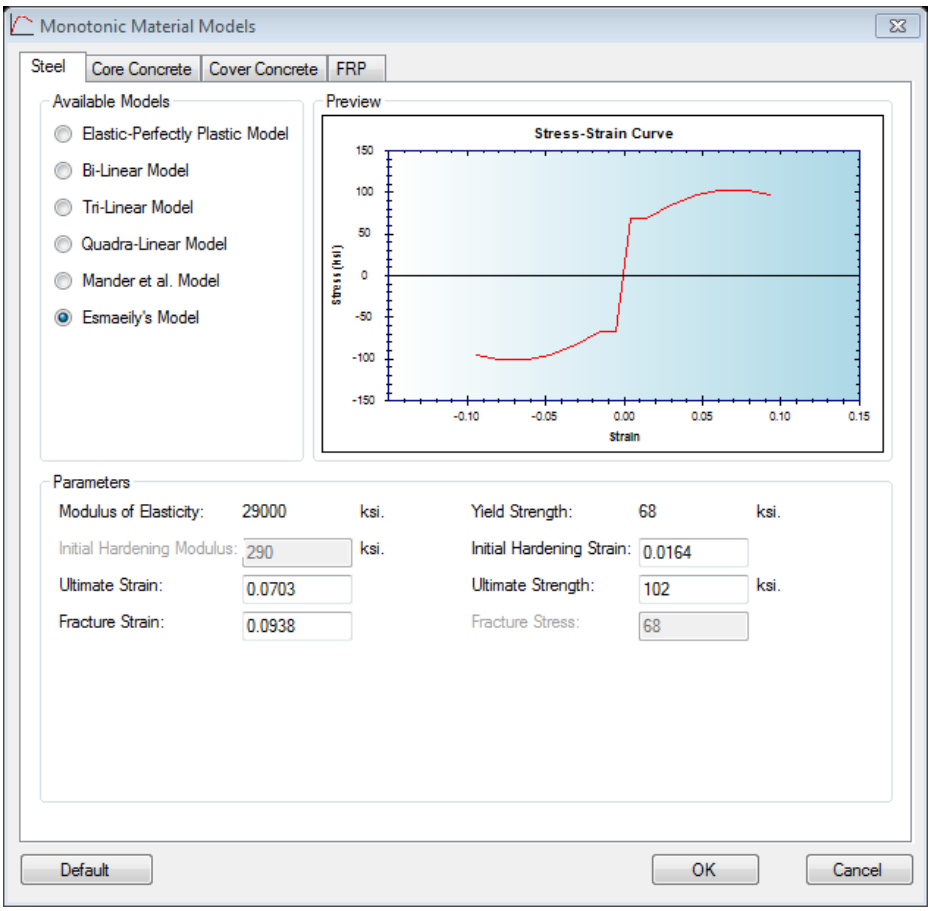
Uniaxial stress-strain behavior of mild steel reinforcement can be represented by analytical models. These models vary in complexity and their accuracy to the actual behavior of steel reinforcement under axial strain. The amount of laboratory testing data for the steel that is available generally governs which model is used. For example, if only yield strength and modulus of elasticity of the steel is known, the Elastic-Perfectly Plastic model should be used. One of six models can be chosen to represent the monotonic steel behavior:

- Elastic-Perfectly Plastic Model
- Bi-Linear Model
- Tri-Linear Model
- Quadra-Linear Model
- Mander's Model
- Esmaeily's Model

Note that FRP longitudinal reinforcement bars can be modeled in place of steel bars by using the Elastic-Perfectly Plastic Model, setting the Fracture Strain of the material to the Yield

Strain of the material, and also setting the longitudinal steel basic material properties to that of the FRP material.

Figure 17. Steel Monotonic Models Tab in Monotonic Material Models Window

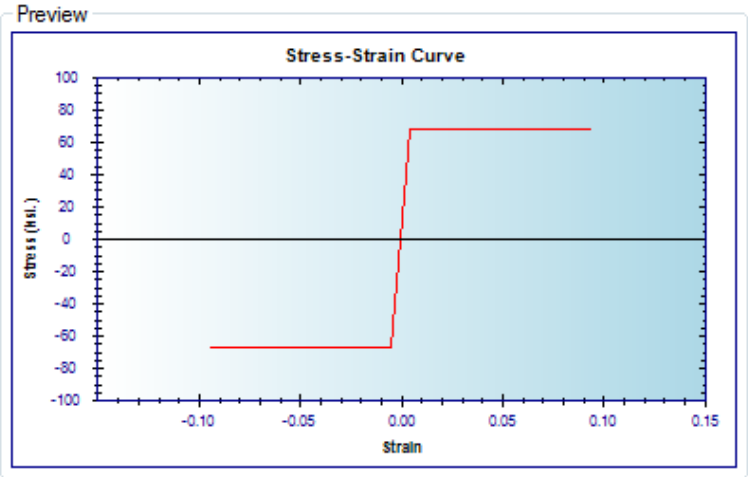


Elastic-Perfectly Plastic Model

When no information except yield strength and modulus of elasticity are available, the Plastic-Perfectly Plastic stress-strain model can be used for monotonic behavior of longitudinal

steel bars. Note that a default fracture strain is used for this model, but can be changed by the user.

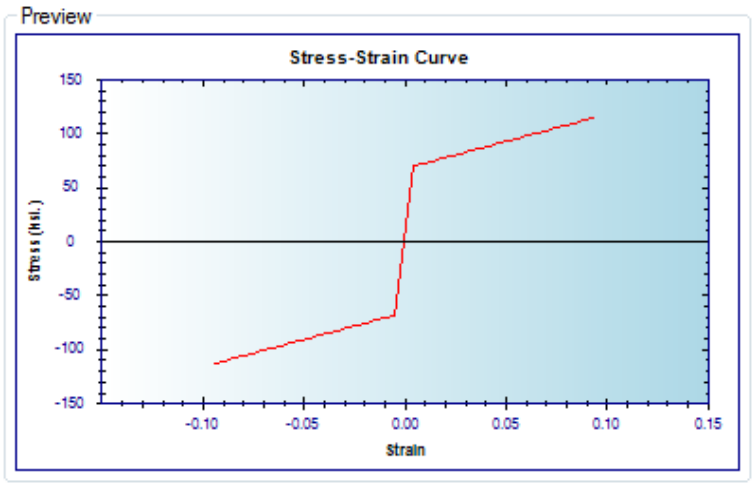
Figure 18. Elastic-Perfectly Plastic Stress-Strain Curve



Bi-Linear Model

The ultimate stress and strain of the steel are required to use the Bi-Linear model. This model uses the modulus of elasticity specified in the steel Material Properties as the slope of the curve until the yield strength. Then, the curve slope is changed to bring the end of the curve to the specified ultimate strain and ultimate stress.

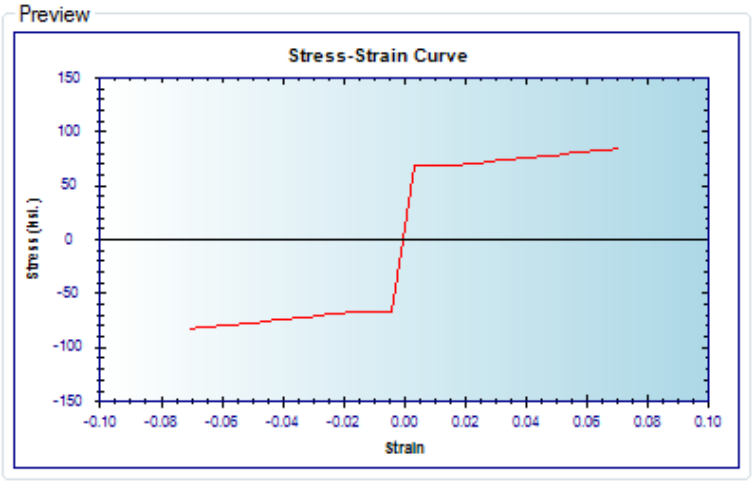
Figure 19. Bi-Linear Stress-Strain Curve



Tri-Linear Model

The Tri-Linear model uses the initial hardening modulus, initial hardening strain, and ultimate strain of the steel. These three values are used to model the steel behavior post-yield.

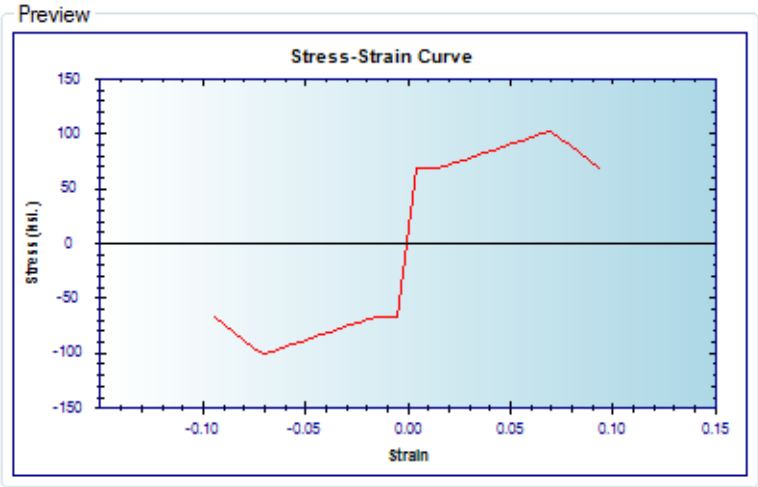
Figure 20. Tri-Linear Stress-Strain Curve



Quadra-Linear Model

The Quadra-Linear model follows linear elastic behavior of the steel up until the specified yield strength. The curve then assumes perfectly plastic behavior up until the initial hardening strain, where the curve slope brings the steel to ultimate stress and strain. After the ultimate strain, the stress in the steel decreases until the fracture strain and fracture stress are reached.

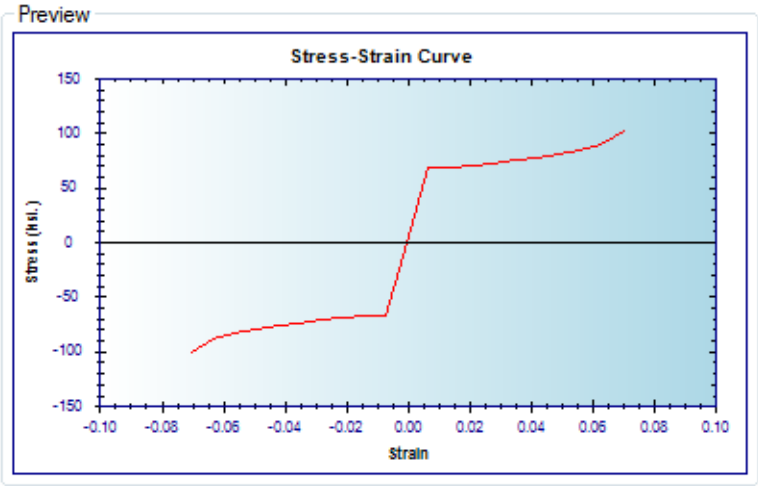
Figure 21. Quadra-Linear Stress-Strain Curve



Mander Model

The Mander model (Mander, et al., 1994) was developed as a result of a variety of tension and compression tests. This model takes into account elastic behavior, yield plateau, and strain hardening of the steel material.

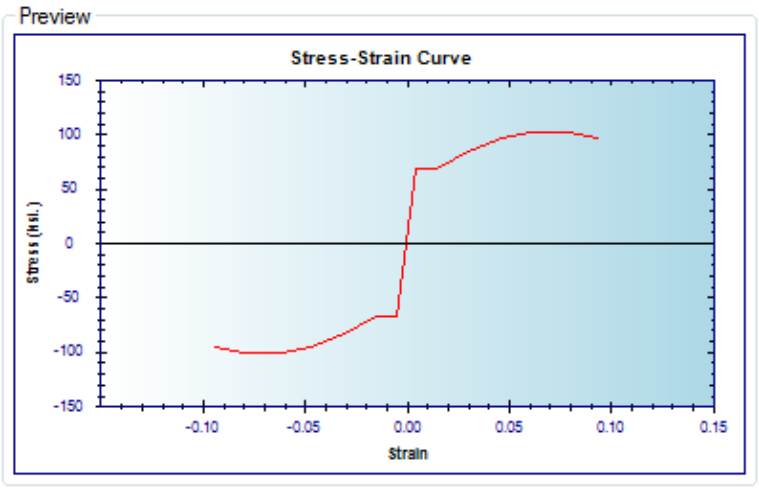
Figure 22. Mander Steel Model Stress Strain Curve



Esmaeily Model

Esmaeily's steel monotonic model can be employed to model behavior of longitudinal reinforcement when additional detailed information about reinforcing steel is available. Stress-strain behavior of various types of steel can be simulated using the yield plateau, strain hardening of the steel. The hardening and softening branch of steel monotonic model can be estimated using coefficients proposed by Esmaeily (2002) when no information is available except steel yield strength and modulus of elasticity. Based on tensile experiments of steel bars, Esmaeily proposed the ultimate strain and strength for steel to be equal to 24.9 times the yield strain, and 1.3 times the yield stress, respectively (Esmaeily, 2002).

Figure 23. Esmaeily Stress-Strain Curve



Core Concrete Monotonic Models

One of 8 material models can be chosen to represent the monotonic core concrete behavior. The models are representative of confined concrete stress-strain behavior. The confinement of concrete depends on the type of lateral reinforcement (steel ties, steel spirals, and FRP wraps). The models available for use in a project depend on the user input section shape, transverse reinforcement type, and concrete strength.

Concrete Confined by Lateral Steel Reinforcement

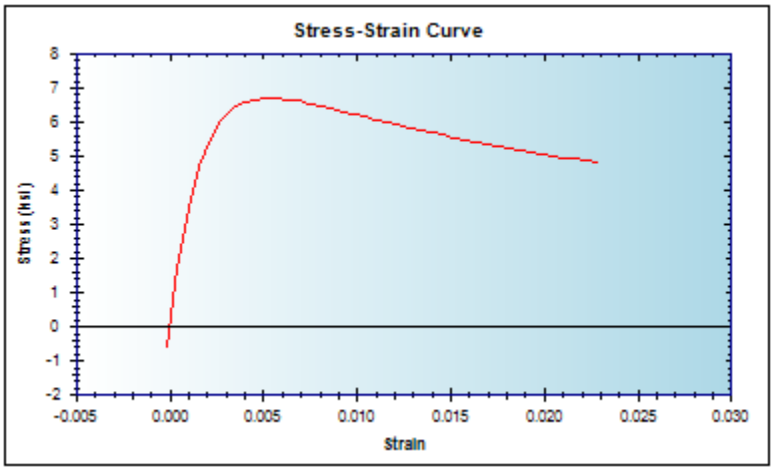
Three models are available to represent the behavior of concrete confined by lateral steel reinforcement.

Mander Model

For concrete confined by steel lateral reinforcement, the Mander et al. model (1988) for low to medium strength concrete was chosen for accuracy in predicting the behavior of circular

and rectangular concrete columns with various longitudinal and lateral steel configurations. The Mander et al. model (1988) was developed analytically for circular or rectangular cross sections under monotonic, cyclic static or dynamic loading. The RC section may have any general confinement type provided by either spiral or circular hoops, or rectangular hoops with or without cross-ties. In this model, the effect of any confinement type was taken into account by defining an effective lateral confining pressure that depends on lateral and longitudinal reinforcement configuration. To predict strain corresponding to first fracture, Mander et al. used an energy balance approach by equating strain energy stored in the concrete caused by confinement to strain energy capacity of the lateral reinforcement (Mander et al. 1988). The stress-strain relationship of the Mander et al. model is based on an equation proposed by Popovics (1973)

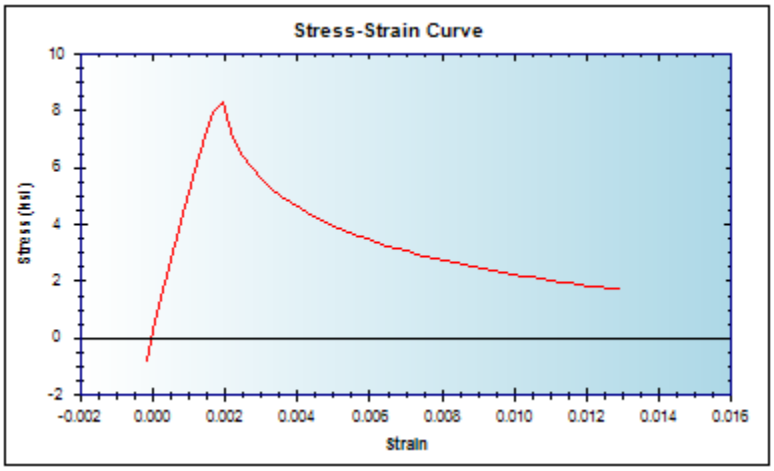
Figure 24. Mander Model for Confined Concrete



Cusson and Paultre Model

Because proposed stress-strain models for normal-strength concrete may overestimate the strength and fracture strain of high-strength concrete, Cusson and Paultre (1995) proposed their model to predict monotonic behavior of high-strength concrete confined by steel ties, using experimental results of 50 large-scale high-strength concrete tied columns tested under eccentric loading. Cusson and Paultre also considered effects of tie-yield strength, concrete compressive strength, tie configuration, and lateral and longitudinal reinforcement ratios when developing their model (Cusson and Paultre 1995). Cusson and Paultre's stress-strain curve for confined and unconfined concrete consists of two parts. The initial part is a relationship originally proposed by Popovics (1973), and the second part is a modification of the relationship proposed by Fafitis and Shah (Fafitis & Shah, 1985) for high-strength confined concrete. As reported by Cusson and Paultre (1995), yield strength of lateral reinforcement is developed at the peak strength of concrete only for well-confined high-strength concrete specimens; therefore, peak strength of confined concrete was computed by employing an iterative process in this model.

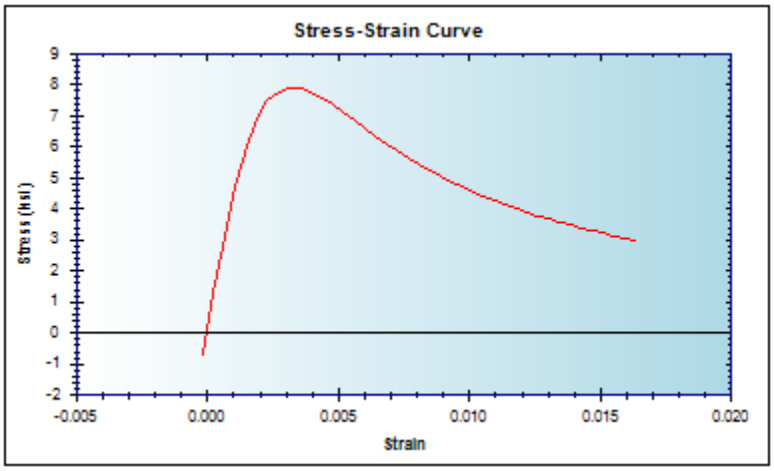
Figure 25. Cusson and Paultre Model for Confined Concrete



Campione and Fossetti Model

Campione and Fossetti (2007) created a simplified model to determine the stress-strain relationship for elliptical shaped concrete sections with steel hoop confinement. Effective stresses in the hoops are evaluated to predict the maximum confinement and the maximum compressive strength of the confined concrete. The proposed stress-strain relationship follows a similar equation to that of Mander (1988), with the consideration of the geometry of the section in the calculation of effective confinement of the concrete from the steel hoops. This model can be used for circular columns, since a circle is an ellipse with equal dimensions in both height and width.

Figure 26. Campione and Fossetti Model for Confined Concrete



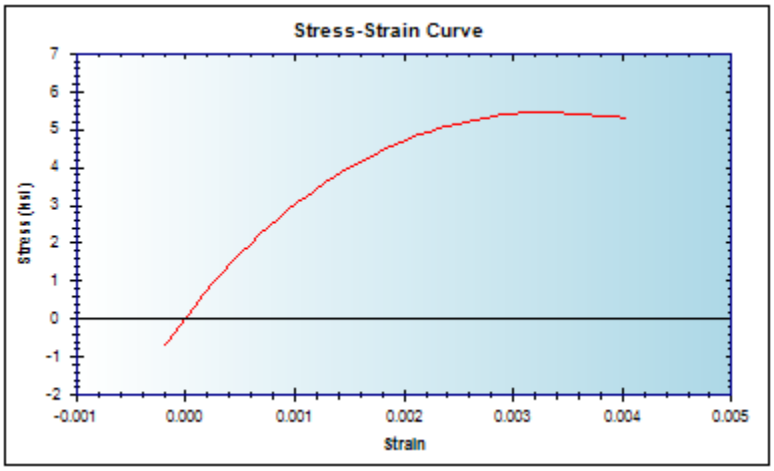
Concrete Confined by FRP

Two models are implemented in the program to model concrete confined by Fiber Reinforced Polymers (FRP).

Youssef Model

Youssef et al. (2007) used results of an experimental program that included large-scale circular, rectangular, and square short columns confined by carbon-FRP and E-glass-FRP wraps in order to develop a stress-strain model for concrete confined by FRP. This model is applicable for predicting monotonic behavior of low- to high-confined concrete. The initial relationship of this model is a polynomial function followed by an ascending or descending linear part that represents low- and moderate- to high-strength confinement, respectively.

Figure 27. Youssef Model for Confined Concrete

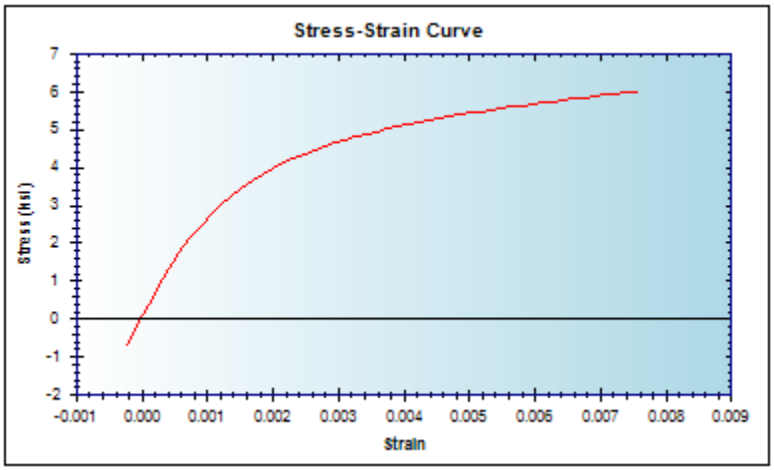


Samaan Model

Samaan et al. developed a bilinear stress-strain model for FRP-confined concrete based on 30 cylindrical specimens tested under uniaxial compression loading (Samaan, et al., 1998). They used a four-parameter relationship to model initial behavior of FRP-confined concrete.

Calculating the fracture strain and its corresponding stress, the initial curve is followed by a line connecting the initial part to the fracture point.

Figure 28. Samaan Model for Confined Concrete



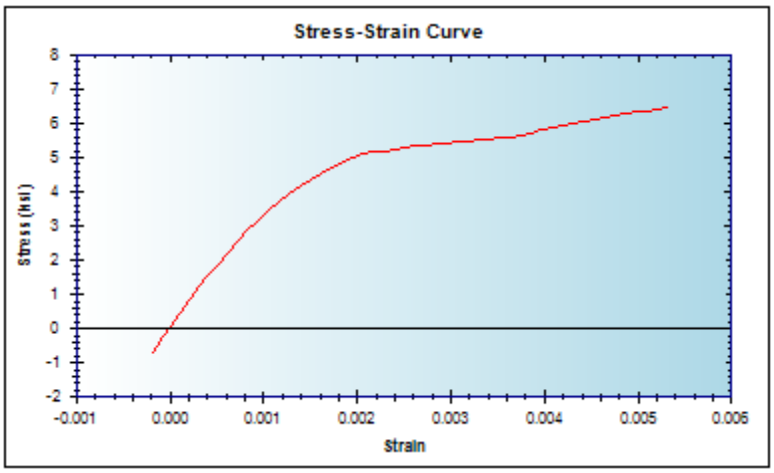
Concrete Confined by FRP and Conventional Steel

Three models are available to model the behavior of concrete confined by the combination of FRP and steel ties or FRP and steel hoops.

Lee Model

Lee, et al. (2009) presented a comprehensive stress-strain model for concrete confined internally by lateral steel and externally by FRP wraps based on experimental results of 24 concrete cylinders subjected to compression. The Lee et al. model consists of three polynomials: a second-order polynomial function, a polynomial at strain corresponding to plain concrete strength, and a polynomial at the point representing the lateral steel yield point (Lee, et al., 2009).

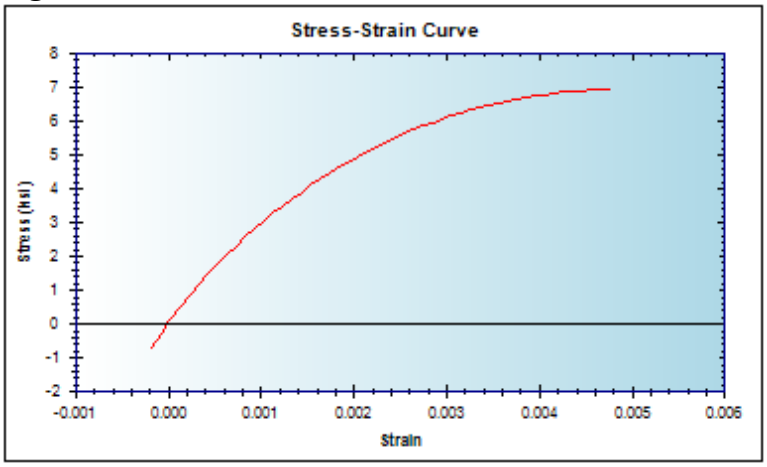
Figure 29. Lee Model for Confined Concrete



Kawashima Model

Kawashima et al. stress-strain model was developed by combining a model for concrete confined by steel tie reinforcement and a model for concrete confined by carbon-FRP. The initial part of this model is a polynomial followed by a linear path. The linear part of this model can be ascending or descending depending on the confinement ratio of the concrete section (Kawashima, et al., 2000).

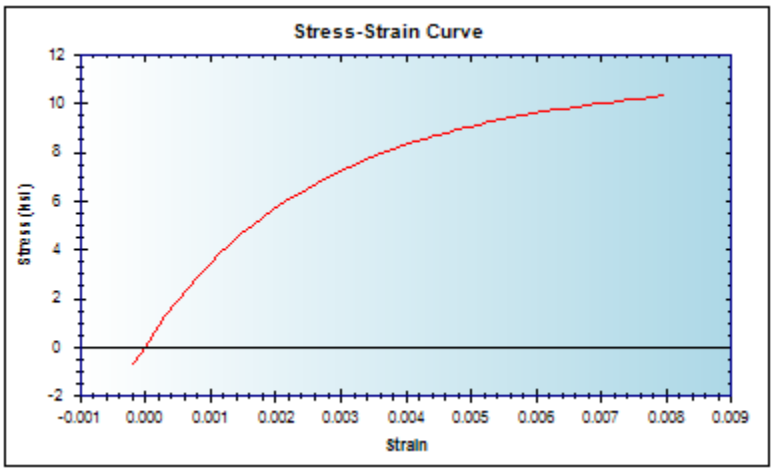
Figure 30**. Kawashima Model for Confined Concrete**



Shirmohammadi Model

Shirmohammadi et al. (2015) developed a constitutive stress-strain relationship to model the behavior of concrete confined by FRP and lateral steel. Using experimental data, Shirmohammadi et al. (2015) proposed two equations for ultimate strain and corresponding stress for FRP and steel confined concrete.

Figure 31. Shirmohammadi Model for Confined Concrete



Custom Model

In addition to the models mentioned above, a Custom Model option is included in the program. Using the Custom Model functionality, users can implement and use their own model, including linear and second-order polynomial segments. The Custom Model may include up to five segments, and each segment may be linear or polynomial.

Note

Concrete tensile strength is assumed to be linear with a slope equal to modulus of elasticity of plain concrete in all aforementioned monotonic models.

Cover Concrete Monotonic Models

One of 5 models can be chosen to represent the monotonic cover concrete behavior. The models available for use in a project depend on the user input transverse reinforcement type and concrete strength. The list below shows the available models grouped by aforesaid variables:

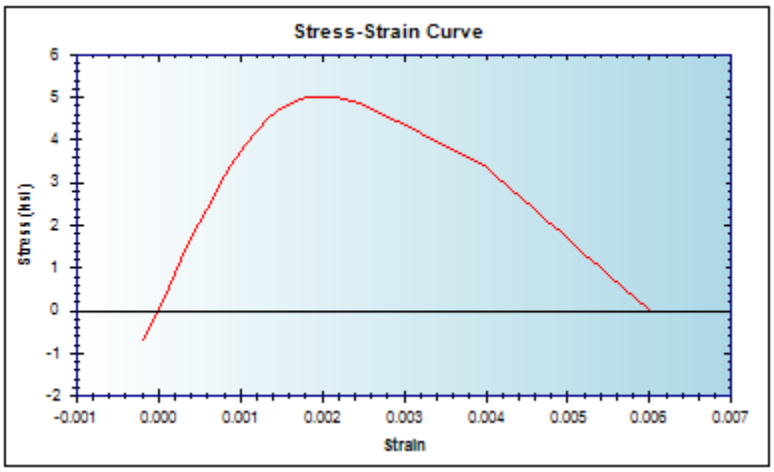
Plain or Unconfined Concrete

Two models can be used to model the behavior of unconfined concrete: The Mander Model or the Cussan and Paultre Model.

Mander

For plain concrete, the Mander et al. model (1988) for low to medium strength concrete was chosen for accuracy in predicting the behavior of circular and rectangular concrete columns. The user is allowed to change the strain corresponding to the ultimate stress and the spalling strain of the unconfined concrete. The Mander et al. model (1988) was developed analytically for circular or rectangular cross sections under monotonic, cyclic static or dynamic loading. The stress-strain relationship of the Mander et al. model is based on an equation proposed by Popovics (1973). Many researchers have used Mander et al. model to simulate model monotonic behavior of plain concrete.

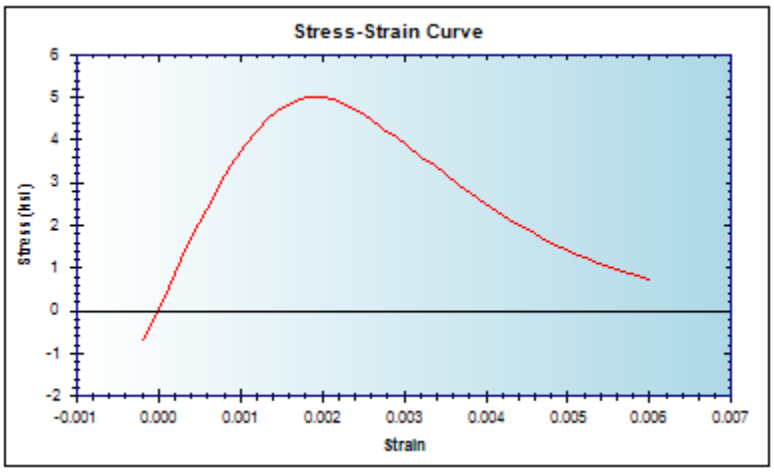
Figure 32. Mander Model for Unconfined Concrete



Cusson and Paultre

Cusson and Paultre (1995) proposed their model to predict monotonic behavior of high-strength concrete because proposed stress-strain models for normal-strength concrete may overestimate the strength and fracture strain of high-strength concrete. The user can change the strain corresponding to the ultimate strength of the concrete and the strain in the unconfined concrete when the stress drops to half of the ultimate strength of the concrete.

Figure 33. Cusson and Paultre Model for Unconfined Concrete



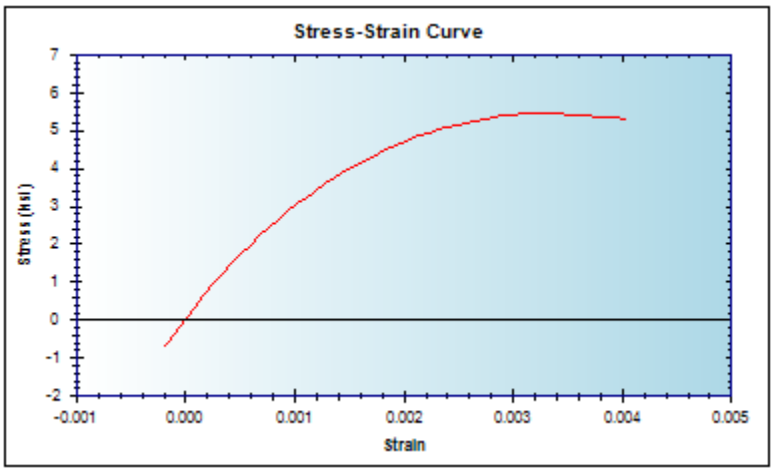
Concrete Confined by FRP

Two models are available to model the material behavior of concrete confined by FRP: The Youssef Model and the Samaan Model.

Youssef Model

Youssef et al. (2007) used results of an experimental program that included large-scale circular, rectangular, and square short columns confined by carbon-FRP and E-glass-FRP wraps in order to develop a stress-strain model for concrete confined by FRP. This model is applicable for predicting monotonic behavior of low- to high-confined concrete. The initial relationship of this model is a polynomial function followed by an ascending or descending linear part that represents low- and moderate- to high-strength confinement, respectively.

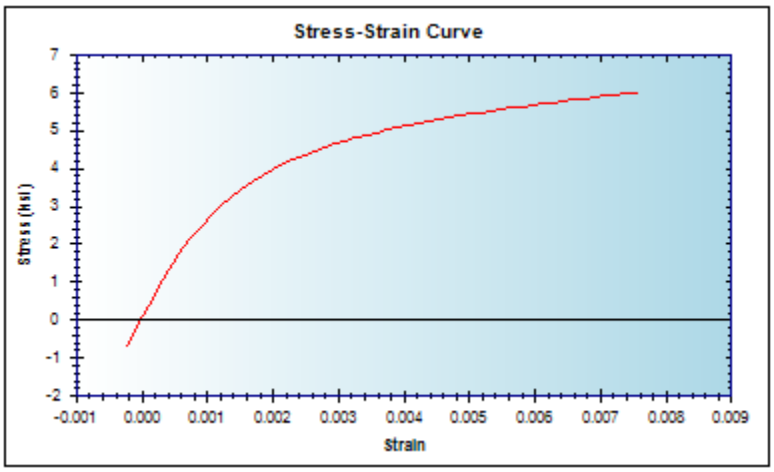
Figure 34. Youssef Model for Confined Concrete



Samaan Model

Samaan et al. developed a bilinear stress-strain model for FRP-confined concrete based on 30 cylindrical specimens tested under uniaxial compression loading (Samaan et al. 1998). They used a four-parameter relationship to model initial behavior of FRP-confined concrete. Calculating the fracture strain and its corresponding stress, the initial curve is followed by a line connecting the initial part to the fracture point.

Figure 35. Samaan Model for Confined Concrete



Custom Model

In addition to these models, a Custom Model option is included in the program. Using the Custom Model functionality, users can implement and use their own model, including linear and second-order polynomial segments. The Custom Model may include up to five segments, and each segment may be linear or polynomial.

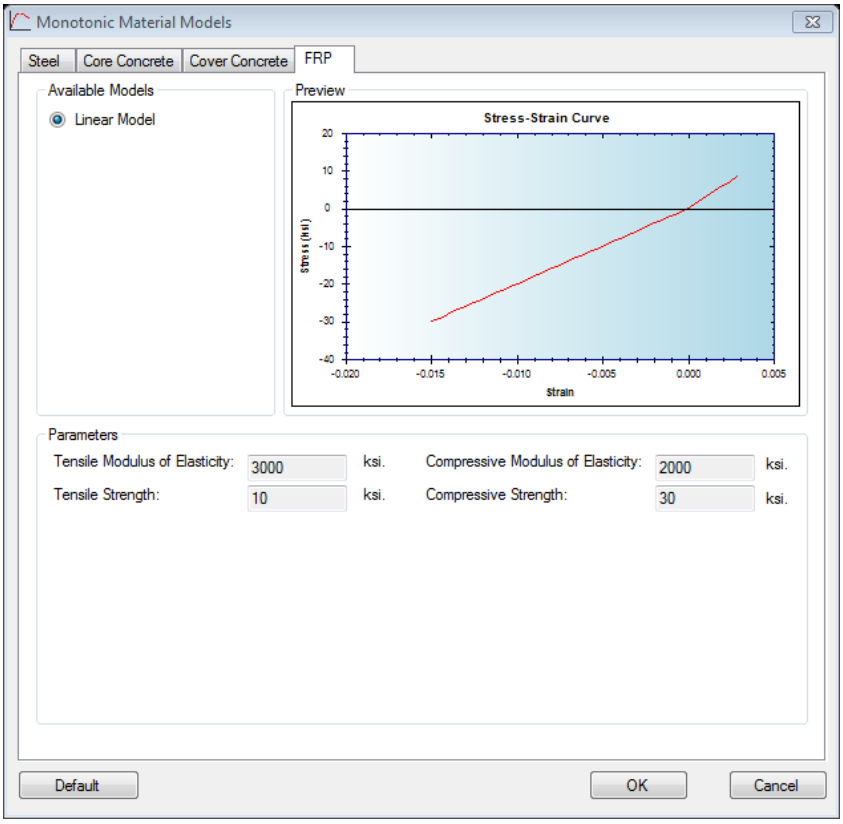
Note

Concrete tensile strength is assumed to be linear with a slope equal to modulus of elasticity of plain concrete in all aforementioned monotonic models.

FRP Monotonic Models

A bilinear model is used in KSU-RC 2.0 to model the uniaxial stress-strain behavior of Fiber Reinforced Polymers (FRP). The slope of the tensile and the slope of the compressive branches were equal to the tensile modulus and compressive modulus of elasticity of, respectively, as specified in the Basic Material Properties window. The monotonic behavior of FRP is used to represent the longitudinal strength of FRP wraps when applicable. The material properties for FRP, cannot be changed in the Monotonic Models Window, and must be changed in the Basic Material Properties window.

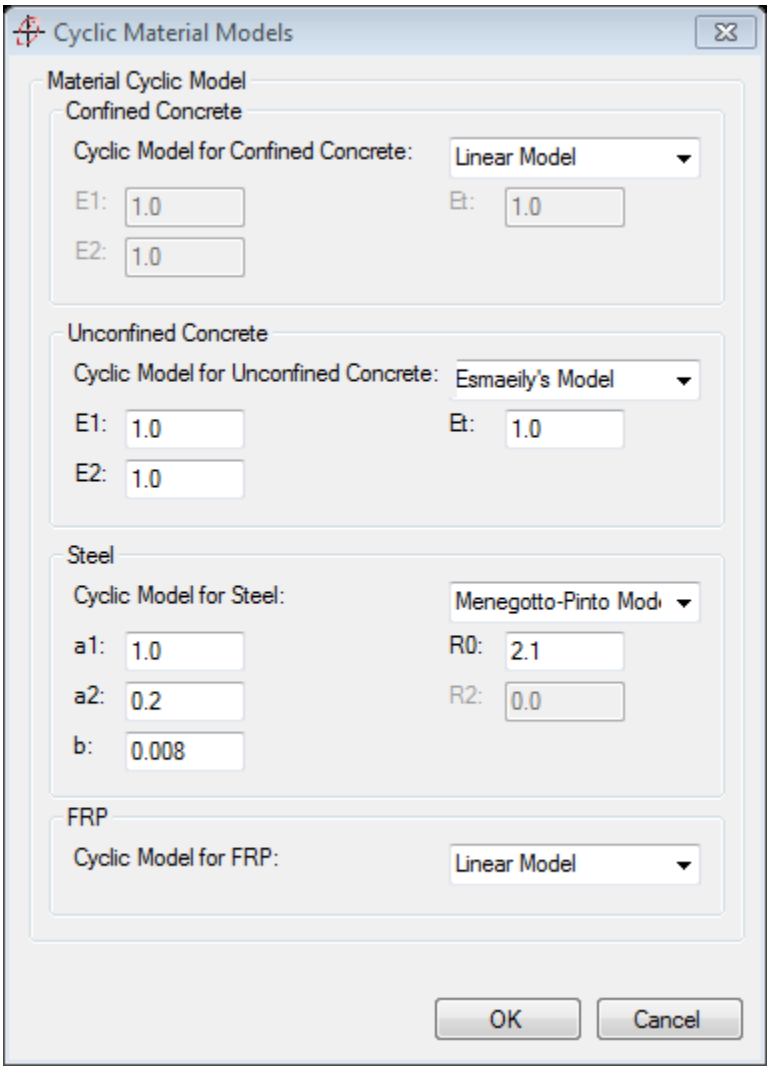
Figure 36. FRP Tab in the Monotonic Material Models Window



Cyclic Material Models

Various cyclic models are used to model cyclic behavior of materials in a composite section. The cyclic models available in KSU RC 2.0 are described in this section. A sample form of cyclic rules is shown in Figure 37.

Figure 37. Cyclic Material Model Window



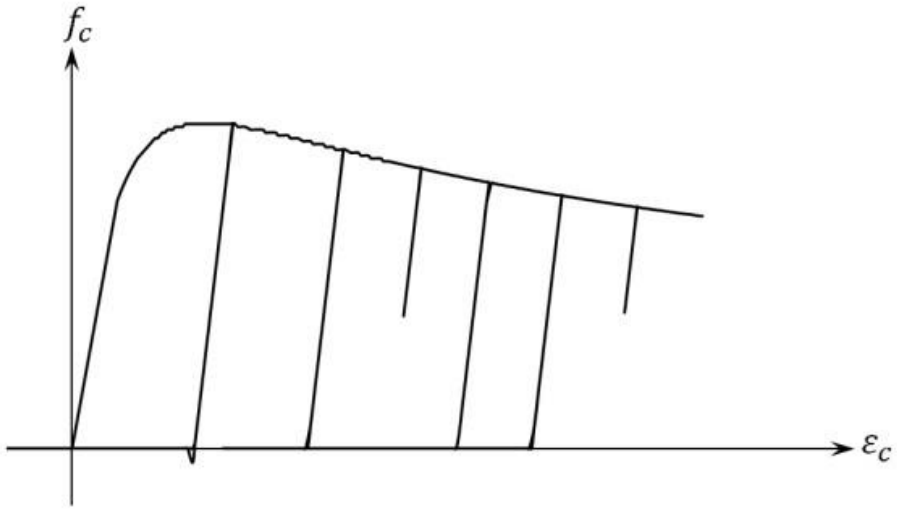
Concrete Cyclic Models

For cyclic behavior of concrete Linear, Mander et al., and Esmaeily cyclic rules were implemented in KSU-RC 2.0. These cyclic models can work with any monotonic model as an envelope curve. Cyclic behavior of concrete confined by FRP or lateral steel and FRP can be modeled by a linear cyclic model with a slope equal to the modulus of elasticity of plain concrete. Mander et al. (1988) and Esmaeily (2005) cyclic models were originally developed to model cyclic behavior of concrete confined by lateral steel; however, they can also be applied to model cyclic behavior of concrete confined by FRP or FRP and lateral steel.

Linear Model

Cyclic behavior of confined concrete can be modeled using a linear model with a slope of unloading and loading paths equal to the modulus of elasticity of concrete.

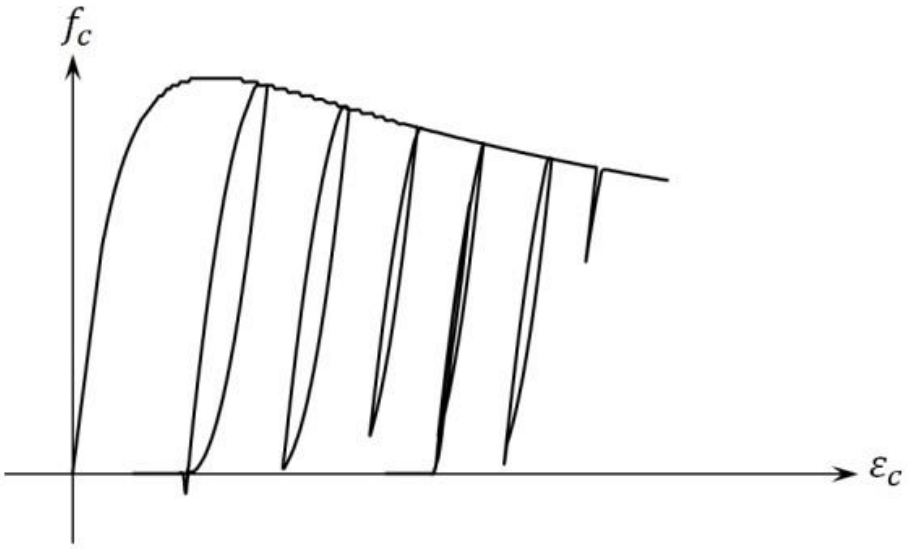
Figure 38. Linear Concrete Cyclic Curve



Mander Model

In the Mander et al. cyclic model, the unloading path follows a concave-upward parabolic path with a zero-slope at the strain-axis. Tensile strength of concrete can be taken into account considering a linear path with a slope of plain concrete modulus of elasticity. With increased strain, stress remains zero up to the last strain corresponding to the zero stress, after which point strain grows in a linear reloading path with a slope equal to plain concrete modulus of elasticity in the strain-axis (Mander et al. 1988).

Figure 39. Mander Concrete Cyclic Curve

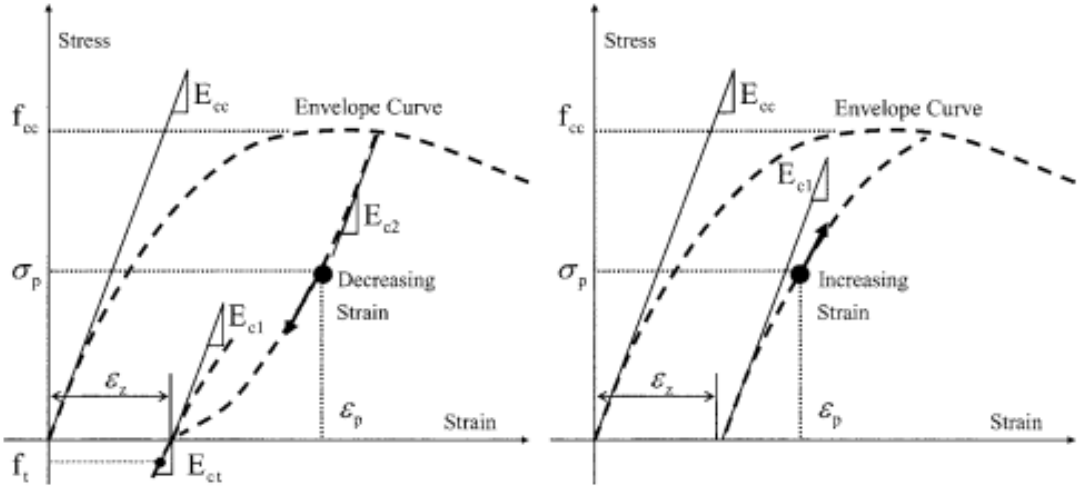


Esmaeily Model

In Esmaeily's cyclic model, the unloading path follows a parabolic path which is concave-upward with a slope of E_{c2} on the envelope curve (monotonic curve). This model may account for the tensile strength of concrete. With decreased strain at the unloading path, stress decreases to zero, after which point, if the tensile strength is ignored, stress remains zero; otherwise, stress decreases to tensile strength using a linear function with a slope of E_{ct}. With

increased strain, stress remains zero up to the latest strain corresponding to zero stress and then stress increases, following a concave-downward parabolic with a slope of E_{c1} at the strain-axis (Esmaeily, 2005).

Figure 40. Esmaeily Concrete Cyclic Curve: Unloading and Loading Segments



Steel Cyclic Models

A Linear model, Menegotto-Pinto's model (1973)and Esmaeily's model (2005) were implemented into the developed computer program to model the cyclic behavior of steel reinforcement.

Linear Model

The linear model uses a slope of the unloading and loading paths equal to the modulus of elasticity of the steel.

Esmaeily Model

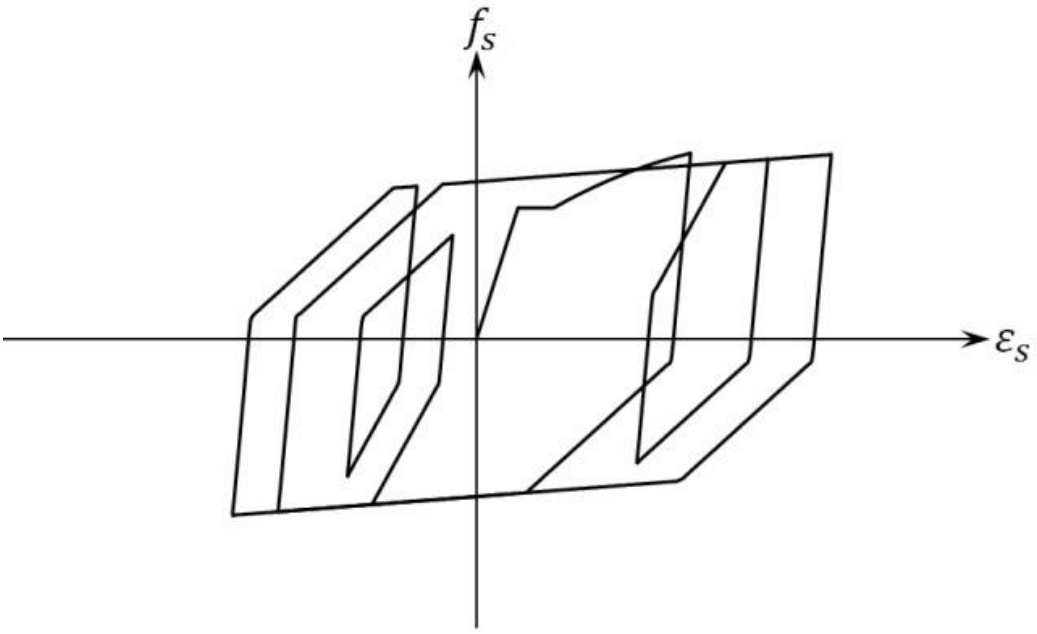
Esmaeily's hysteretic model for steel is a multilinear model. At the reversal point, the unloading path is a linear function with a slope equal to modulus of elasticity of steel material. The Bauschinger effect is taken into account in this model by changing the slope of the first unloading part into a portion of steel's modulus of elasticity beyond a certain stress (Esmaeily, 2005). In order to realistically simulate cyclic behavior of steel material, this ratio and the strain at which the slope change occurs are different in the second (tensile strain and compressive stress) and fourth (compressive strain and tensile stress) quarters from their values in the first (tensile strain and stress) and third (compressive strain and stress) quarters. Esmaeily's model uses five ratios (P_1, P_2, P_3, R_1, and R_2) to change the hysteretic behavior of steel material. Unlike Menegotto-Pinto's model, linear and Esmaeily's cyclic models can be used in conjunction with any steel monotonic model as an envelope curve. In Esmaeily's model the following relations should be followed to have a numerical stability:

$$0 < P_1 < P_2 < 1.0$$

$$0 < R_2 < R_1 < 1.0$$

$$0 < P_3 < 1.0$$

Figure 41. Esmaeily Steel Cyclic Model



Menegotto-Pinto Model

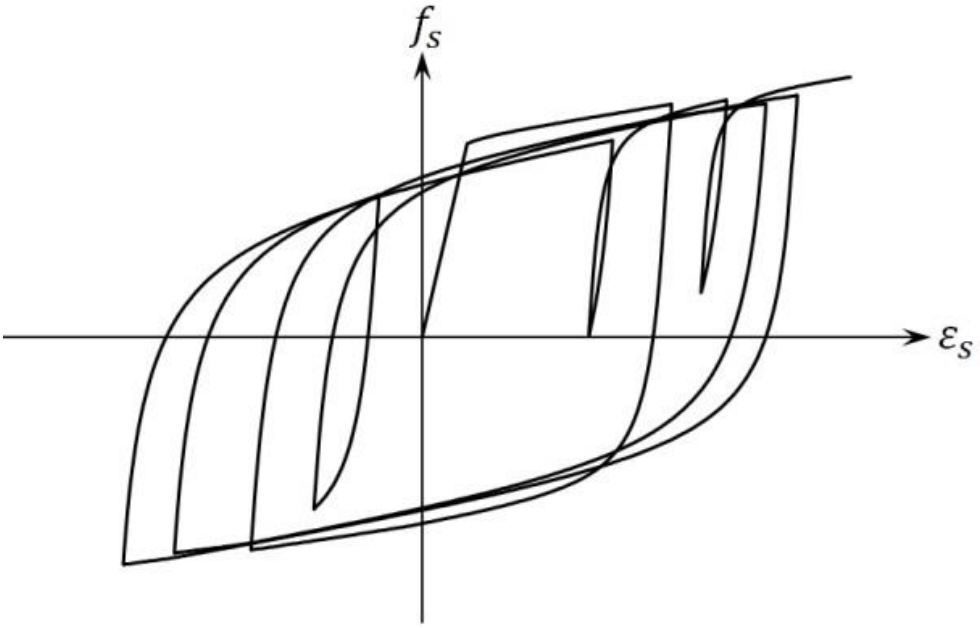
Menegotto-Pinto's model has a bilinear backbone curve. Cyclic response of steel material was defined using a nonlinear equation. The shape of unloading and reloading curves are defined by four parameters, R_0, a_1, a_2 and b. Because of numerical stability and realistic predictions, many researchers have used this model as a basis to propose new models for steel material. The parameter R_0 should be less than 20.0 and greater than 2.0 to have numerical stability in this model. The following limitation should be followed for the same reason:

$0 < a_1$

$0.1 < a_2$

$0 < b < 0.5$

Figure 42. Menegotto-Pinto Steel Cyclic Model



FRP Cyclic Models

Linear Model

KSU-RC 2.0 uses a linear model to represent cyclic behavior of FRP.

Graphic Test of Cyclic Models

A test of a cyclic material model can be performed using the Cyclic Model Test function; the user can chose to test the steel, confined concrete, or plain concrete models. First, click in the test window, then move the mouse to the right and left. The strain is calculated based on the location of the mouse. Using the selected cyclic and material models and having strain calculated from the location of mouse, the stress is calculated. In each strain the stress is shown in the

textbox below left of the window. To restart the form, click the “Restart” button. When the Restart button is clicked the form will be cleaned and the model can be tested again by moving the mouse to the left and right. To save the drawn cyclic path as a text file, click the “Save” button.

Figure 43. Test of Steel Cyclic Model

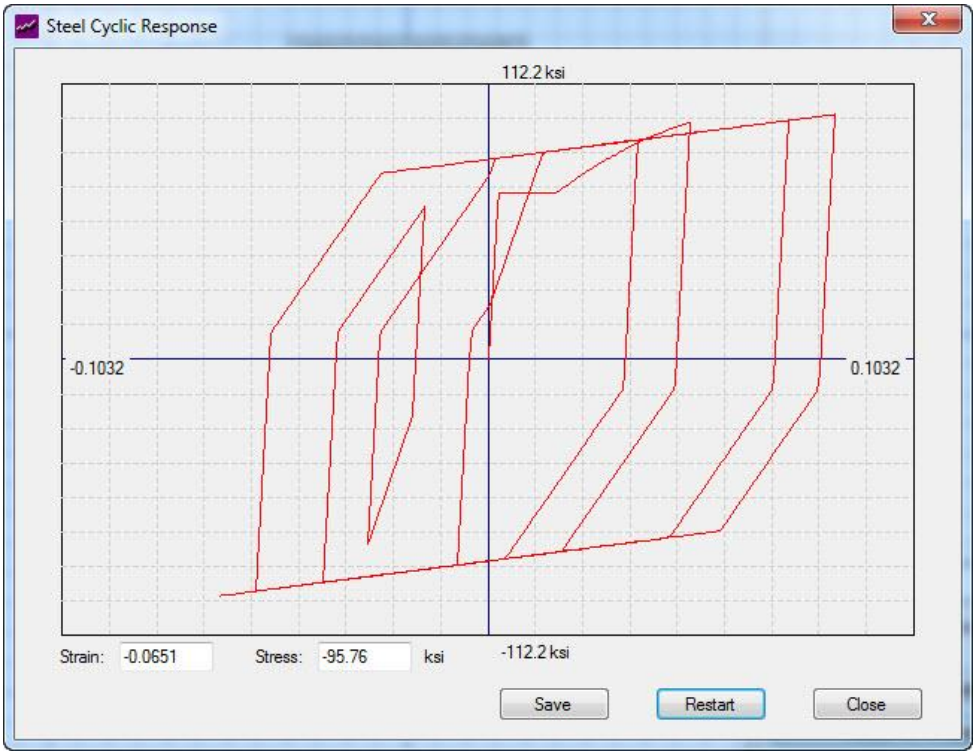


Figure 44. Test of Core Concrete Cyclic Model

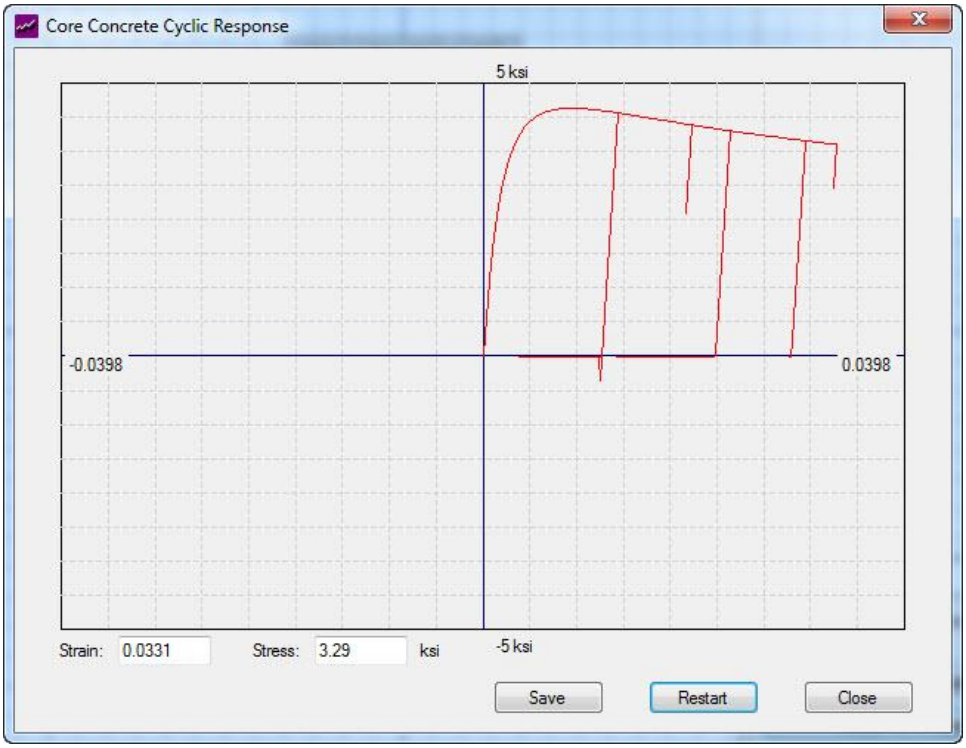
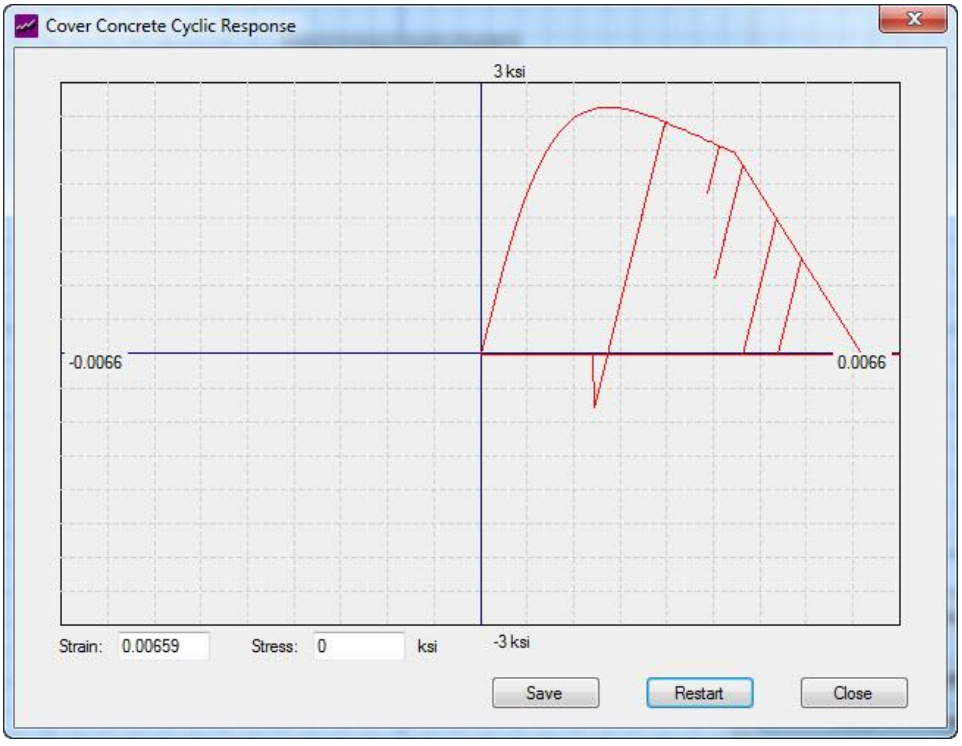


Figure 45. Test of Cover Concrete Cyclic Model



Analysis Methods

The main purpose of KSU RC 2.0 is to analyze reinforced concrete sections. Performance analysis can be run once a section and its related mesh, material properties, and material behavior models have been input into the program by the user. The three measures of the performance of the concrete section in KSU RC 2.0 are axial force-bending moment interaction analysis, moment-curvature analysis, and force-deflection analysis.

Axial Force-Bending Moment Interaction

Four types of Interaction Analysis Methods are available in KSU RC 2.0. These methods include ACI code-base interaction, interaction based on actual material properties, interaction based on moment curvature analysis, and interaction considering load pattern.

ACI Interaction Curves

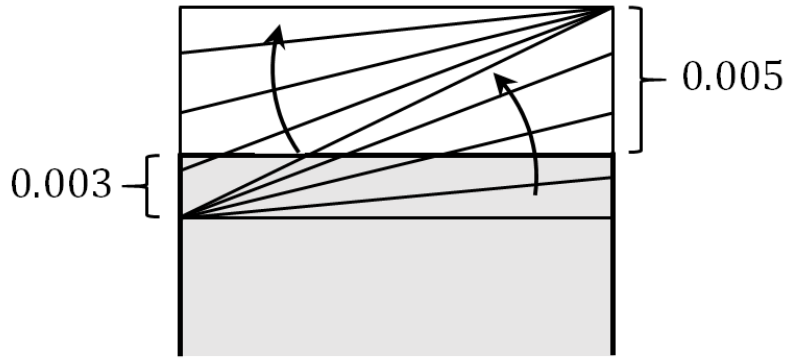
The ACI Interaction curve is calculated considering concrete equivalent stress block parameters. The steel behavior is considered as Elastic-Perfectly Plastic. This type of interaction curve is available only for sections reinforced laterally by internal steel ties or spirals, and longitudinally by steel bars. Two methods are available in KSU-RC 2.0 to calculate the ACI Interaction Curve. In both methods, strain at the extreme concrete fiber is equal to 0.003 initially.

Method 1

In the first method, the strain at the extreme concrete fiber is equal to 0.003. The curvature along the analysis axis is increased from zero to the point that strain at the extreme

tensile steel bars becomes equal to 0.005. Then the curvature is decreased in a way that strain at the extreme steel fiber remains 0.005.

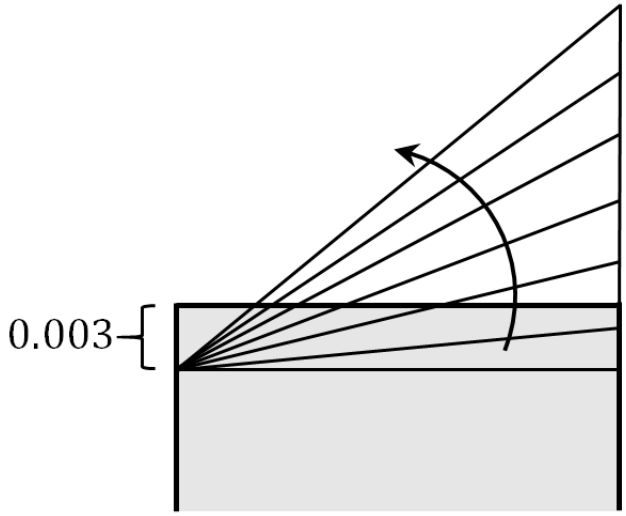
Figure 46. Curvature Change in Method 1



Method 2

In the second method, the strain at the extreme concrete fiber is kept constant and equal to 0.003 and the curvature will increase to the point when the section fails.

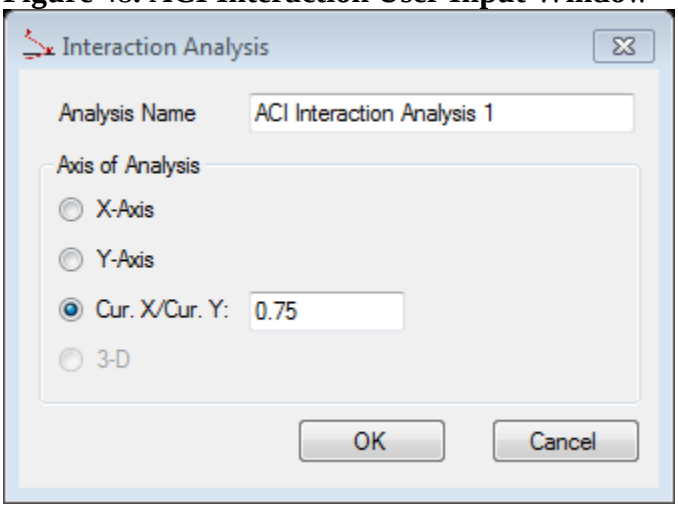
Figure 47. Curvature Change in Method 2



User Input

The user should specify the axis about which the analysis will be performed. Note that future versions of KSU-RC will have the capability of producing a 3D interaction failure surface. An optional user input is the Analysis Name. Analysis names can be used to distinguish between analyses based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-Int-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. KSU-RC 2.0 automatically populates the Analysis Name text box with a name for each analysis that is performed.

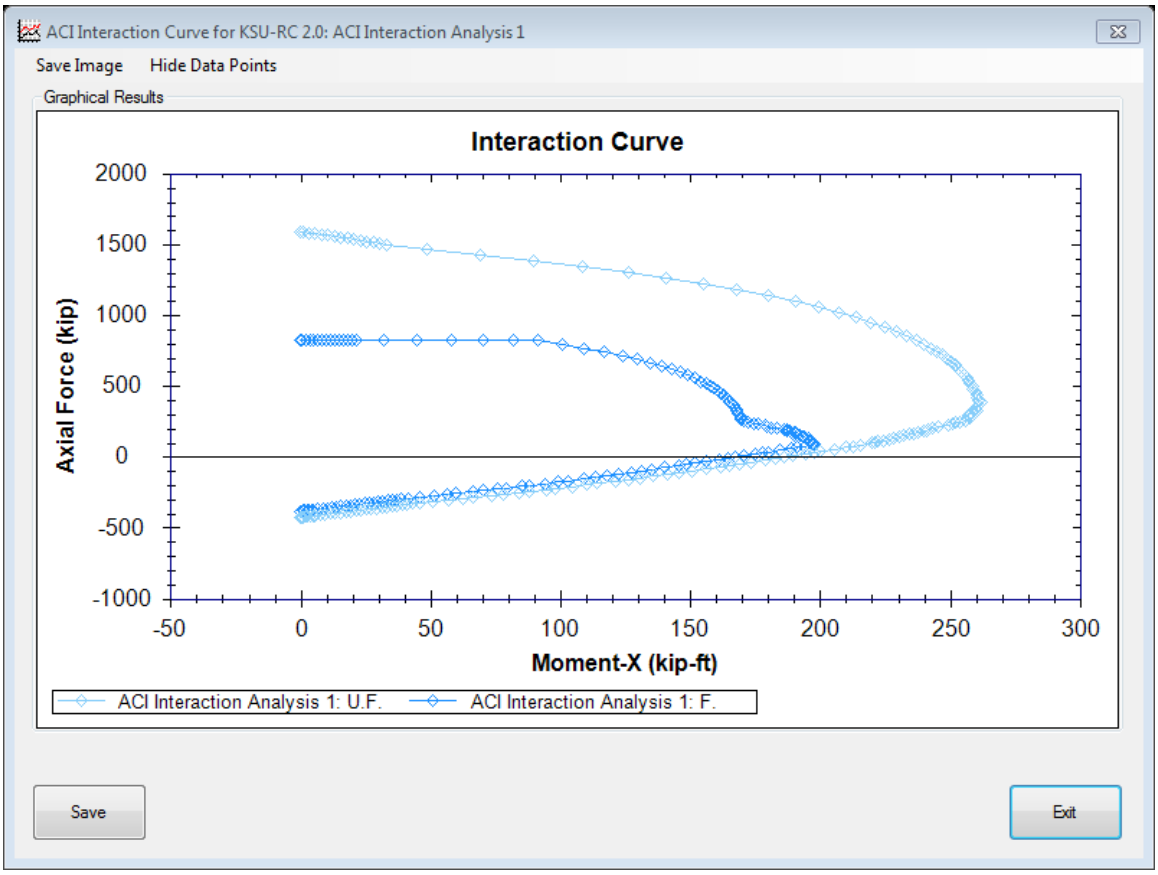
Figure 48**. ACI Interaction User Input Window**



Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the factored and un-factored moment in each analysis direction, factored and un-factored axial force, depth to neutral axis (c), and curvature in each analysis direction for each point calculated in the analysis.

Figure 49. ACI Interaction Curve Produced by KSU-RC 2.0



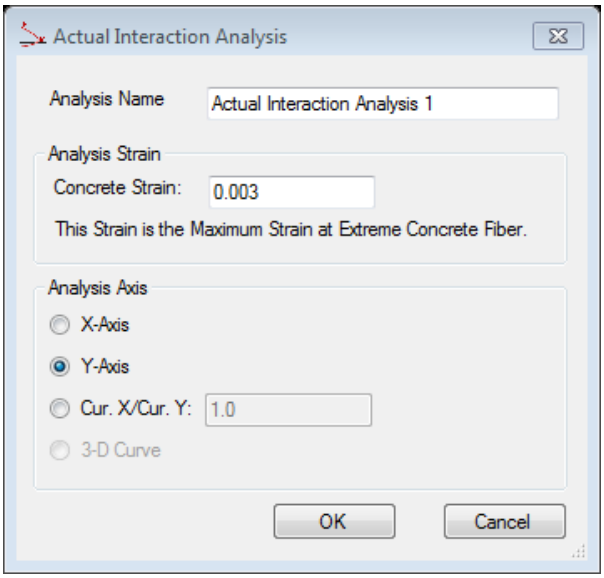
Interaction Using Material Models

Another type of axial force-moment interaction surface is calculated considering user-selected material models for plain concrete, confined concrete, reinforcing steel, and FRP. This type of interaction surface can be calculated for a concrete section confined by conventional steel (tie/spiral), FRP wraps, or FRP wraps and conventional steel. The strain at the extreme compressive fiber is assumed to be constant and can be defined by the user. In a moment-curvature analysis using material models, the curvature path along the x- and y-axis is known and the ultimate bending moments along the x- and y-axis are calculated using material models.

User Input

The user should specify the constant strain at the extreme concrete fiber for the analysis, as well as the axis about which the analysis will be performed. Note that future versions of KSU-RC will have the capability of producing a 3D interaction failure surface. An optional user input is the Analysis Name. Analysis names can be used to distinguish between analyses based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-Int-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. KSU-RC 2.0 automatically populates the Analysis Name text box with a name for each analysis that is performed.

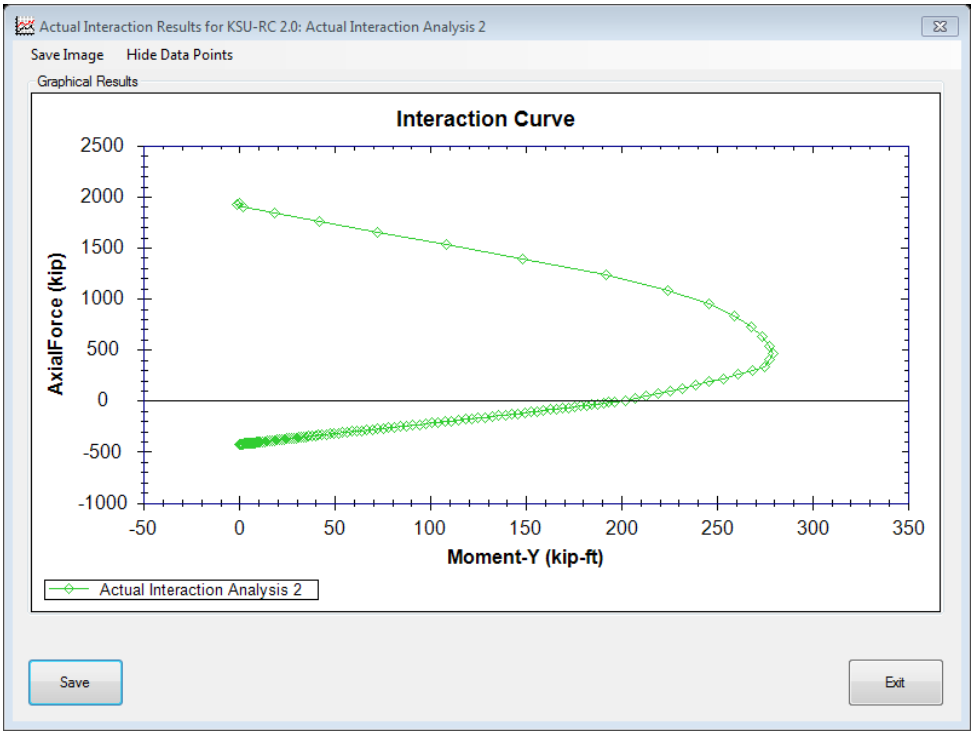
Figure 50. Interaction Analysis Input Window



Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the moment in each analysis direction, axial force, depth to neutral axis (c), and curvature in each analysis direction for each point calculated in the analysis.

Figure 51. Interaction Curve Considering Material Models Produced by KSU-RC 2.0



Interaction Analysis Considering Moment Curvature

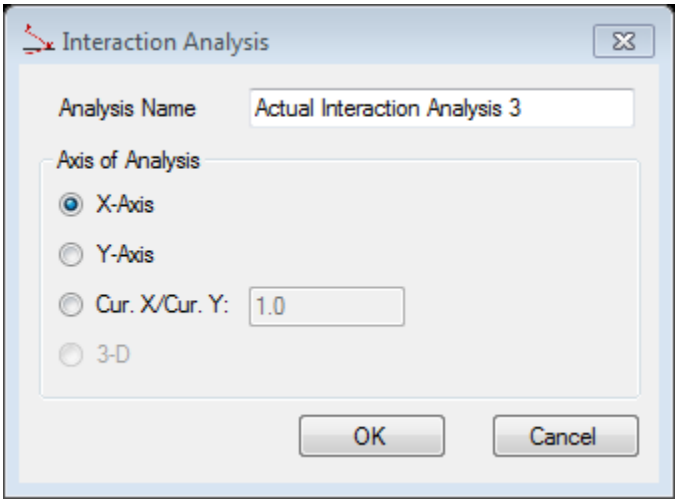
Another type of interaction curve is referred to as axial force-bending moment interaction failure curve in literature. The KSU-RC analysis uses a constant neutral axis orientation. This interaction method assumes an axial force for each step, and then a monotonic moment-curvature

analysis along the defined axis is conducted. The maximum moment of the moment-curvature analysis is picked as the corresponding moment to the axial force level.

User Input

The user should specify the constant strain at the extreme concrete fiber for the analysis, as well as the axis about which the analysis will be performed. Note that future versions of KSU-RC will have the capability of producing a 3D interaction failure surface. An optional user input is the Analysis Name. Analysis names can be used to distinguish between analyses based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-Int-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. KSU-RC 2.0 automatically populates the Analysis Name text box with a name for each analysis that is performed.

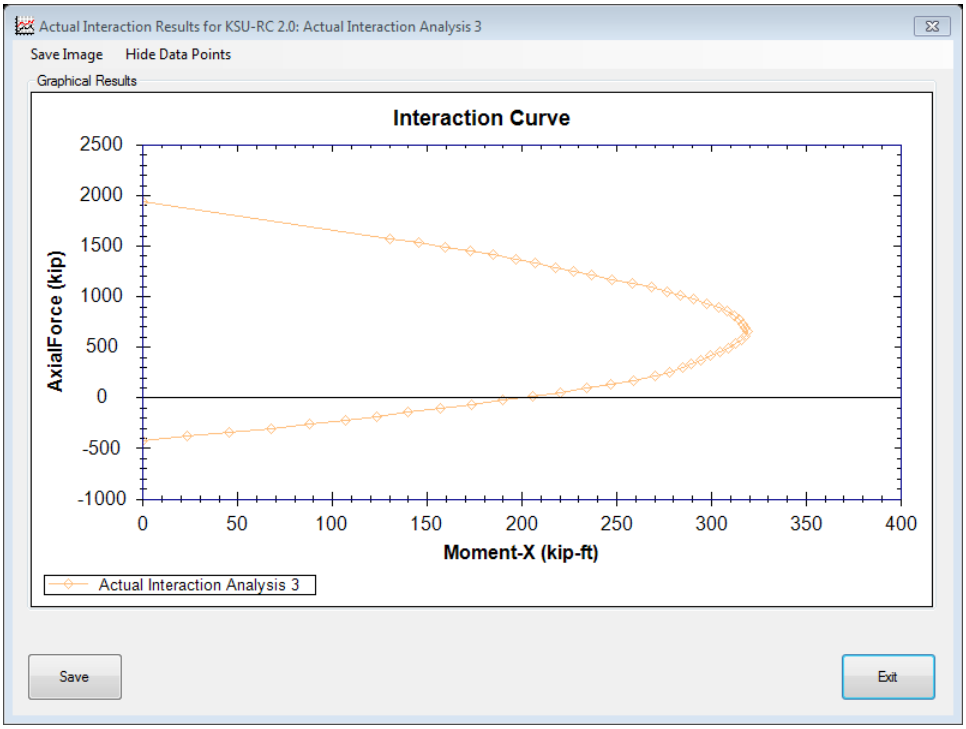
Figure 52. Interaction User Input Window



Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the moment in each analysis direction, axial force, depth to neutral axis (c), and curvature in each analysis direction for each point calculated in the analysis.

Figure 53. Interaction Curve Considering Moment-Curvature Produced by KSU-RC 2.0



Interaction Analysis Considering Load Pattern

In various algorithms developed by researchers to calculate failure surface, ultimate moment capacity of an RC section is defined as the maximum moment in monotonic moment-

curvature analysis considering constant axial force (not considering the loading pattern of axial force). However, columns exposed to a dynamic excitation are subjected to a loading pattern in combined but non-proportional lateral and axial directions. Consideration of the axial force loading pattern when calculating bending capacity of an RC section is essential in structural columns located in near-fault regions.

KSU-RC 2.0 is capable of generating the failure surface of a composite section using proper material models while considering the axial force loading pattern. To calculate the failure surface of a section while considering the axial force loading pattern for a specific neutral axis orientation and axial force level (P), a moment-curvature analysis is performed considering the pattern of axial force against curvatures along x- and y-directions. The maximum moment was selected as the ultimate capacity of the section in that level of axial force.

User Input

In this type of analysis, the loading path should be entered as a text file. To avoid any errors while reading data from the text file, the following format should be followed. The loading path should include three columns of data. The first column is an axial force coefficient which is between -0.1 and 1. The second column is curvature along the x-axis and the third column is curvature along the y-axis. The axial force coefficient (ac) for any combination of x- and y-curvature cannot exceed 1.0, meaning that the maximum compressive axial force (ac*P) in that pattern is equal to the specified level of axial force (P). The axial force coefficient must be greater than -0.1, meaning that the maximum tensile axial force capacity of the section should not be considered more than 0.1*P. It is recommended that the columns are created in Excel and

then copy-pasted into a text file. Empty cells should not be copied from Excel. Below is an example of text file arrangement:

ac_1 $\phi_{x,1}$ $\phi_{y,1}$

ac_2 $\phi_{x,2}$ $\phi_{y,2}$

ac_3 $\phi_{x,3}$ $\phi_{y,3}$

⋮

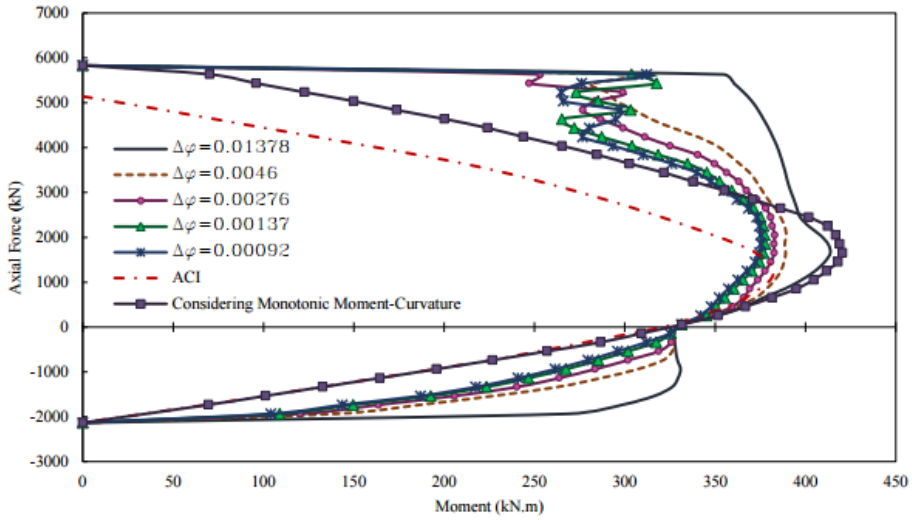
ac_n $\phi_{x,n}$ $\phi_{y,n}$

Analysis names can be used to distinguish between analyses based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-Int-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. The Analysis Name for this interaction method is taken from the name of the text input file.

Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the moment in each analysis direction, axial force, depth to neutral axis (c), and curvature in each analysis direction for each point calculated in the analysis.

Figure 54. Interaction Curve Considering Axial Load Pattern



Moment Curvature Analysis Methods

Two types of Moment-Curvature Analysis Methods are available in KSU RC 2.0. These methods include ACI code-based moment curvature and moment curvature based on material models.

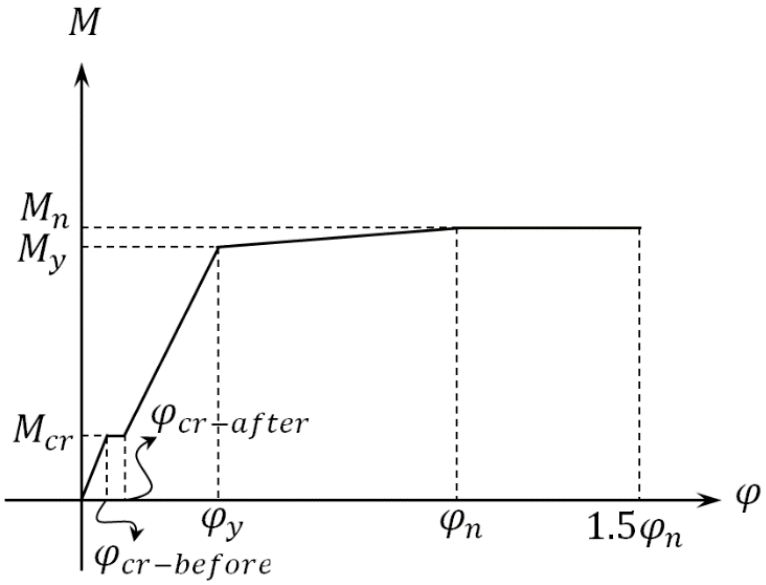
ACI Code-Based Moment-Curvature Analysis

In code-based moment-curvature analysis, the "ACI stress-block" (Hognestad stress block) is used without considering the confinement effect of lateral reinforcement. The approximate moment-curvature analysis can be conducted considering one of two loading conditions: force-controlled and displacement controlled.

The approximate code-based moment-curvature graphs contain 5 points. The first point is related to the starting point of which curvature and moment are equal to zero. The second point is related to the "First Crack." For displacement control analysis, when the curvature changes step-

by-step and moment is calculated, the bending moment drops when a crack develops. Therefore, at the same curvature (ϕ_{cr}), two moments are present: the moment before ($M_{cr,before}$) and the moment right after the crack ($M_{cr,after}$). For force-controlled analysis when curvature related to a given moment is calculated at the first crack point, the curvature jumps to a new curvature immediately following the first crack, thereby demonstrating the presence of two curvatures: one before the first crack and one after the first crack. The third point of the code-based moment-curvature graph is related to "Steel Yield," at which point the steel bars initially yield. The fourth point is related to the nominal moment strength of the RC section. The last point is related to "Ultimate Strength," which considers the code-based ultimate strength of an RC section.

Figure 55. ACI Approximate Moment-Curvature Nomenclature

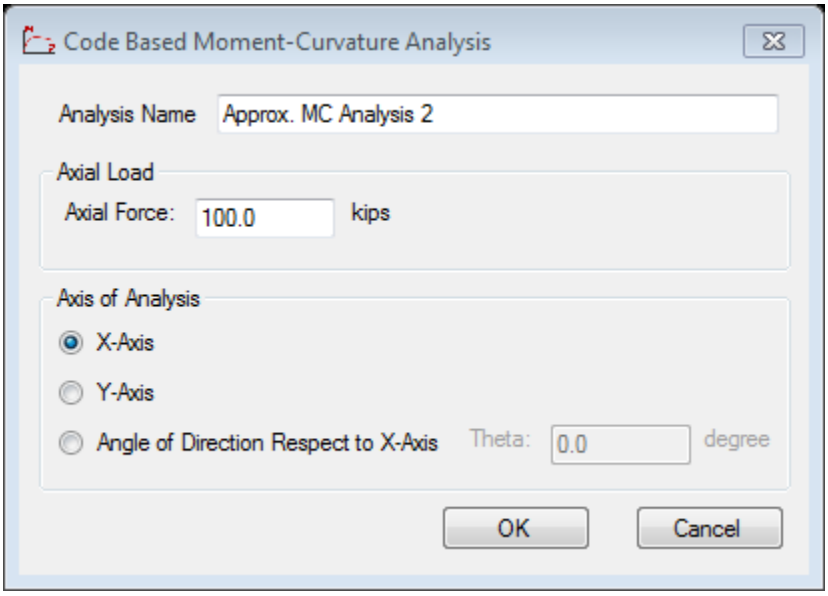


User Input

The user should specify the axis about which the analysis will be performed. An optional user input is the Analysis Name. Analysis names can be used to distinguish between analyses

based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-MC-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. KSU-RC 2.0 automatically populates the Analysis Name text box with a name for each analysis that is performed.

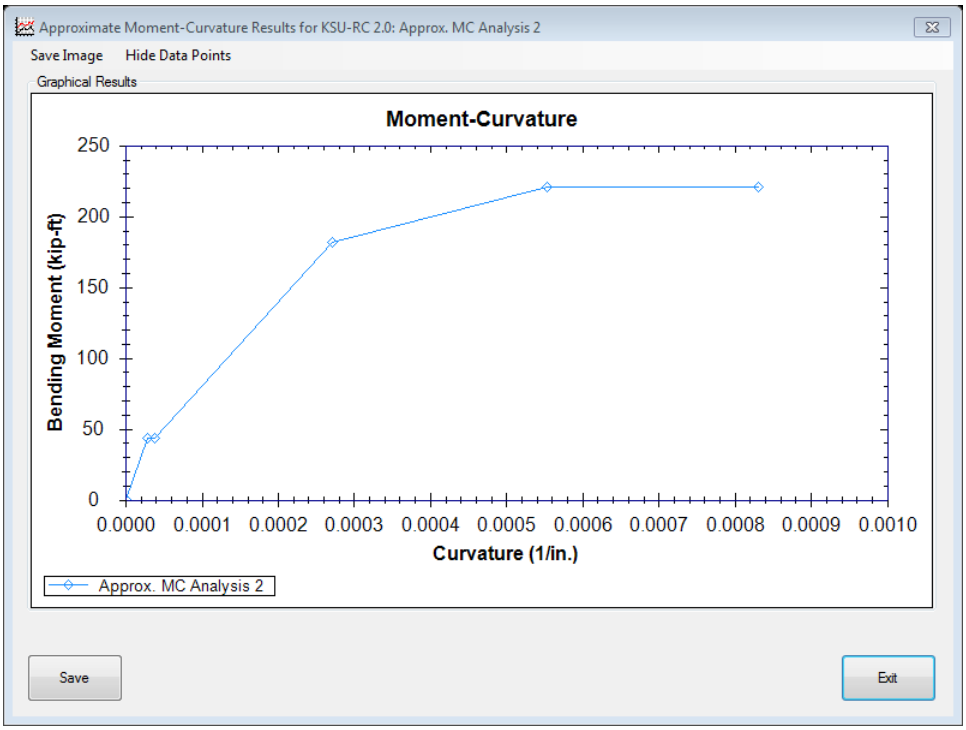
Figure 56. ACI Moment-Curvature User Input Window



Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the moment, axial force, strain at the centroid of the section (d), and curvature in each analysis direction for each point calculated in the analysis.

Figure 57. ACI Moment-Curvature Curve Produced by KSU-RC 2.0



Moment Curvature Based on Material Models

The actual moment curvature analysis using material models uses a known curvature path along the x and y axis. The ultimate bending moments along both axes are calculated using material models. Having the curvature in both directions in each iterative step, the value of D is calculated to set the axial force equal to the applied axial force at the top of the beam-column section. Next, bending moments along both axes are calculated. Unlike code-based moment-curvature analysis, KSU-RC can perform monotonic and cyclic moment-curvature analysis using material models. Monotonic moment-curvature analysis can be performed by using a fixed axial force in the analysis. A cyclic moment-curvature analysis can be performed by importing a text file containing a list of axial forces and curvatures in the window shown in Figure 58.

User Input

The user should specify the constant axial force at the extreme concrete fiber for the analysis, as well as the axis about which the analysis will be performed. An optional user input is the Analysis Name. Analysis names can be used to distinguish between analyses based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-MC-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. KSU-RC 2.0 automatically populates the Analysis Name text box with a name for each analysis that is performed.

By clicking the 'Import File' button, an Open File Dialog will prompt the user to select a text file containing the necessary data for the moment curvature analysis. It is recommended that the columns are created in Excel and then copy-pasted into a text file. Empty cells should not be copied from Excel. Below is an example of the proper text file arrangement:

P_1 $\phi_{x,1}$ $\phi_{y,1}$

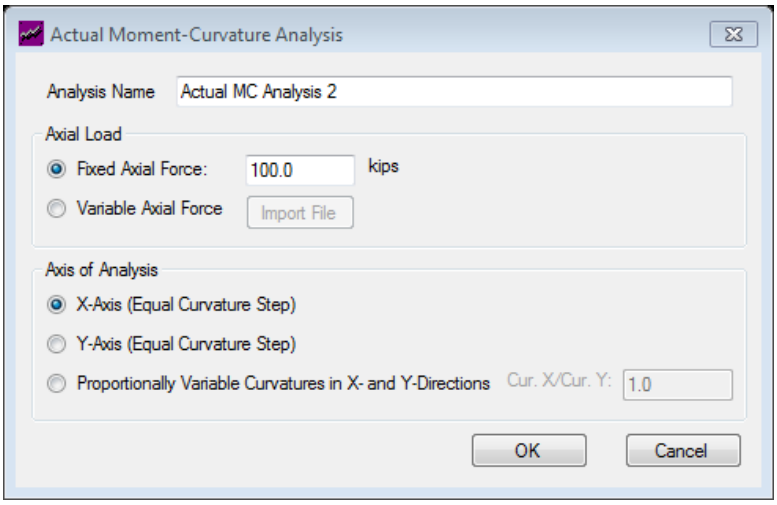
P_2 $\phi_{x,2}$ $\phi_{y,2}$

P_3 $\phi_{x,3}$ $\phi_{y,3}$

⋮

P_n $\phi_{x,n}$ $\phi_{y,n}$

Figure 58. Moment-Curvature User Input Window



Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the moment in each axis direction, axial force, strain at the centroid of the section (d), and curvature in each axis direction for each point calculated in the analysis.

Figure 59. Monotonic Moment-Curvature Results Produced by KSU-RC 2.0

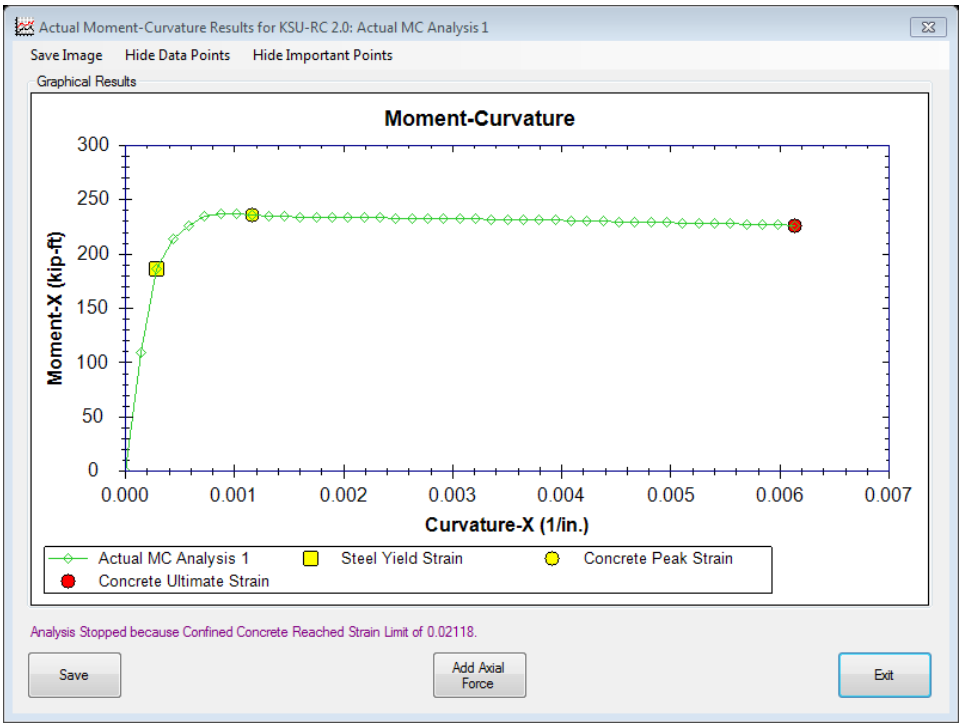


Figure 60. Variable Axial Load Moment-Curvature Results Produced by KSU-RC 2.0

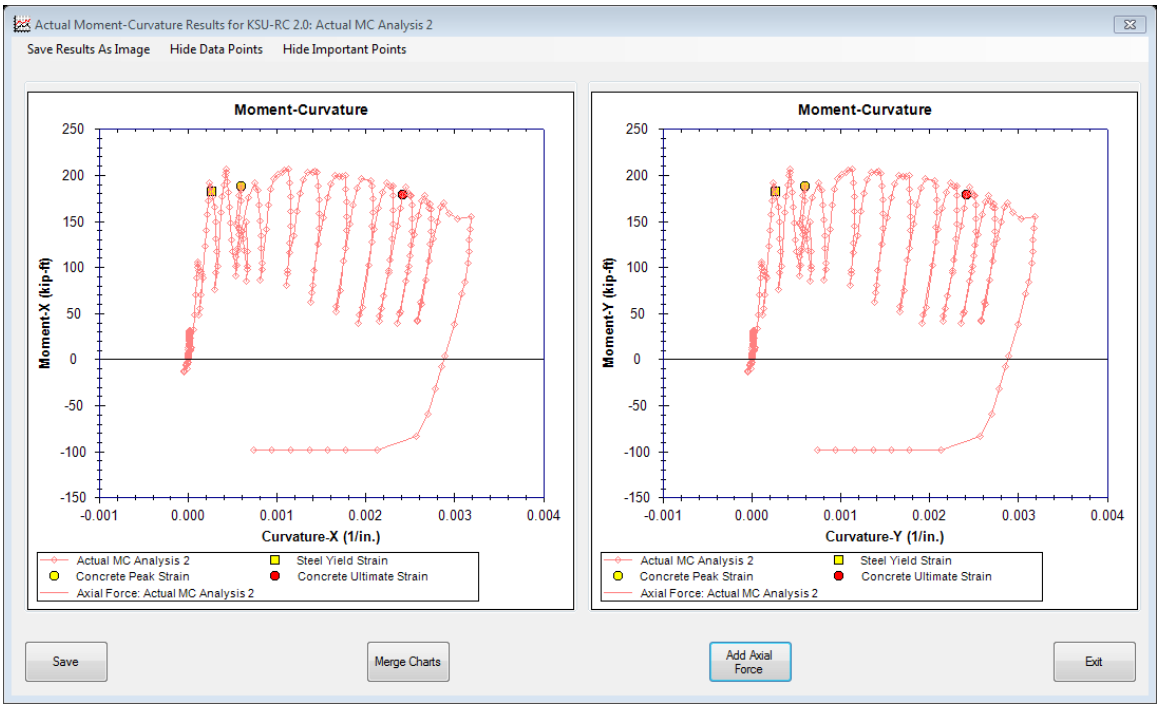
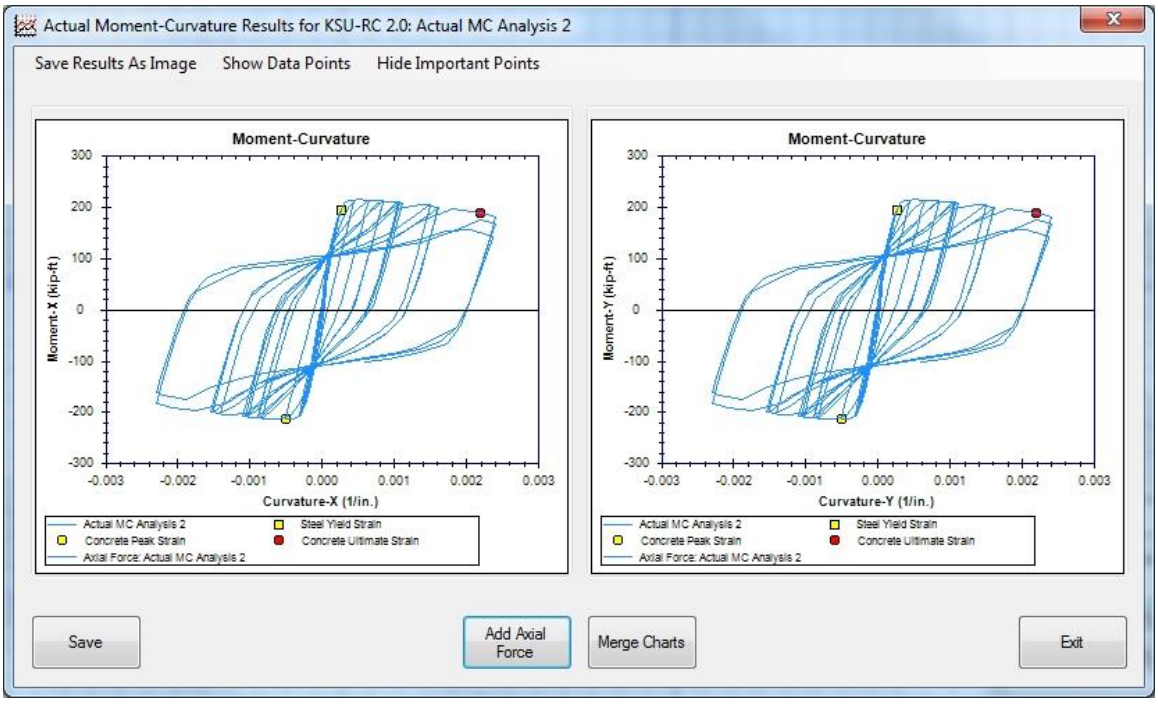


Figure 61. Cyclic Moment-Curvature Results Produced by KSU-RC 2.0



Force-Deflection Analysis Methods

The plastic hinge concept of the critical section (or transition area) and a proper curvature distribution along the beam-column specimen height are employed to solve the force-deflection problem when curvature falls into the descending branch of the moment-curvature curve. In the plastic hinge approach, column height is divided into two elastic and plastic elements. Depending on the curvature distribution method, the length of the transition area (plastic hinge length) may or may not change during force-deflection analysis. Total displacement at the top of a column is a summation of elastic and plastic deformation caused by elastic and plastic elements, respectively.

The plastic hinge method is used by KSU-RC to perform force-deflection analysis of an RC beam-column under constant or variable axial force and cyclic or monotonic lateral forces/displacements. Nearly 30 plastic hinge models are available in literature. Priestley and Park's model (Priestley & Park, 1987), Esmaeily's models (Esmaeily, 2002) were implemented into KSU-RC. Although a majority of existing plastic hinge models in the literature were developed considering uniaxial behavior of RC columns, experimental evidence has confirmed plastic hinge length is not affected by biaxial loading (Rodrigues, et al., 2013); therefore, the plastic hinge models developed based on uniaxial testing are applicable for biaxial force-deflection analysis.

KSU-RC 2.0 can conduct biaxial force-deflection analysis. Axial force and lateral forces and displacements are applied at the top of a column. For each combination of axial force and lateral displacement in the x and y direction, two sets of trail-error inputs for the strain at the centroid of the section and the curvature in both directions, axial force and lateral displacements

are set to the applied values. Force-deflection analysis of an RC section under constant or variable axial force and cyclic or monotonic lateral displacements can be conducted using aforesaid plastic hinge models implemented in the developed computer application. A simplified flowchart for force-deflection analysis of an RC column under axial force and biaxial lateral displacements is shown in Figure 63.

Priestley and Park

Priestley and Park's plastic hinge length depends on column height, longitudinal steel yield strength, and rebar diameter. Curvature distribution was considered to be uniform along the plastic hinge length, and axial force effect was not taken into account in Priestley and Park's model (Priestley & Park, 1987).

Priestley and Park (modified by Esmaeily)

(Esmaeily, 2002) later modified Priestly and Park's plastic hinge length by employing maximum tensile stress at longitudinal bars instead of steel yield strength. These plastic hinge models work well for RC columns under constant axial force and monotonic lateral force/displacement.

Esmaeily's First Method

Esmaeily's first plastic hinge method considered a linear distribution along transition zone or plastic hinge length. Plastic hinge length applied in this model depends on yield moment and moment at critical section at each step of loading. Therefore, axial force effect and cyclic

and monotonic behavior of all materials are taken into account in calculation of plastic hinge length. Esmaeily's first plastic hinge length may increase by decreasing the ratio of yield moment to moment of critical section. Plastic hinge length simulates severe damage at column footing; therefore, it cannot be decreased after formation (Esmaeily, 2002).

Esmaeily's Second Model

Esmaeily's second plastic hinge model divides the transition zone into two parts. Curvature distribution along the part closet to the critical section is assumed to be uniform. The length of this part is constant and is equal to the section depth for columns with high to depth ratio of less than 12.5; otherwise, it is equal to $0.08\times L$, where L is the column height. Curvature distribution along the second part is assumed to be linear; its length depends on existing axial force and level of lateral force. Esmaeily's first and second models work more accurately compared to Priestley and Park and Priestley and Park-Revised by Xiao models for RC columns under variable axial force and cyclic lateral force/displacement (Esmaeily, 2002). Curvature distribution along column height as assumed in aforesaid models is shown in Figure 62.

Figure 62. Curvature Distributing along Column Height as Assumed in (a) Priestley and Park and Priestley and Park revised by Esmaeily method, (b) Esmaeily's First method, and (c) Esmaeily's Second Method

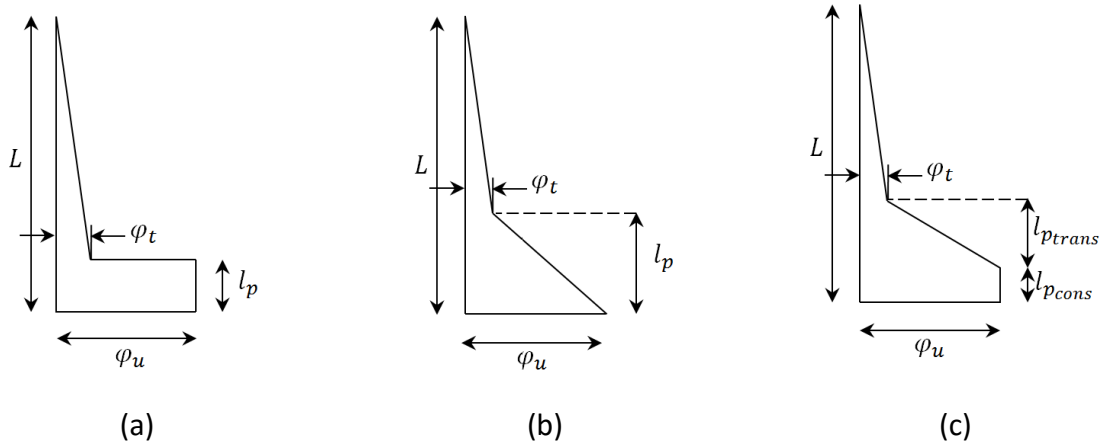
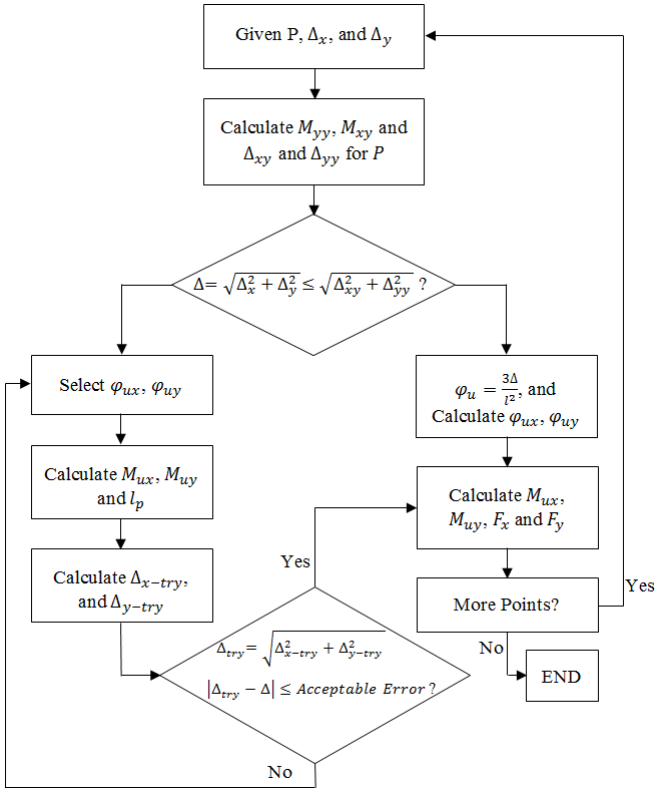


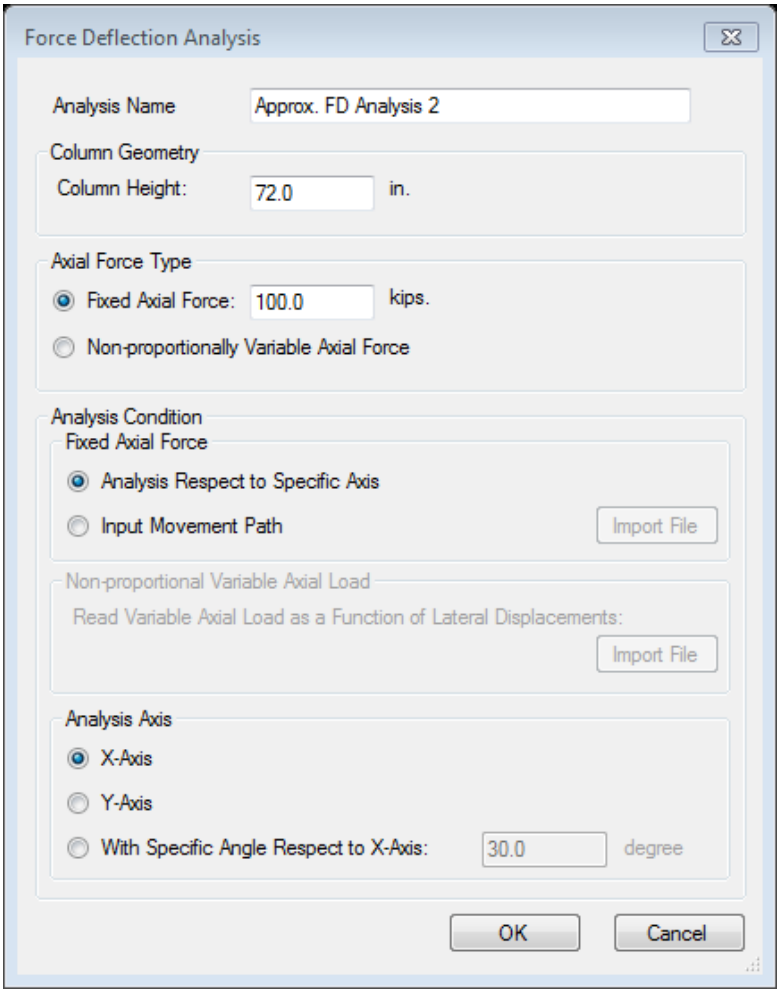
Figure 63. Force-Deflection Analysis Flowchart



User Input

The user input varies based on the type of force-deflection analysis the user would like to perform. The force-deflection types are fixed axial force and variable axial force. Figure 64 shows the window where necessary user input can be entered. The necessary information to run a force-deflection analysis is described in detail in the following subtopics.

Figure 64. Force-Deflection User Input Window



Fixed Axial Force

Two options for analysis are available within the fixed axial force analysis:

- Analysis with Respect to a Specific Axis

In this type of analysis, a monotonic lateral force is applied to the top of the beam-column about a specific axis, with a fixed axial force. The user should input the column height, fixed axial force magnitude, and analysis axis.

- Analysis Follows Movement Path

A cyclic displacement path is followed in this type of analysis with a fixed axial force. The user should input the column height, fixed axial force magnitude, and a text file movement path. It is recommended that the columns in the text file are created in Excel and then copy-pasted into a text file. Empty cells should not be copied from Excel. Below is an example of text file arrangement, where δ is the displacement in either inches or millimeters based on the unit system:

$\delta_{x,1}$ $\delta_{y,1}$

$\delta_{x,2}$ $\delta_{y,2}$

$\delta_{x,3}$ $\delta_{y,3}$

⋮

$\delta_{x,n}$ $\delta_{y,n}$

Non-Proportionally Variable Axial Force

This type of analysis process follows a movement path with a variable axial force. In this type of analysis, the loading path should be entered as a text file. It is recommended that the columns are created in Excel and then copy-pasted into a text file. Empty cells should not be copied from Excel. Below is an example of the proper text file arrangement is shown below, where P is the axial force (in kips or kN, depending on the unit system) and δ is the displacement (in inches or mm):

P_1 $\delta_{x,1}$ $\delta_{y,1}$

P_2 $\delta_{x,2}$ $\delta_{y,2}$

P_3 $\delta_{x,3}$ $\delta_{y,3}$

⋮

P_n $\delta_{x,n}$ $\delta_{y,n}$

An optional user input is the Analysis Name. Analysis names can be used to distinguish between analyses based on specific properties that a user is changing in each analysis. For example, if the user is changing steel yield strength and concrete compressive strength properties between each consecutive analysis, an appropriate name would be ACI-Int-fc5-fy60. This name tells the user what analysis method is used as well as the concrete compressive strength and steel yield strength used in the analysis. KSU-RC 2.0 automatically populates the Analysis Name text

box with a name for each analysis that is performed, but this name can be changed by the user as mentioned above.

Results

The graphical results for an analysis are plotted in real time during an analysis. From this window, the results can be saved in image format or in text file results. The text file results contain the deflection in each analysis direction, axial force, and lateral force in each analysis direction for each point calculated in the analysis.

Figure 65. Monotonic Force-Deflection Results Produced by KSU-RC 2.0

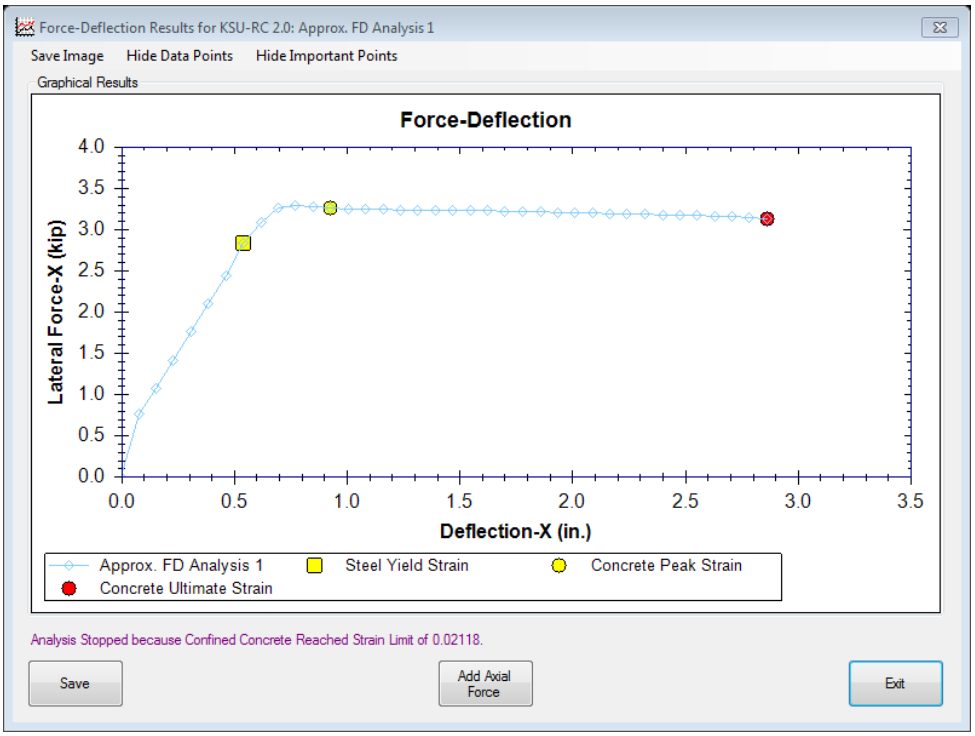
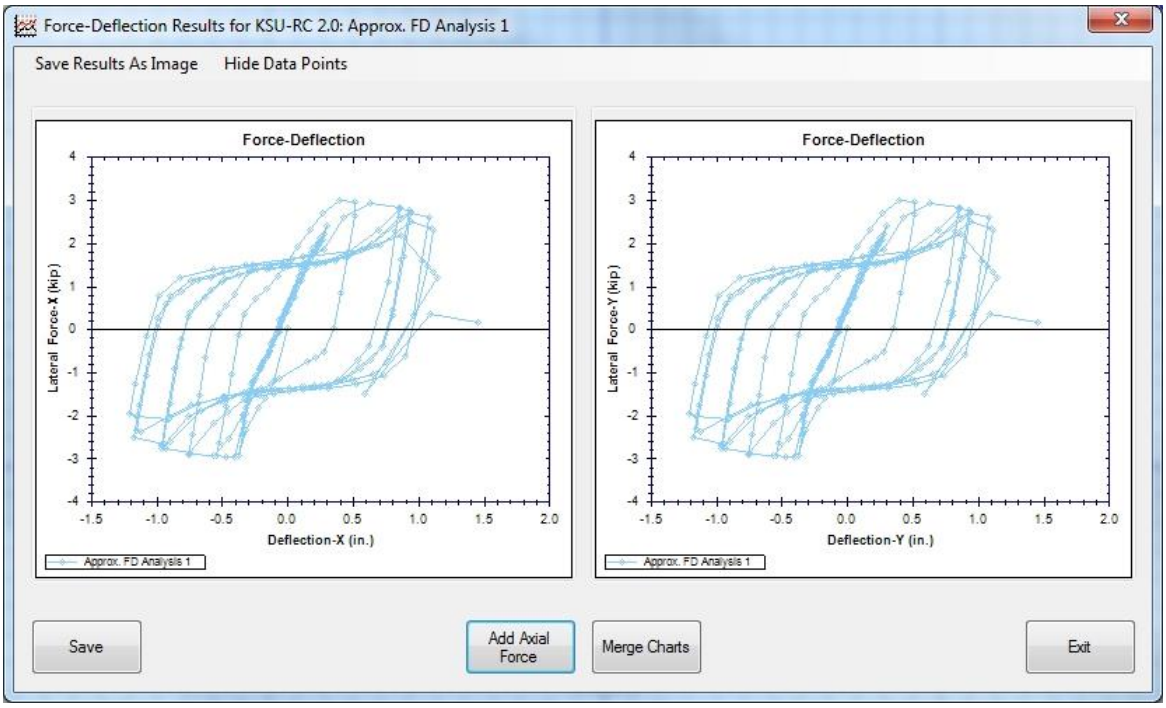


Figure 66. Cyclic Force-Deflection Results Produced by KSU-RC 2.0



Viewing and Manipulating Result Graphs

All of the analytical results are shown on one graph during the analysis process in a 2-D graph. When an analysis is performed in two directions, “x” and “y”, two graphs will be shown, one per each direction. Dual-Axis graphs can be merged to show the results pertaining to each axis on the same graph. Additionally, each analysis result graph can be shown in its own window or combined with other results from the same type of analysis using the Result List feature. Right-clicking the graph provides various options to view the data. The graph can be zoomed to a specific area, or the user can pan the graph by holding the middle (wheel) button and moving the mouse. A graph can be saved as a picture, or the related data can be saved in a text file. These options are in addition to the results being saved when the whole project is saved in a .krc2 file.

Figure 67. Single Axis Analysis Results Window

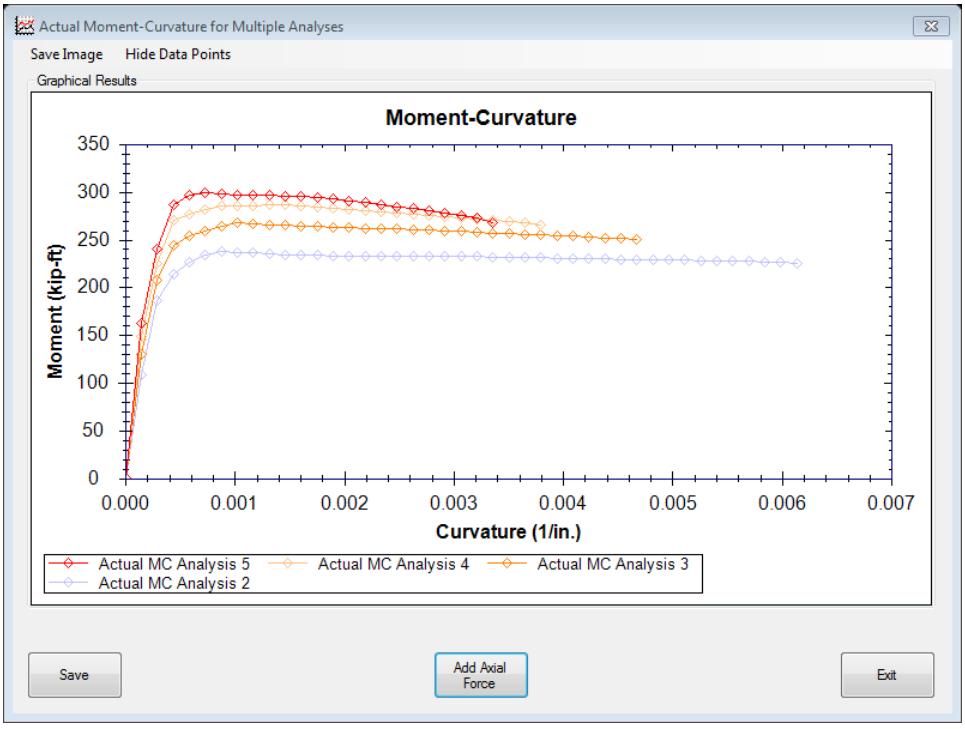
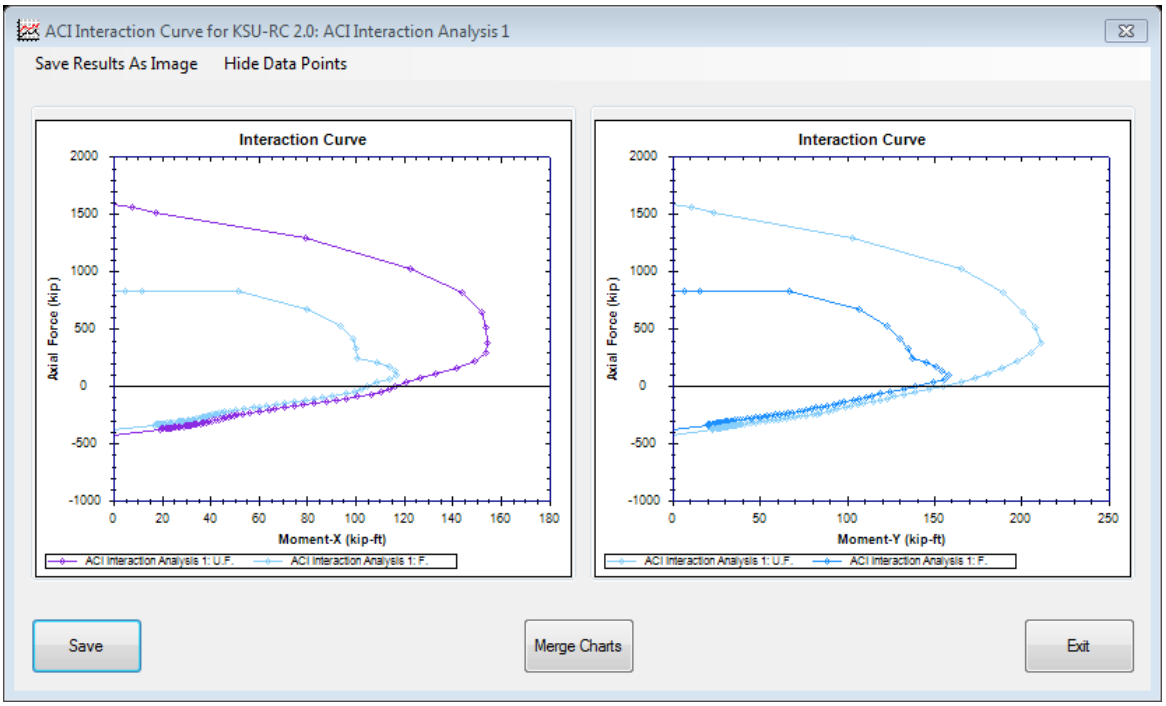


Figure 68. Dual Axis Analysis Results Window



Chapter 5 - Conclusion and Recommendations

This paper is a detailed review of the research of the performance of concrete beam-columns under any type of lateral bi-axial curvature or displacement and variable axial load. Research started by Esmaeily in the late 1990's has carried through to the research that is part of this thesis. Through the last couple of decades, analytic models and methods have been created and calibrated with experimental data to form a group of analytic procedures that accurately predict the performance of reinforced concrete columns under variable axial loads and lateral displacements. Three generations of a fiber based computer program have been used to analyze the performance of reinforced concrete beams and columns and assess the accuracy of the underlying analytical models. The latest generation of the fiber-based program, KSU RC 2.0, specifically the algorithms used in the program and the user interface are refined as part of this thesis. Graphic features were added to the program to allow the user to easily manipulate the data created in analyses. The current version of the program has many features compared to those in the original version of the program. Additionally, it is recommended to add the following features to the program in the future to make the program even more powerful:

- Upgrade the CharAsad (the ActiveX control, DLL program) to allow the user to input any arbitrary shape as a section, and mesh a section with arbitrary geometry and reinforcement arrangement.

- Implement additional material models into the program to allow other shapes, including sections with an arbitrary geometry and reinforcement, to be confined with FRP wraps.

- Add 3-Dimensional graphing capabilities to the program to create a spacial representation of the analytical results, especially for axial force-bending moment interaction of a section under specific load patterns; this will be different from conventional curves in various lateral directions.

Chapter 6 - Bibliography

Campione, G. & Fossetti, M., 2007. Compressive Behaviour of Concrete Elliptical Columns Confined by Single Hoops. *Engineering Structures,* 29(3), pp. 408-417.

Cusson, D. & Paultre, P., 1995. Stress-Strain Model for Confined High-Strength Concrete. *Journal of Structural Engineering,* 121(3), pp. 468-477.

Esmaeily, A. & Lucio, K., 2006. Analytical Performance of Reinforced Concrete Columns using Various Confinement Models. *ACI SP-238-6,* pp. 95-110.

Esmaeily, A. & Peterman, R., 2007. Performance Analysis Tool for Reinforced Concrete Members. *Computers and Concrete,* 4(5), pp. 331-346.

Esmaeily, A. & Shirmohammadi, F., 2014. *Performance and Capacity Assessment of Reinforced Concrete Bridge Piers Considering the Current Load and Resistance Factor Design Provisions and Plastic Hinge Length in Kansas,* Topeka, KS: Kansas Department of Transportation.

Esmaeily, A. & Xiao, Y., 2002. Seismic Behavior of Bridge Columns Subjected to Various Loading Patterns. *PEER Journal,* 2002(15), p. 321.

Esmaeily, A. & Xiao, Y., 2004. Behavior of Reinforced Concrete Columns Under Variable Axial Loads. *ACI Structural Journal,* 101(1), pp. 124-132.

Esmaeily, A. & Xiao, Y., 2005. Behavior of Reinforced Concrete Columns under Variable Axial Loads: Analysis. *ACI Structural Journal,* 102(5), pp. 736-744.

Fafitis, A. & Shah, S., 1985. Predictions of Ultimate Behavior of Confined Columns Subjected to Large Deformations. *ACI Journal,* 82(4), pp. 423-433.

Kawashima, K., Hostani, M. & Yoneda, K., 2000. Carbon Fiber Sheet Retrofit of Reinforced Concrete Bridge Piers. *Proceedings of the International Workshop on Annual Commemoration of Chi-Chi Earthquake,* Volume 2, pp. 124-135.

Lee, J. et al., 2009. Compressive Response of Concrete Confined with Steel Spirals and FRP Composites. *Journal of Composite Materials,* 44(4), pp. 1-24.

Mander, J., Panthaki, F. & Kasalanati, A., 1994. Low-Cycle Fatigue Behavior of Reinforcing Steel. *Journal of Materials in Civil Engineering,* 6(4), pp. 453-468.

Mander, J., Priestley, M. & Park, R., 1988. Theoretical Stress-Strain Model for Confined Concrete. *Journal of Structural Engineering,* 114(8), pp. 1804-1826.

Menegotto, M. & Pinto, P., 1973. Method of Analysis for Cyclically Loaded Reinforced Concrete Plane Frames Including Changes in Geometry and Non-Elastic Behavior of Elements under Combined Normal Force and Bending. *Proceedings of IABSE Syposium*

of Resistance and Ultimate Deformability of Structures acted on by Well-Defined Repeated Loads, pp. 12-15.

Popovics, S., 1973. A Numerical Approach to the Complete Stress-Strain Curve of Concrete. *Cement and Concrete Research,* 3(5), pp. 583-599.

Priestley, M. & Park, R., 1987. Strength and Ductility of Concrete Bridge Columns under Seismic Loading. *ACI Structural Journal,* 84(1).

Rodrigues, H., Arede, A., Varum, H. & Costa, A., 2013. Experimental Evaluation of Rectangual Concrete Column Behavior under Biaxial Cyclic Loading. *Journal of Earthquake Engineering and Structural Dynamics,* 42(2), pp. 239-259.

Samaan, M., Mirmiran, A. & Shahawy, M., 1998. Model of Concrete Confined by Fiber Composites. *Journal of Structural Engineering,* 124(9), pp. 1025-1031.

Shirmohammadi, F., 2015. *Effect of Load Pattern and History on Performance of Reinforced Concrete Columns,* Manhattan, KS: s.n.

Shirmohammadi, F. & Esmaeily, A., 2015. Performance of Reinforced Concrete Columns under Bi-Axial Lateral Force/Displacement and Axial Load. *Engineering Structures,* Volume 99, pp. 63-77.

Shirmohammadi, F. & Esmaeily, A., 2015. Stress-Strain Model for Circular Concrete Column Confined by FRP and Conventional Lateral Steel. *Engineering Structures,* Volume 84, pp. 395-405.

Shirmohammadi, F. & Esmaeily, A., 2016. Software for Biaxial Cyclic Analysis of Reinforced Concrete Columns. *Computers and Concrete,* 17(3), pp. 353-386.

Youssef, M., Feng, M. & Mosallam, A., 2007. Stress-Strain Model for Concrete Confined by FRP Composites. *Composites Part B: Engineering,* 38(5), pp. 614-628.