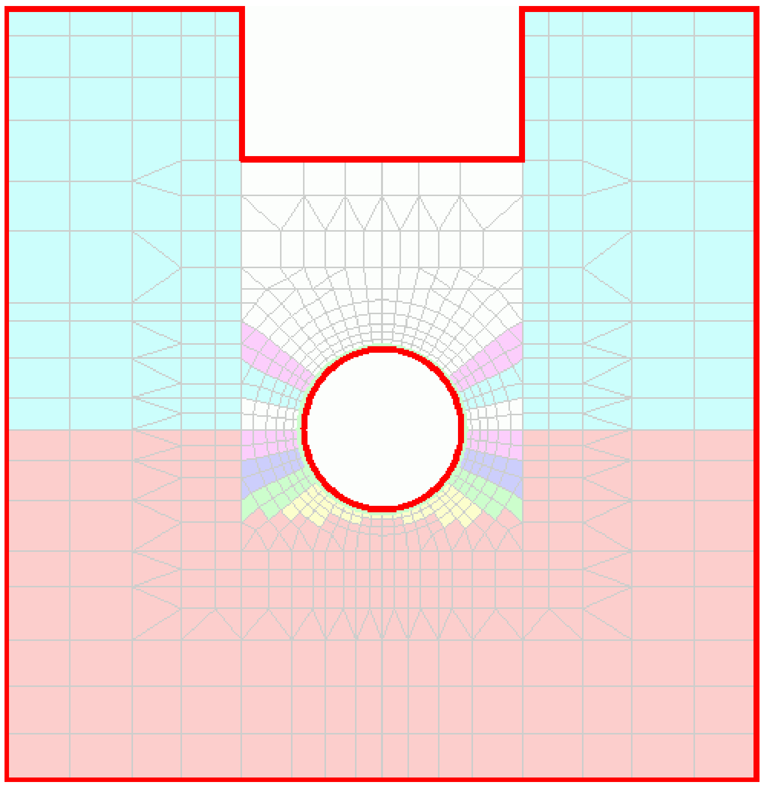
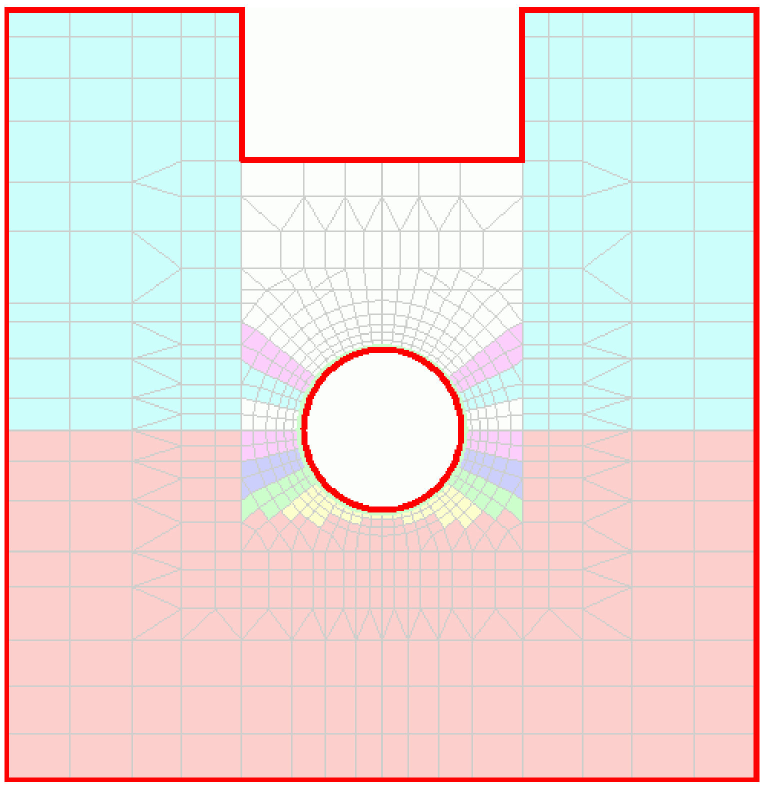
CANDE-2007
Culvert Analysis and Design
Tutorial of Applications



Developed under National Cooperative Highway Research Project NCHRP 15-28

CANDE-2007
Culvert Analysis and Design
Tutorial of Applications



Developed under National Cooperative Highway Research Project NCHRP 15-28

Timothy J. McGrath – Simpson, Gumpertz & Heger Inc.
Arlington, MA

Mark Mlynarski – Michael Baker Jr. Inc.
Moon Township, PA

Michael G. Katona – Consultant
Gig Harbor, WA

(This page intentionally left blank)

INTRODUCTION

This tutorial presents 16 example problems to assist users in learning the features and operations of CANDE-2007. The introductory section discusses the overall approach that should be taken to solving design and analysis problems in CANDE, each problem is then presented in terms of input and output.

The tutorial is developed to demonstrate the process of creating and running files with the new graphical user interface (GUI) and the CANDE-2007 User Manual and Guideline, a companion manual for this tutorial. The User Manual presents all there is to know about setting up input files in CANDE, thus, this tutorial will reference the appropriate section of the User Manual and will not repeat.

Getting Started

Documentation for CANDE-2007 is contained in three stand-alone documents referred to as the theoretical manual, user manual, and the tutorial. The user manual and tutorial follow the same sequence through five Sections of input required to set up a CANDE run. These are:

A. Master control – The input under master control defines the overall approach to the problem. Choices include analysis vs. design, LRFD design vs. service load design, solution level and number of structures to be analyzed. The new CANDE program allows analyses of multiple pipes of multiple materials in a single run.

B. Pipe Definition – In this section the user identifies each pipe material and all associated material properties needed to complete the input file.

C. Solution Level – While the solution level is identified in Section A, the details of the solution are provided in Section C. The benefits of the three solution levels are discussed in the User Manual. They include:

Level 1 is a simplified analysis based on a closed form elastic solution for a conduit embedded in an elastic medium. Level 1 is limited to deeply buried, circular pipes in a homogenous soil without stiff beddings or soft haunch zones. Two limiting soil-structure interface conditions are available, bonded or frictionless.

Level 2 is a finite element analysis method utilizing predefined meshes, soil zones, and construction sequences for symmetrical culvert installations. Automated culvert shapes include pipes, ellipses, boxes and arches installed in embankment or trench installations. Level 2 is considered the “work horse” and is applicable to the large majority of culvert problems. The so-called Level 2-extended option allows users to modify the predetermined input for placement of live loads and inclusion of soft haunch zones.

Level 3 is a finite element analysis method with no restrictions on the shape, symmetry, or number of structures within the two-dimensional framework. Level 3 requires the user to define the finite-element mesh topology either by generating the mesh data with direct input or importing the mesh data from another source.

D. Material Definition – Material definition allows the user to specify the soil model and properties to be used in the analysis and all necessary parameters.

E. LRFD Design – The section “LRFD Design” is new to CANDE. In this section the user provides composite factors for each load increment to allow analysis for factored loads.

In general, most questions are answered in the User Manual. Even relatively new users should be able to proceed in order through the five sections of the User Manual and step by step build an input file. The tutorial problems provide examples of using the GUI for establishing the input file as opposed to the so-called batch mode input method of directly typing an input file from the formatted input instructions in the detailed User Manual.

The general sequence for analyzing a problem is:

1. Define problem – Defining a problem is perhaps the most important part of analysis. Identifying known parameters and making rational judgments about unknown parameters is the key to constructing accurate models. While the structure itself is often well defined, it is very common to lack sufficient information on the in situ soils, backfills, and density levels. These problems can result from inconsistent or unknown native soils, construction specifications that permit an excessively wide range of backfill materials, uncertainty about the enforcement of compaction specifications, or a number of other sources.

2. Select analysis level – After defining the problem, a designer can select an appropriate level for the analysis. The level of sophistication will often be dictated by the level of knowledge of installation conditions and the expected quality control during installation. In situations where little knowledge exists, then designers should probably use simplified analyses with conservative inputs. Because of lack of information many buried pipe and culvert installations are best addressed by simplified procedures such as those in the AASHTO LRFD Bridge Design Specifications. A sophisticated analysis of uncertain conditions is rarely productive; however, as the level of knowledge increases or when designs encompass special conditions, tremendous knowledge can be gained through the use of high level analyses, such as those offered in CANDE. Decisions regarding the appropriate analysis level to be used within CANDE should follow a similar approach. Use the simplest analysis that can address the issues at hand. Generally use Level 2 when it is applicable to the problem at hand and use default values for soil and culvert properties as they represent conservative conditions. If unknown conditions exist, such as the soil-structure interface condition, it is

recommended to solve the problem two or more times using alternative assumptions to see how changing the unknown variables affects the culvert performance.

3. Execute problem – Once the solution is obtained the designer needs to review the output and assess whether or not the intended input was correctly interpreted by the program, or if it addresses all the potential conditions being investigated. It is a very common practice in buried pipe design to analyze several sets of conditions to test a design for sensitivity to known variables, such as backfill type, compaction levels or haunch support. It is important to note that various CANDE input parameters can have a significant impact on the response of the structure. Care should be taken in researching, selecting, and testing input parameters.

4. Document findings, assess performance, improve procedures for subsequent designs – The greatest educator is often experience, and culvert designers, like all engineers should take advantage of knowledge gained from one design by applying it to future designs. This might include streamlining procedures for conducting an analysis to inspecting a pipe after installation and evaluating how well the design model captured the actual conditions.

The CANDE GUI allows input through interactive screens and through a text editor. The interactive screens identify individual variables, with separate fields for entering each parameter. As screens are updated, the interface writes the input file that CANDE will use for the analysis. This will likely be the preferred method for most people to use the program. The text editor allows line by line input in much the same manner as CANDE files have been constructed in previous versions using the batch-mode input method. The benefit of the new text editor is that the individual fields are identified as a user moves across an input line. The text edited file is constructed in fixed field format.

Tutorial Problems

The sixteen example problems presented in the following sections demonstrate the majority of features encompassed by CANDE-2007. The examples are described in Table 1-1 which lists the solution level, material type, structure type, installation type, execution mode, method of analysis/design, and details any special design or interface features exercised in the problem. Table 1-2 which describes the details of each of the problems. For these test problems assumptions are made for many test parameters. In actual design or analysis situations accurate decision about input parameters is paramount to the success of a computer modeling effort.

Table 1-1 – Tutorial Examples

Example No.	Solution Level	Material Type	Structure Type	Installation Type	Execution Mode	Method of Analysis / Design	Design Features	CANDE Interface Features
1	1	Steel	Pipe	N/A	Design	LRFD	N/A	N/A
2	1	R/C	Pipe	N/A	Design	LRFD	N/A	N/A
3	1	Plastic	Pipe	N/A	Design	LRFD	Uses long-term load duration	N/A
4	2	Alum.	Pipe	Trench	Design	Service	N/A	N/A
5	2	Plastic	Pipe	Embankment	Analysis	Service	Uses large deformation / buckling analysis	N/A
6	2	R/C	Arch	Trench	Analysis	LRFD	Uses interface elements, Level 2 extended to apply live load, and includes live load rating calculation	Shows how to modify the input file using the text input, the shortcut method for inputting interface angles, viewing live load boundary conditions, and how to compute interface angles for an arch
7	2	R/C	Box	Embankment	Analysis	LRFD	N/A	N/A
8	2	Steel	Pipe	Embankment	Analysis	LRFD	Uses slotted joints and Level 2 extended to add smaller construction steps above the crown of the pipe	N/A
9	2	Steel	Arch	Trench	Analysis	Service	Uses interface elements, Level 2 extended to apply live load, and large deformation / buckling analysis	Shows inputting interface angles using the graphic interface.
10	2	R/C	Pipe	Embankment	Design	LRFD	Includes soft backpacking soil around Pipe	Shows how to modify chart properties of graphs plotted using 'view/graphs.'
11	2	Plastic	Pipe	Trench	Analysis	Service	Profile analysis, includes interface elements, extended Level 2 to add soft haunches, long-term load duration, and global and local buckling calculations	Shows how to modify the input file using the text input including inputting a user defined soil material and how to compute interface angles for a pipe
12	3	R/C	Box	Embankment	Analysis	LRFD	Identical to Problem 7, except Level 3	Shows a user generated Level 3 mesh
13	3	Steel	Arch	Trench	Analysis	Service	Identical to Problem 9, except Level 3	Shows how to import a Level 3 mesh using xml format
14	3	Alum., R/C	Arch	Embankment	Analysis	Service	Two material structure with pin connection at point where materials intersect, manual input of interface elements to model pin connection	Shows how to import a Level 3 mesh using xml format
15	3	Plastic	Multiple Arches	Trench	Analysis	LRFD	Includes global and local buckling calculations and live load	Input file only, no written tutorial
16	3	Plastic, Steel	Pipe	Trench	Analysis	Service	Three material structure with a plastic liner grouted to a steel host pipe, includes interface elements between grout and plastic liner	Input file only, no written tutorial

Table 1-2 – Tutorial Example Descriptions

Problem	Description
1	**Level-1, Corrugated Steel Pipe, Design mode (LRFD).** Design a 60 in. inside diameter corrugated steel pipe with 30 ft of fill over the top of the pipe using LRFD design. The design will be with Level 1, which is based on the Burns and Richard elasticity solution. The desired result is the corrugation size and thickness.
2	**Level-1, Reinforced Concrete Pipe, Design mode (LRFD).** Design a 60 in. inside diameter reinforced concrete pipe with 30 ft of fill over the top of the pipe using LRFD design. The design will be with Level 1, which is based on the Burns and Richard elasticity solution. The desired result is the required inner and outer reinforcement.
3	**Level-1, HDPE Plastic pipe, Design mode (LRFD).** Design a 36 in. outside diameter smooth wall HDPE plastic pipe with 40 ft of fill over the top of the pipe using LRFD design. The design will be with Level 1, which is based on the Burns and Richard elasticity solution. The desired result is the wall thickness.
4	**Level-2, Corrugated Aluminum Pipe, Design mode (Working Stress).** Design a 60 in. inside diameter corrugated aluminum pipe with 30 ft of fill over the top of the pipe using Working Stress (service) design. The design will be with Level 2, using an automated finite element pipe mesh for a trench installation having no interface elements. The desired result is the corrugation size and thickness.
5	**Level-2, HDPE Plastic Pipe, Analysis mode (Working Stress).** Analyze a 36 in. outside diameter smooth wall HDPE plastic pipe with 40 ft of fill over the top of the pipe using Working Stress (service) analysis. The analysis will be with Level 2, using an automated finite element pipe mesh for an embankment installation having no interface elements.
6	**Level-2, Reinforced Concrete Arch, Analysis mode (LRFD).** Analyze a 237-inch span (90-inch rise) reinforced concrete arch supported on spread footings with 2 ft of fill over the top of the arch, using LRFD analysis. The problem is shown schematically in Figure 6-1. The analysis will be with Level 2, using an automated finite element arch mesh for a trench installation having interface elements. The automated finite element mesh will be modified using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch. Additionally, the live load rating procedure will be demonstrated using CANDE output.
7	**Level-2, Reinforced Concrete Box, Analysis mode (LRFD).** Analyze a 120 in. x 84 in. reinforced concrete box culvert with standard ASTM steel placement with 2 ft of fill over the top of the culvert using LRFD analysis. The analysis will be with Level 2, using an automated finite element box mesh for an embankment installation.
8	**Level-2, Corrugated Steel Pipe, Analysis mode (LRFD).** Analyze a 144 in. corrugated steel pipe with 8 slotted joints and 60 ft of fill over the top of the pipe using LRFD analysis. The problem is shown schematically in Figure 8-1. The analysis will be with Level 2, using an automated finite element pipe mesh for an embankment installation having no interface elements. The automated finite element mesh will be modified using Level 2-extended to reduce the thickness of the construction steps above the crown of the pipe.

Problem	Description
9	**Level-2, Corrugated Steel Long Span, Analysis mode (Working stress).** Analyze a 217-inch span (82-inch rise) 3-segment type corrugated steel long span arch supported on spread footings with 3 ft of fill over the top of the arch, using Working Stress (service) analysis. The problem is shown schematically in Figure 9-1. The analysis will be with Level 2, using an automated finite element arch mesh for a trench installation having interface elements. The automated finite element mesh will be modified using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch.
10	**Level-2, Reinforced Concrete Pipe, Design mode (LRFD).** Design a 72 in. inside diameter concrete pipe set on gravel bedding with 60 ft of fill over the top of the pipe using LRFD design. The analysis will be with Level 2, using an automated finite element pipe mesh for an embankment installation having a 6 in. layer of soft backpacking soil around the circumference of the pipe and no interface elements. The desired result is the required inner and outer reinforcement.
11	**Level-2, Plastic Pipe (Profile), Analysis mode (Working Stress).** Analyze a 48 in. inside diameter corrugated plastic (profile) pipe with 40 ft of fill over the top of the pipe using Working Stress (service) analysis. The problem is shown schematically in Figure 11-1. The analysis will be with Level 2, using an automated finite element pipe mesh for a trench installation having interface elements. The automated finite element mesh will be modified using Level 2-extended to change the haunch zones to a user defined soil material and the thickness of bedding layer to 6 in.
12	**Level-3, Reinforced Concrete Box, Analysis mode (LRFD).** Analyze a 120 in. x 84 in. reinforced concrete box culvert with standard ASTM steel placement with 2 ft of fill over the top of the culvert using LRFD analysis. The analysis will be with Level 3, using a user generated finite element mesh for an embankment installation. This problem analyzes the reinforced concrete box culvert from Tutorial Problem 7, which was performed using a Level 2 analysis.
13	**Level-3, Corrugated Steel Long Span, Analysis mode (Working stress).** Analyze a 217-inch span (82-inch rise) 3-segment type corrugated steel long span arch setting on concrete footings with 3 ft of fill over the top of the arch using Working Stress (service) analysis. The analysis will be with Level 3, using an imported finite element arch mesh in xml format from Tutorial Problem 9 for a trench installation having interface elements. This problem analyzes the corrugated steel long span arch from Problem 9, which was performed using a Level 2 analysis.
14	**Level-3, Reinforced Concrete and Corrugated Aluminum Arch, Analysis mode (Working Stress).** Analyze a two material structure composed of a reinforced concrete U-shaped base with 15-foot span and 5-foot rise supporting a pin connected, corrugated aluminum arch-shaped roof with 13 ft of fill over the top of the arch. The problem is shown schematically in Figure 14-1. The analysis will be with Level 3, using an imported finite element mesh in xml format.
15	**Level-3, Multiple Plastic Arches, Analysis mode (LRFD).** Analyze three corrugated plastic arches with 42 in. span and 27 in. rise placed side by side with 8.5 in. spacing between the legs (storm water retention chambers) with 2 feet of soil over the top of the arches. The analysis will be with Level 3, using a user generated finite element mesh for a trench installation. The desired analysis result is to evaluate LRFD local and global buckling. **Input File Only**
16	**Level-3, Corrugated Steel Pipe Retrofitted with Plastic Pipe Liner, Analysis mode (Working Stress).** Analyze a 48 in. corrugated steel pipe with an eroded invert and retrofitted with a profile plastic pipe with 5 feet of fill over the top of the pipe. The analysis will be with Level 3, using a user generated finite element mesh for a trench installation. **Input File Only**

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 1

May 2008

This page intentionally left blank

1. CANDE TEST PROBLEM 1

1.1 Problem Definition

Design a 60 in. inside diameter corrugated steel pipe with 30 ft of fill over the top of the pipe using LRFD design. The problem is shown schematically in Figure 1-1. The design will be with Level 1, which is based on the Burns and Richard elasticity solution. The desired result is the corrugation size and thickness.

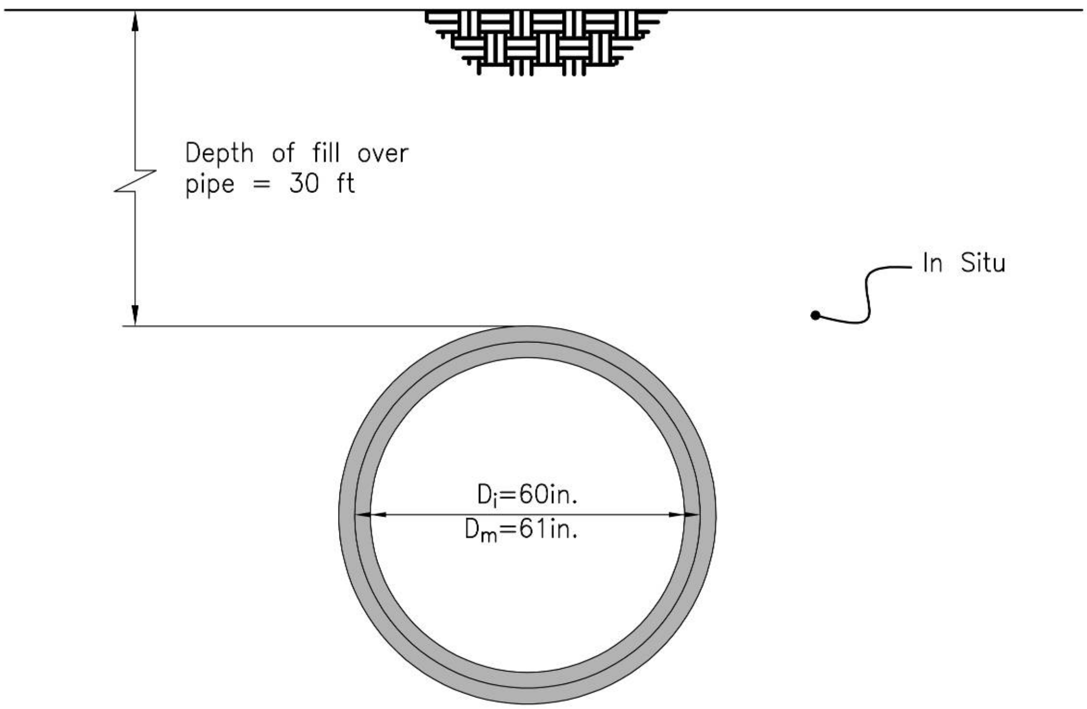


Figure 1-1 – Details of Problem 1 – Level 1, infinite elastic soil

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Design

Method of analysis/design - LRFD

Solution level - Elasticity (Level 1)

Pipe material type - Steel

Joint slip - No joint slip

Material behavior - Linear stress/strain

Analysis mode - Small deformation

Average diameter of pipe - Set pipe diameter to centroid of pipe wall. Assume design will be met with a 1 in. deep corrugation, therefore set to nominal diameter + 1 in. = 61 in.

Soil density of backfill material - 120 lb/ft^3

Number of load steps - Apply load in 10 equal load steps of 3 ft of soil.

Interface - Fully bonded

Soil height - 10 equal load steps of 3 ft – Input for Solution Level 1 assumes height above crown

Soil parameters - Elastic soil model, Solution Level 1 is suitable for homogenous soil conditions only. Assume lightly compacted silt, Young's modulus = 1,000 psi, Poisson's ratio = 0.25, and soil density = 120 lb/ft^3.

LRFD load factor - 1.95

Load modifier - 1.05 (non-redundant for earth load)

1.2 Creating the CANDE Input Document

Figures 1-2 through 1-3 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 2, CANDE will display a screen indicating that an input path is initiated – click “Finish.” The user is prompted for a filename and directory.

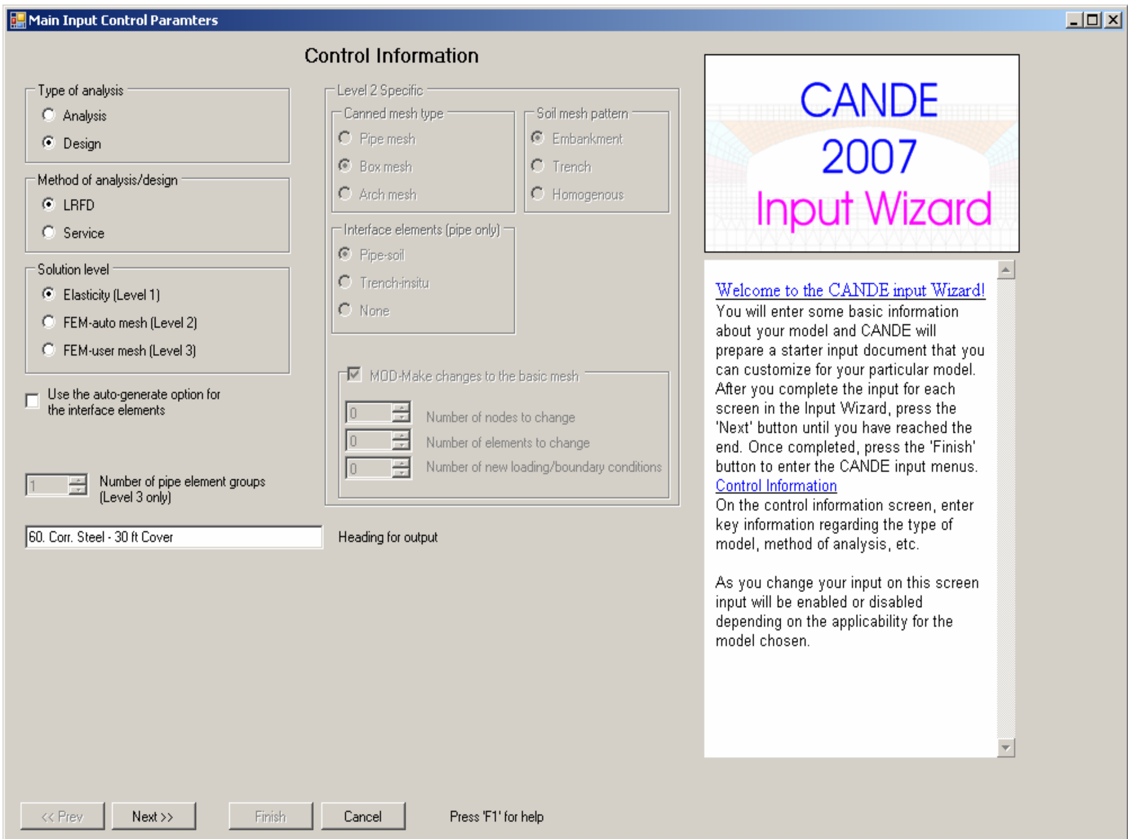


Figure 1-2 – Input Wizard, Screen 1

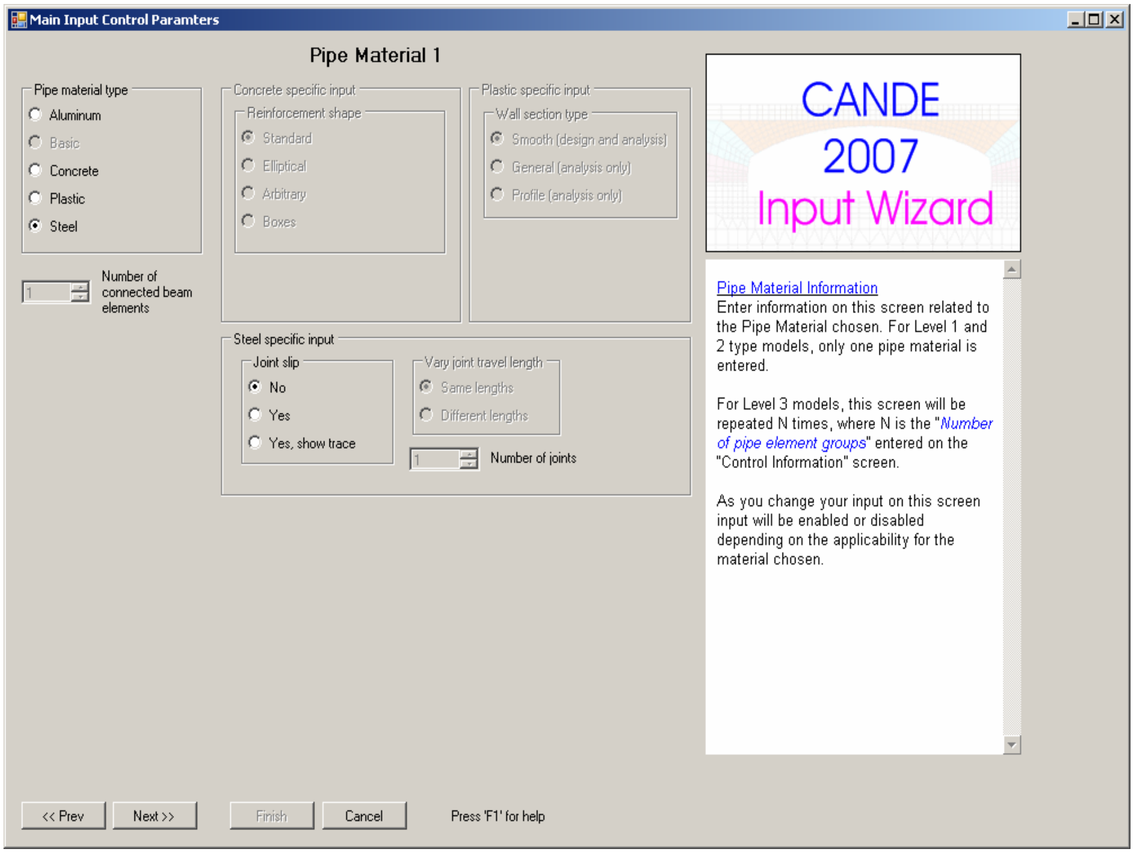


Figure 1-3 – Input Wizard, Screen 2

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 1-4. Enter an appropriate heading for output and click “Accept Input.”

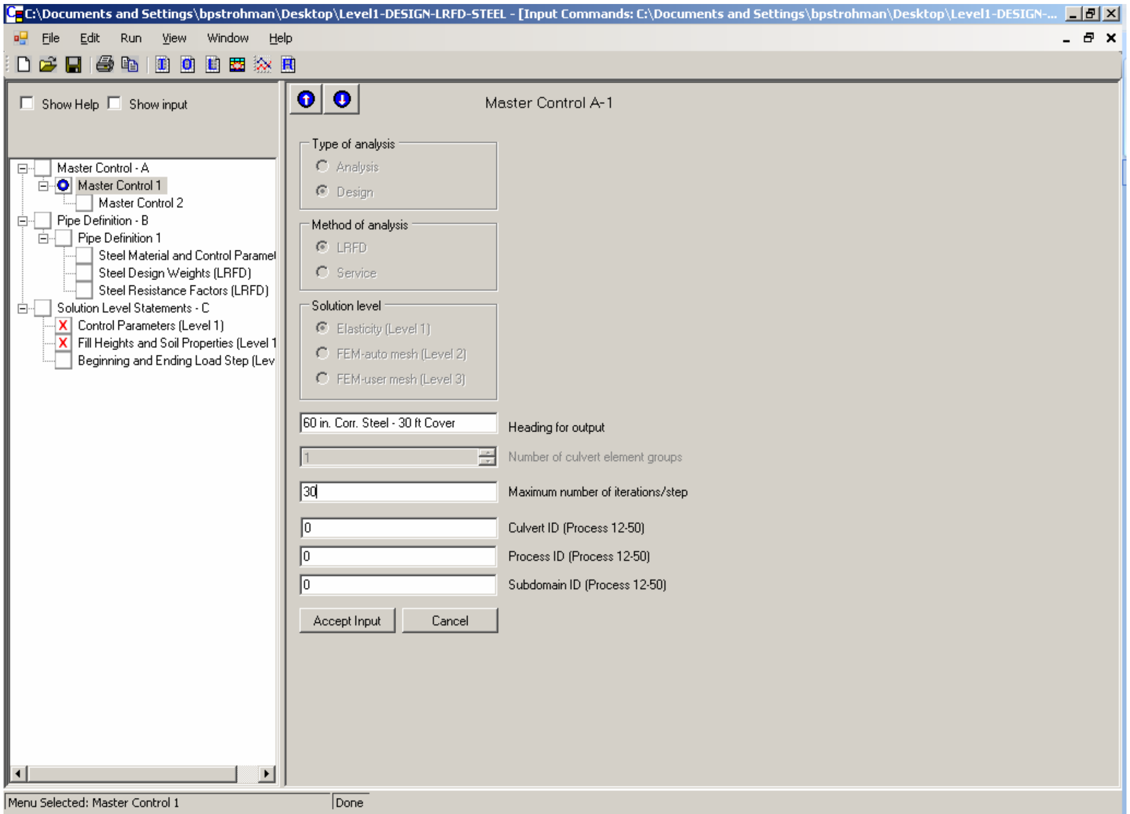


Figure 1-4 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 1-4 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 1-5 through 1-8 show the completed input for the screens requiring data for the tutorial. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

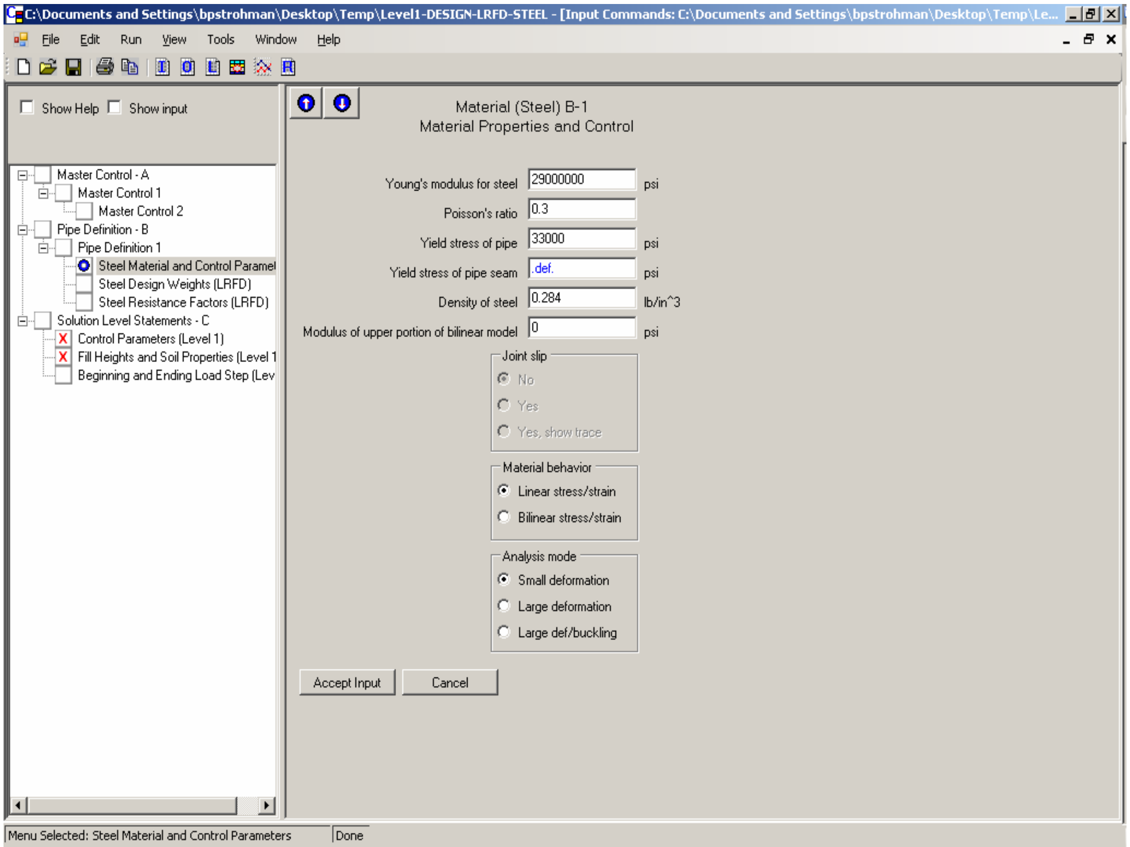


Figure 1-5 – Input Screen B-1

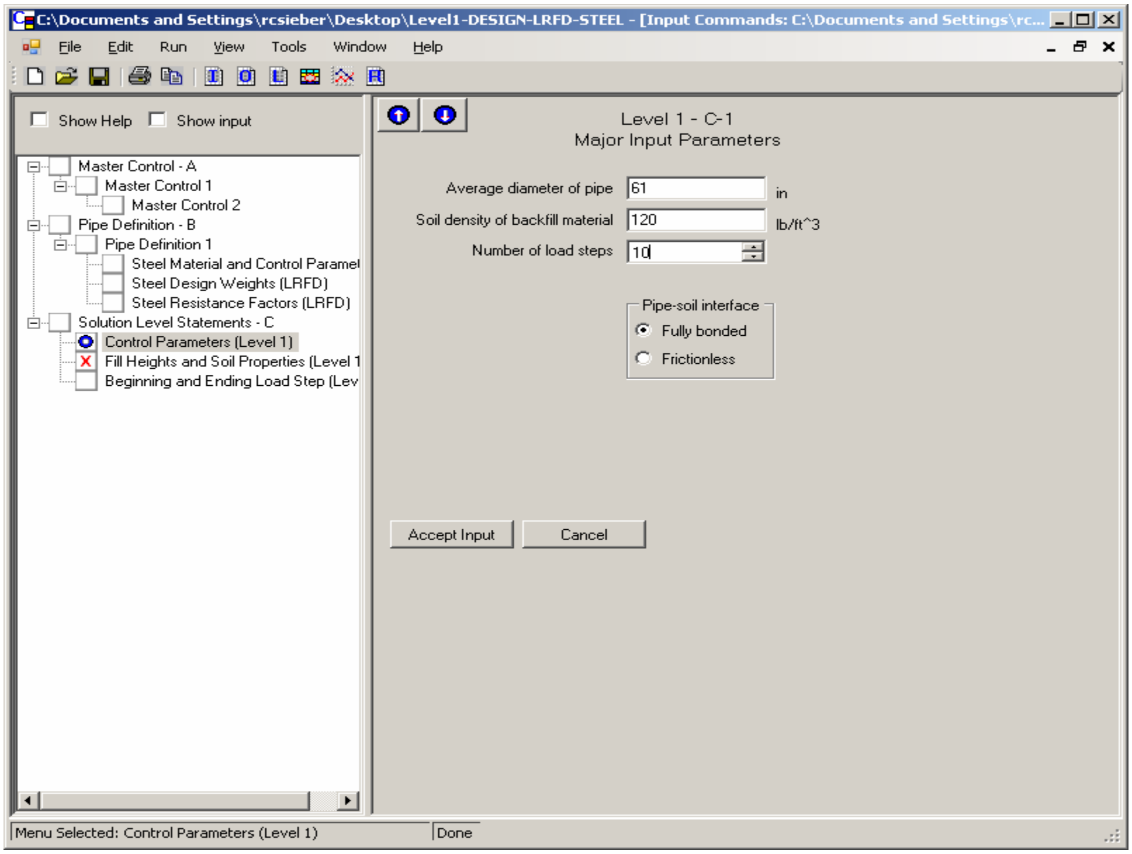


Figure 1-6 – Input Screen C-1

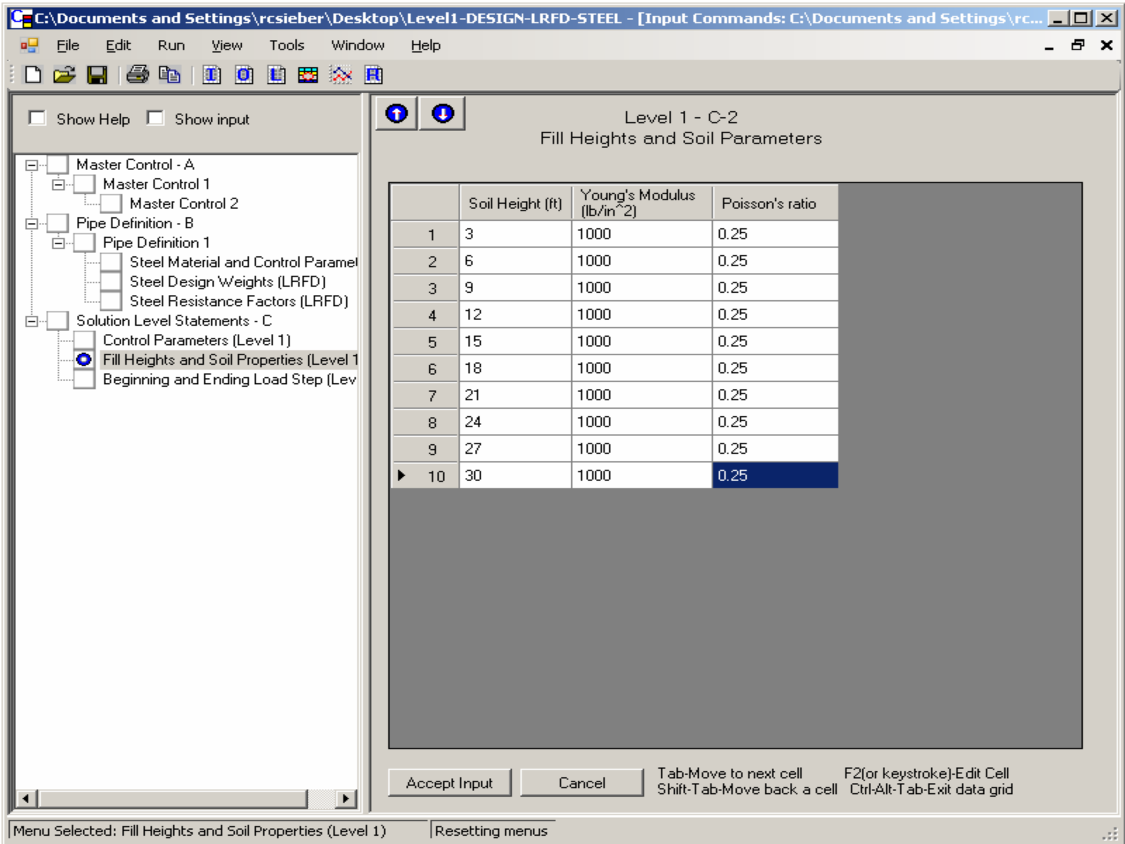


Figure 1-7 – Input Screen C-2

Under LRFD load factor in Figure 1-8 enter the combined value of the load factor and load modifier, i.e. – 1.95 * 1.05 = 2.05

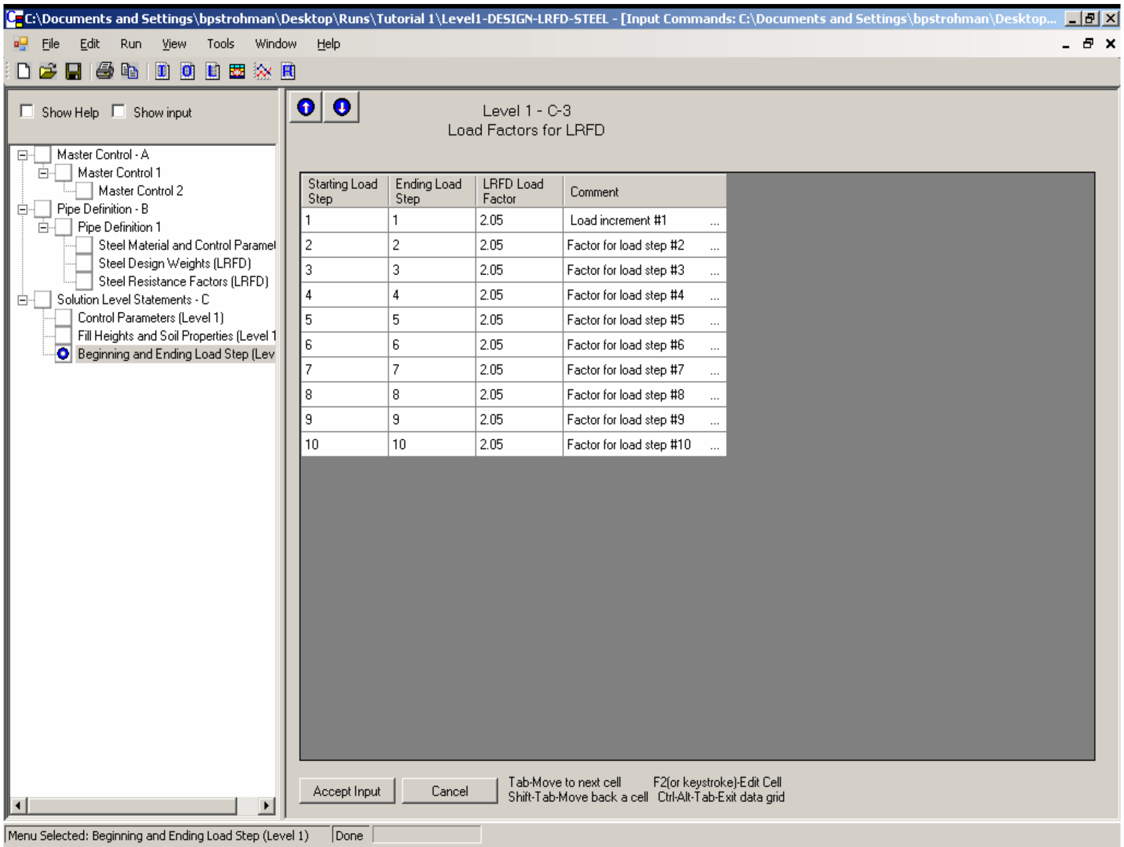


Figure 1-8 – Input Screen C-3

When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow the user to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

1.3 Reviewing and Interpreting the Output

Now proceed to check the output file. The user can select “View/Output Report (CANDE)” or select the “View CANDE output” button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data, the design solution (Figure 1-9), or the design assessment (Figure 1-10) by clicking on the appropriate node of the ‘Table of Contents’ browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 1-10 shows the final assessment printed at the end of the file.

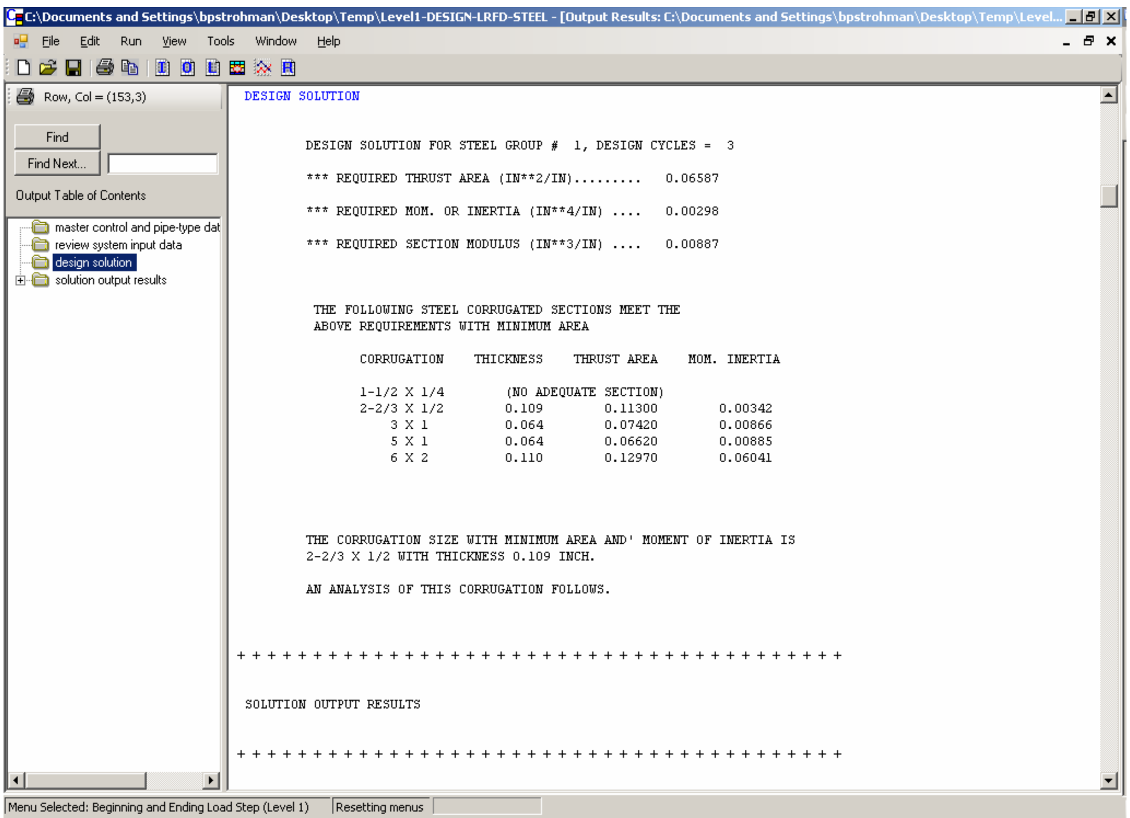


Figure 1-9 – Design Solution

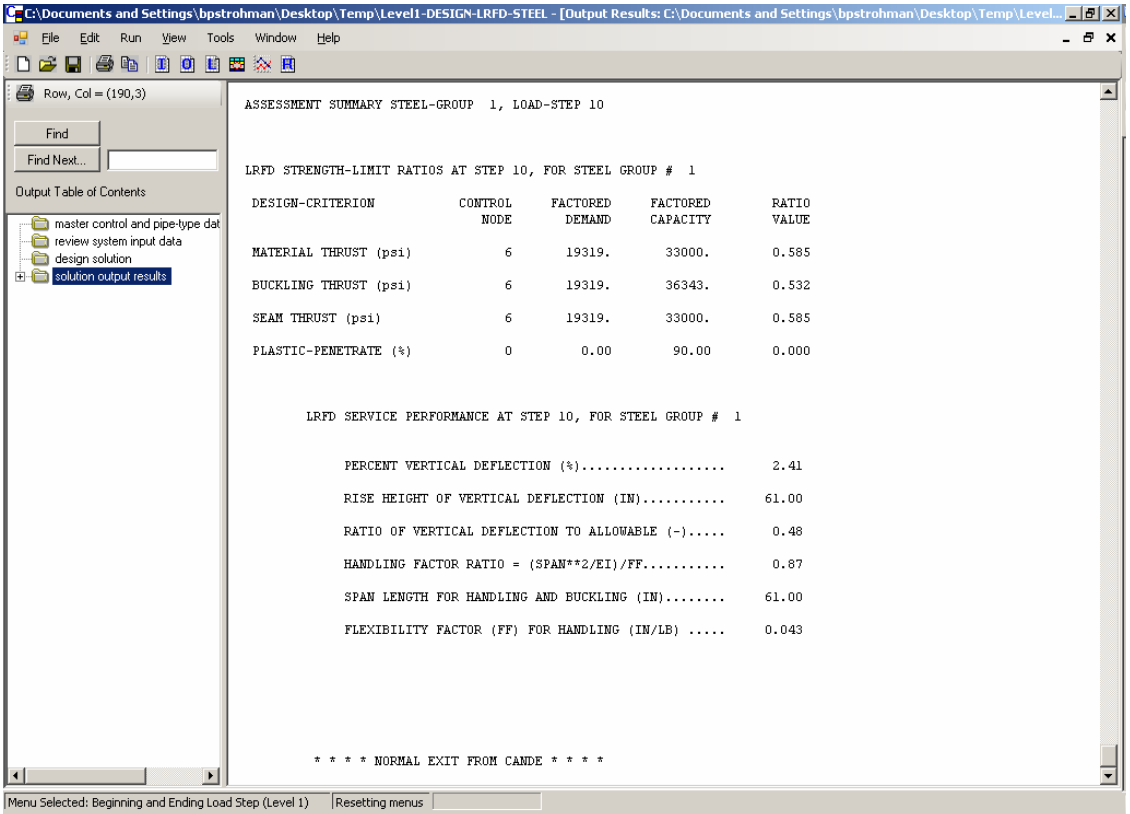


Figure 1-10 – Design Assessment Summary – Load Step 10

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 2

May 2008

This page intentionally left blank

2. CANDE TEST PROBLEM 2

2.1 Problem Definition

Design a 60 in. inside diameter reinforced concrete pipe with 30 ft of fill over the top of the pipe using LRFD design. The problem is shown schematically in Figure 2-1. The design will be with Level 1, which is based on the Burns and Richard elasticity solution. The desired result is the required inner and outer reinforcement.

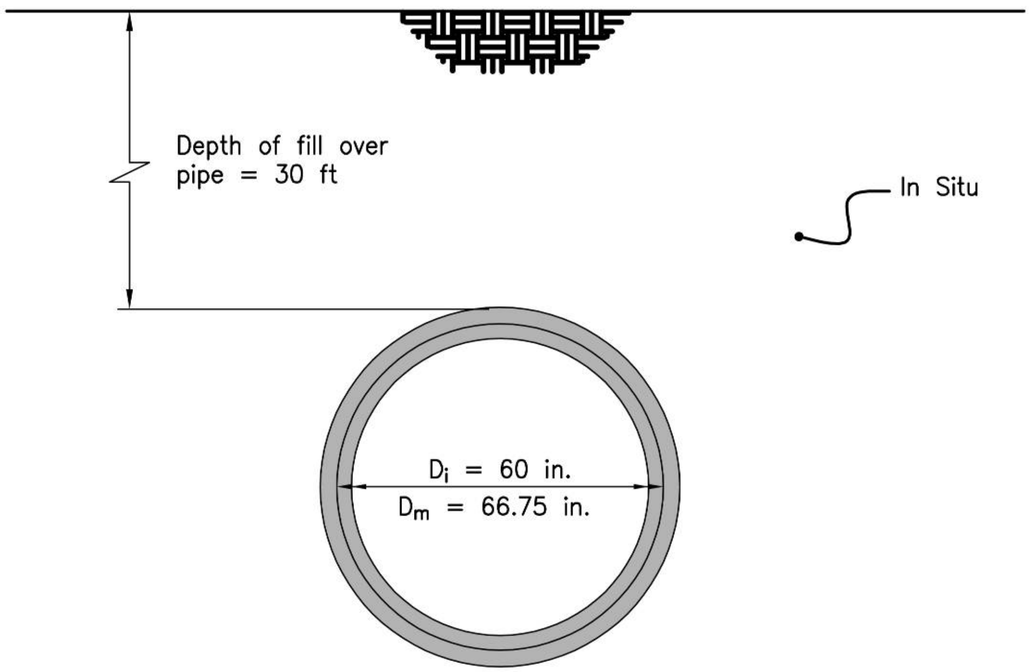


Figure 2-1 Details of Problem 2 – Level 1, infinite elastic soil

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Design

Method of analysis/design - LRFD

Solution level - Elasticity (Level 1)

Pipe material type - Concrete

Reinforcement shape - Standard

Compressive strength of concrete (f'c) - 5,000 lb/in.^{2}

Shear strength equation - Pipes/arches (AASHTO 12.10.4.2.5)

Concrete strain at tension rupture - 0.0 in./in.

Note: Assuming zero tensile strength for the concrete assures the design will be for a cracked section.

Compressive strain at the initial strength limit - 0.002 in./in.

Crack width model - Heger-McGrath (AASHTO 12.10.4.2.4d)

Analysis mode - Small deformation

Yield stress of reinforcing steel - 65,000 lb/in.^{2}

Young's modulus of reinforcing steel - 29,000,000 lb/in.^{2}

Poisson's ratio of reinforcement - 0.3

Inner surface spacing between rows of rebar - 2 in.

Outer surface spacing between rows of rebar - 2 in.

Number of inner cage layers of reinforcement - 1

Number of outer cage layers of reinforcement - 1

Type of reinforcement - Welded wire fabric

Nonlinear behavior selection - Option 3 plus steel yielding behavior

Concrete wall thickness - 6.75 in.

Concrete cover to c.l. of steel rebar cage - 1.25 in. (inside and outside reinforcement)

Ratio of steel areas of outer-to-inner cages - 0.75

Average diameter of pipe - Set pipe diameter to centroid of pipe wall. Assume design will be met with a 6.75 in. thick wall, therefore set to nominal diameter + 6.75 in. = 66.75 in.

Soil density of backfill material - 120 lb/ft^3

Number of load steps - Assume 1 load step of 30 ft

Interface - Fully bonded

Soil height - 30 ft – Input for Solution Level 1 assumes height above crown

Soil parameters - Elastic soil model; solution Level 1 is suitable for homogenous soil conditions only. Assume granular soil with good compaction, Young's modulus = 1,000 psi, Poisson's ratio = 0.30, and soil density = 120 lb/ft^3.

LRFD load factor - 1.35

Load modifier - 1.05 (non-redundant for earth load)

2.2 Creating the CANDE Input Document

Figures 2-2 through 2-3 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 2, CANDE will display a screen indicating that an input path is initiated – click “Finish.” The user is prompted for a filename and directory.

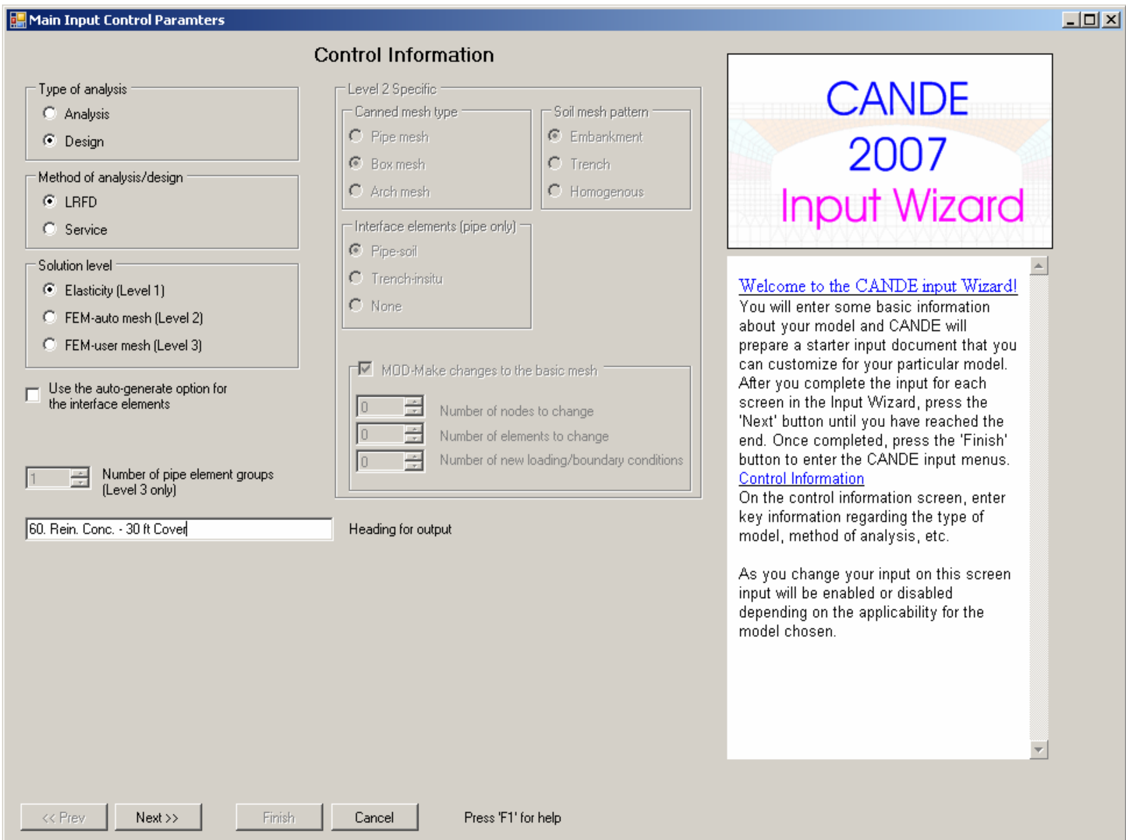


Figure 2-2 – Input Wizard, Screen 1

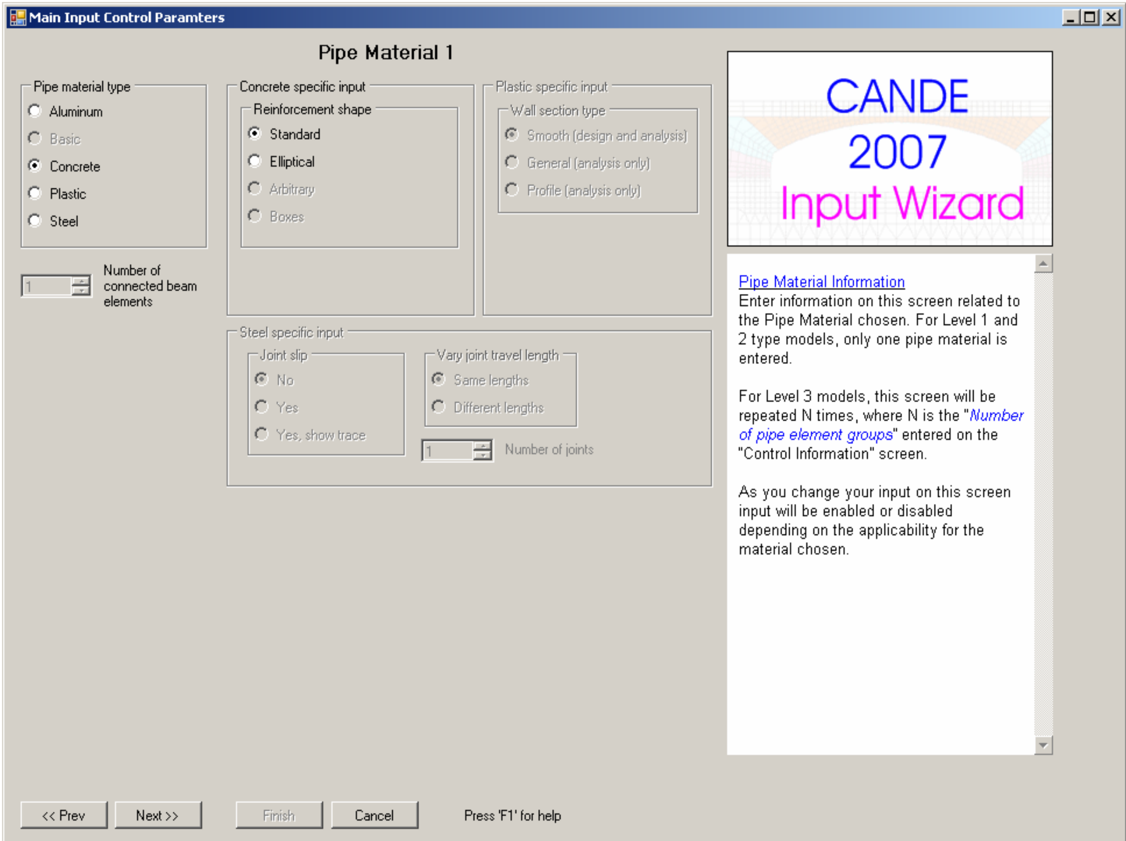


Figure 2-3 – Input Wizard, Screen 2

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 2-4. Enter an appropriate heading for output and click “Accept Input.”

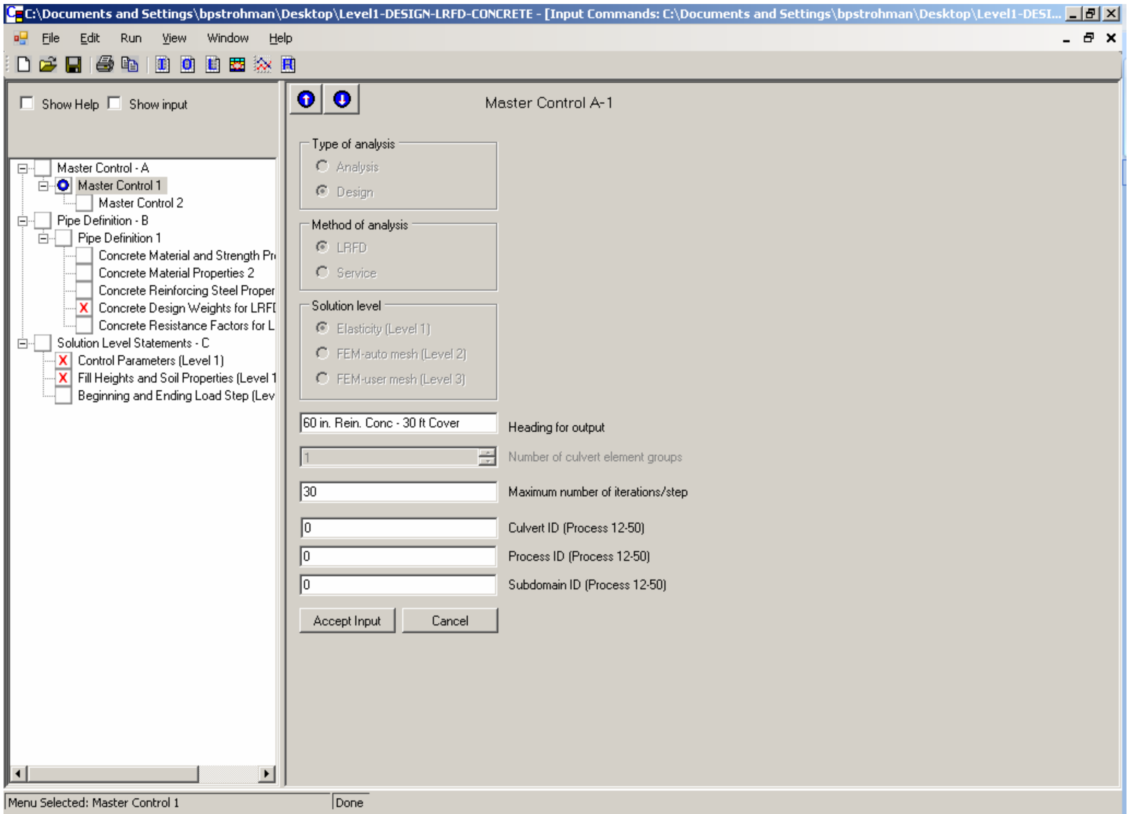


Figure 2-4 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 2-4 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 2-5 through 2-11 show the completed input for the screens requiring data for the tutorial. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

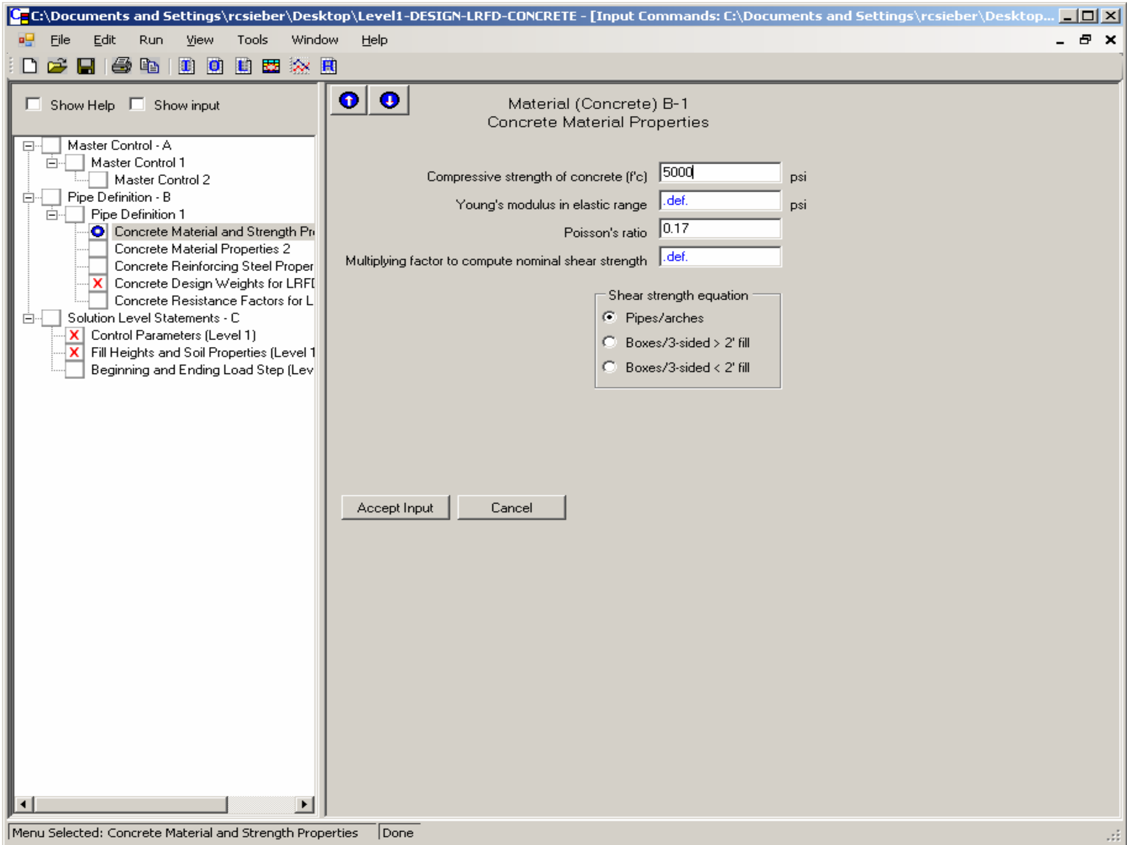


Figure 2-5 – Input Screen B-1

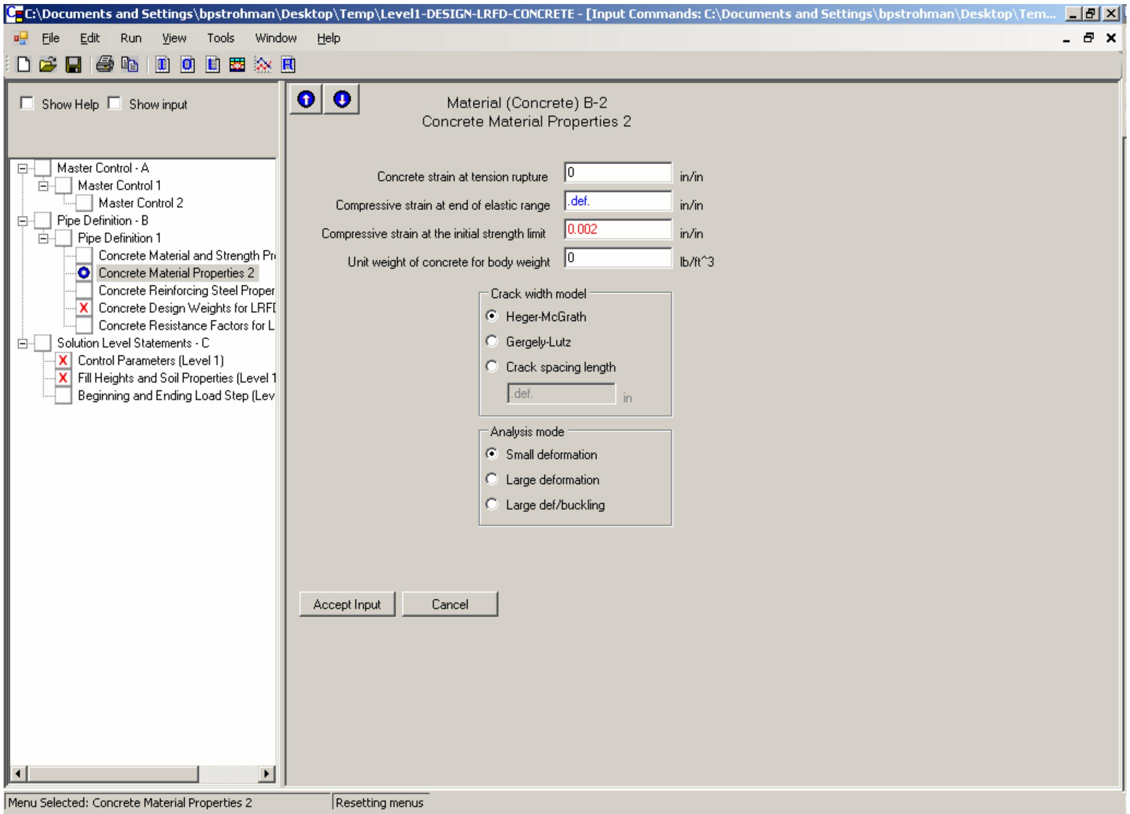


Figure 2-6 – Input Screen B-2

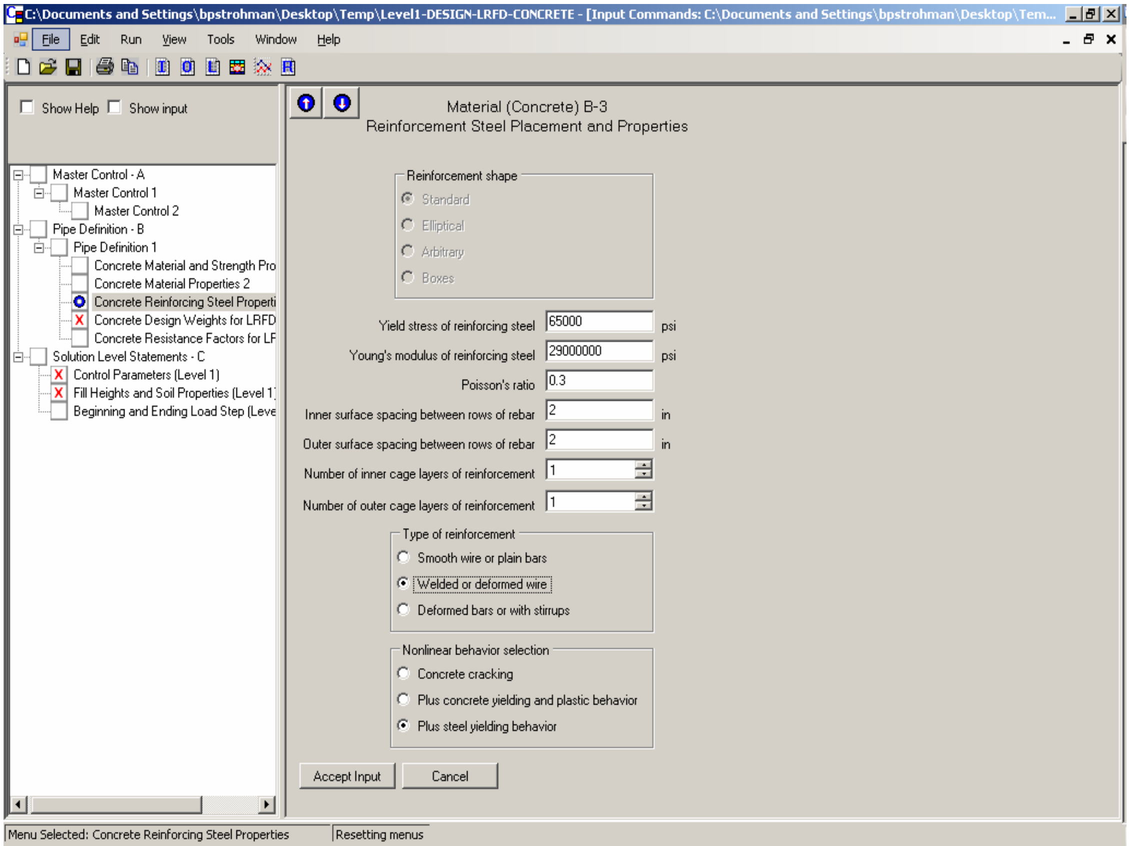


Figure 2-7 – Input Screen B-3

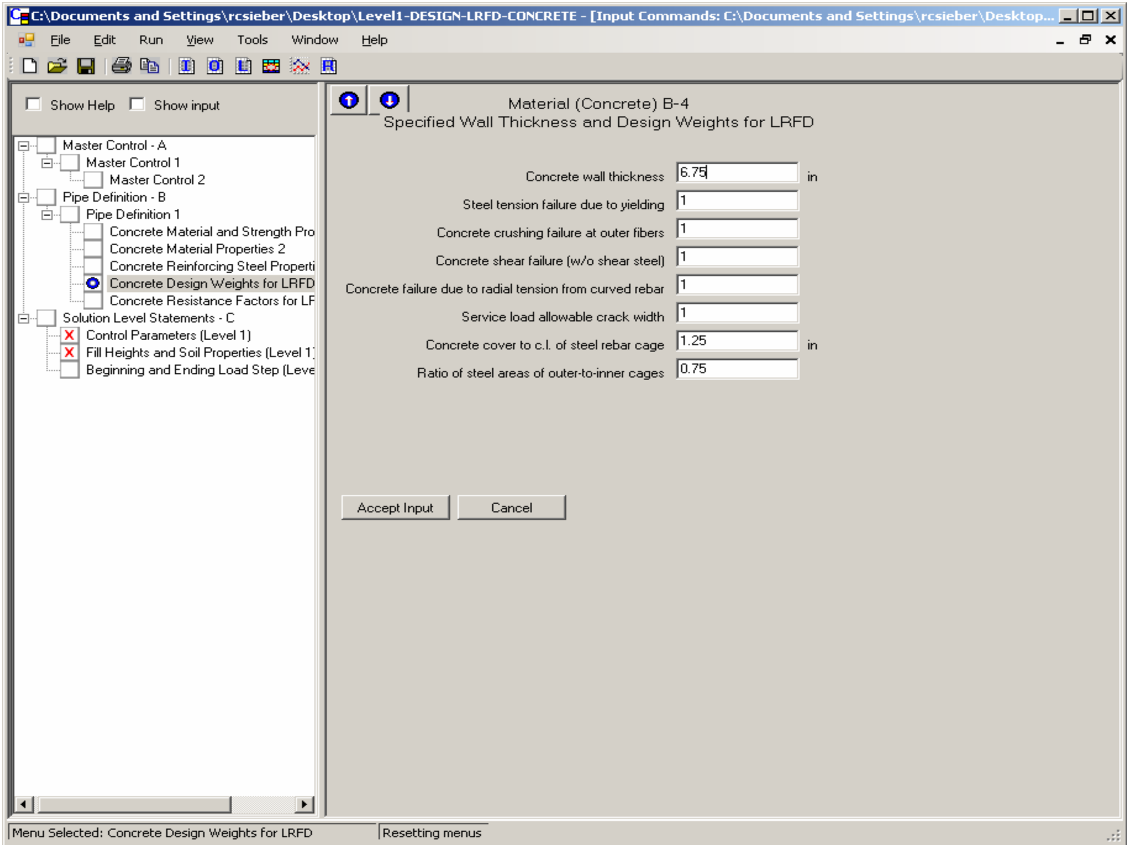


Figure 2-8 – Input Screen B-4

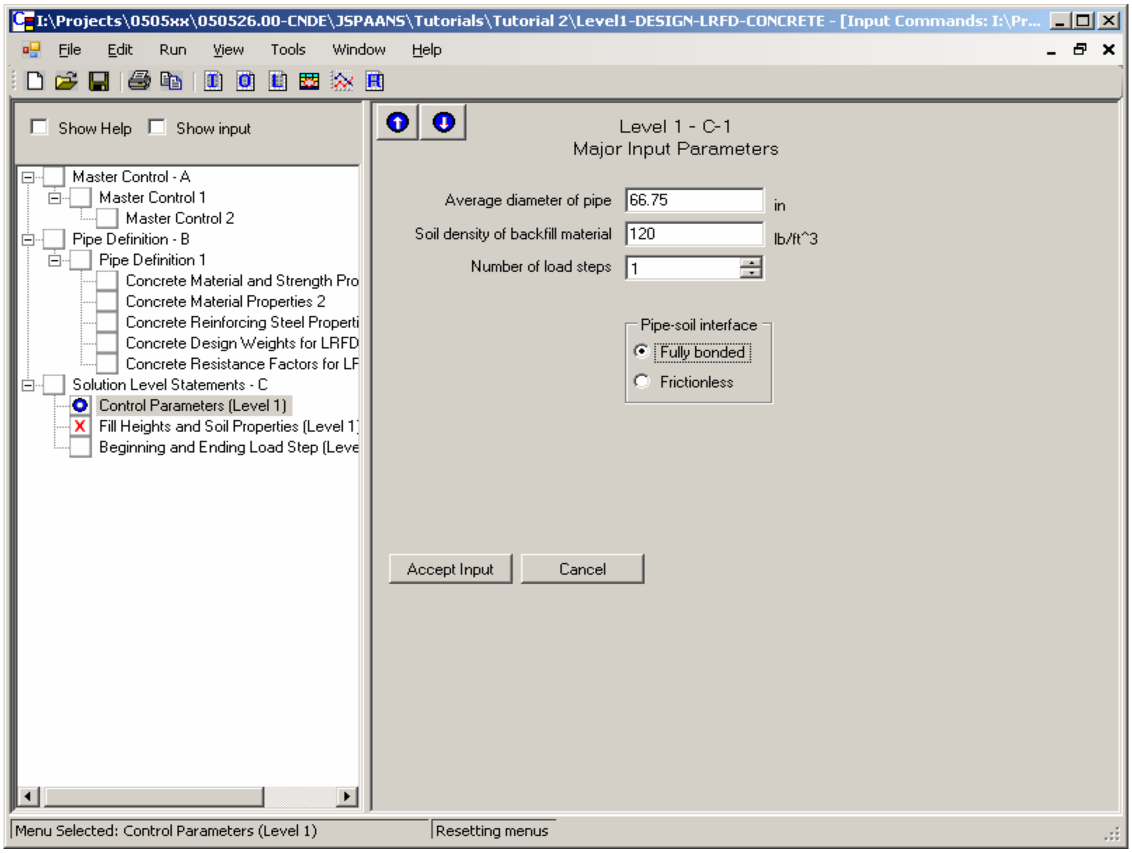


Figure 2-9 – Input Screen C-1

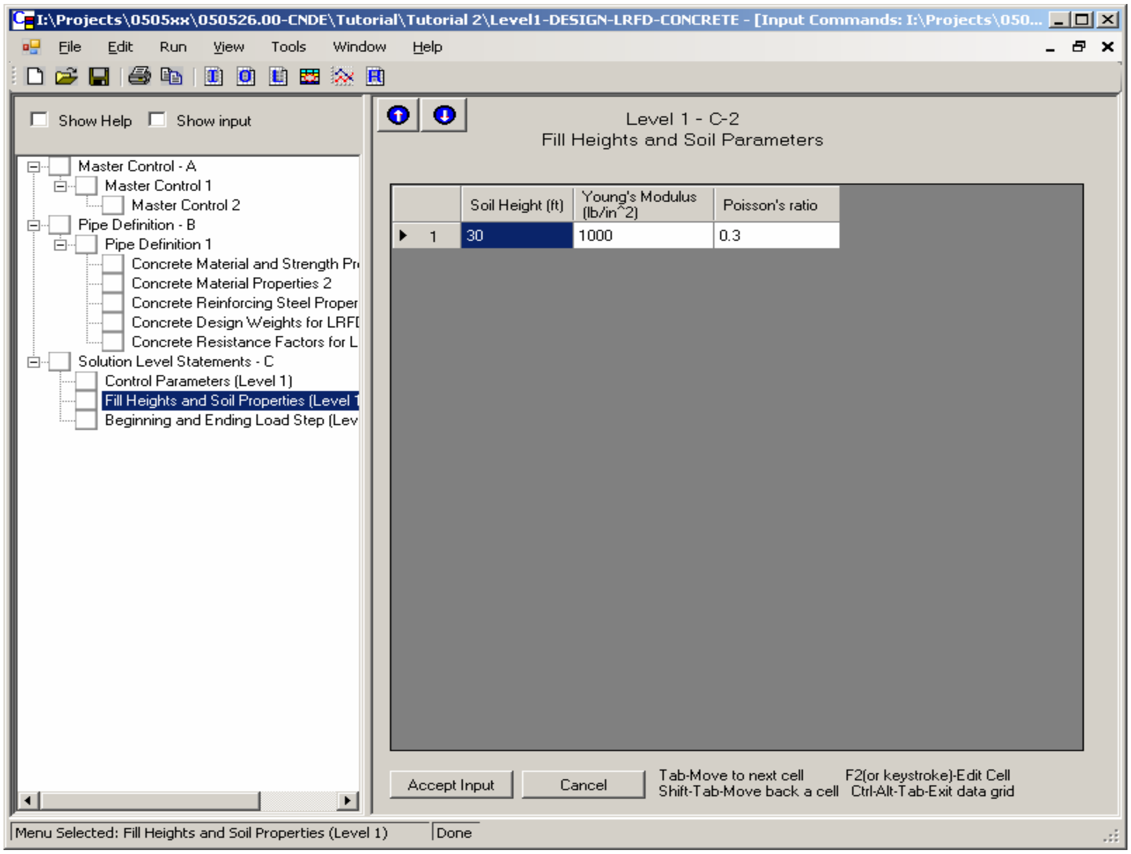


Figure 2-10 – Input Screen C-2

Under LRFD load factor in Figure 2-11 enter the combined value of the load factor and load modifier, i.e. – 1.35 * 1.05 = 1.42

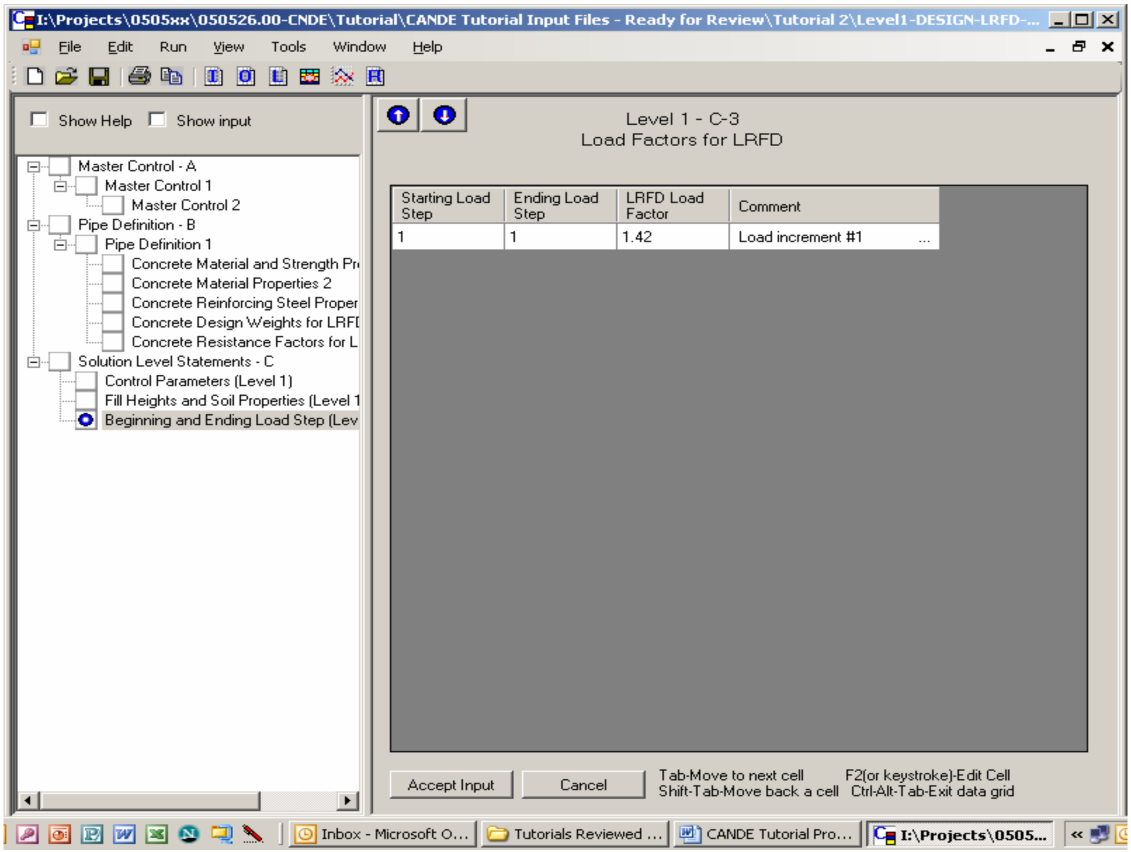


Figure 2-11 – Input Screen C-3

When all input is complete and saved, click "Run" and "CANDE-2007" on the main toolbar to execute the program. This will open the "Running CANDE" window which will allow you to monitor the run as it progresses. When the program is complete, a check box "CANDE Analysis Complete" window will appear. Click "OK" and check if the "Running CANDE" window states "Normal Exit From CANDE." If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the "Close" button in the "Running CANDE" window to return to the user interface.

2.3 Reviewing and Interpreting the Output

Now proceed to check the output file. The user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data, the design solution (Figure 2-12), or the design assessment (Figure 2-13) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 2-13 shows the final assessment printed at the end of the file.

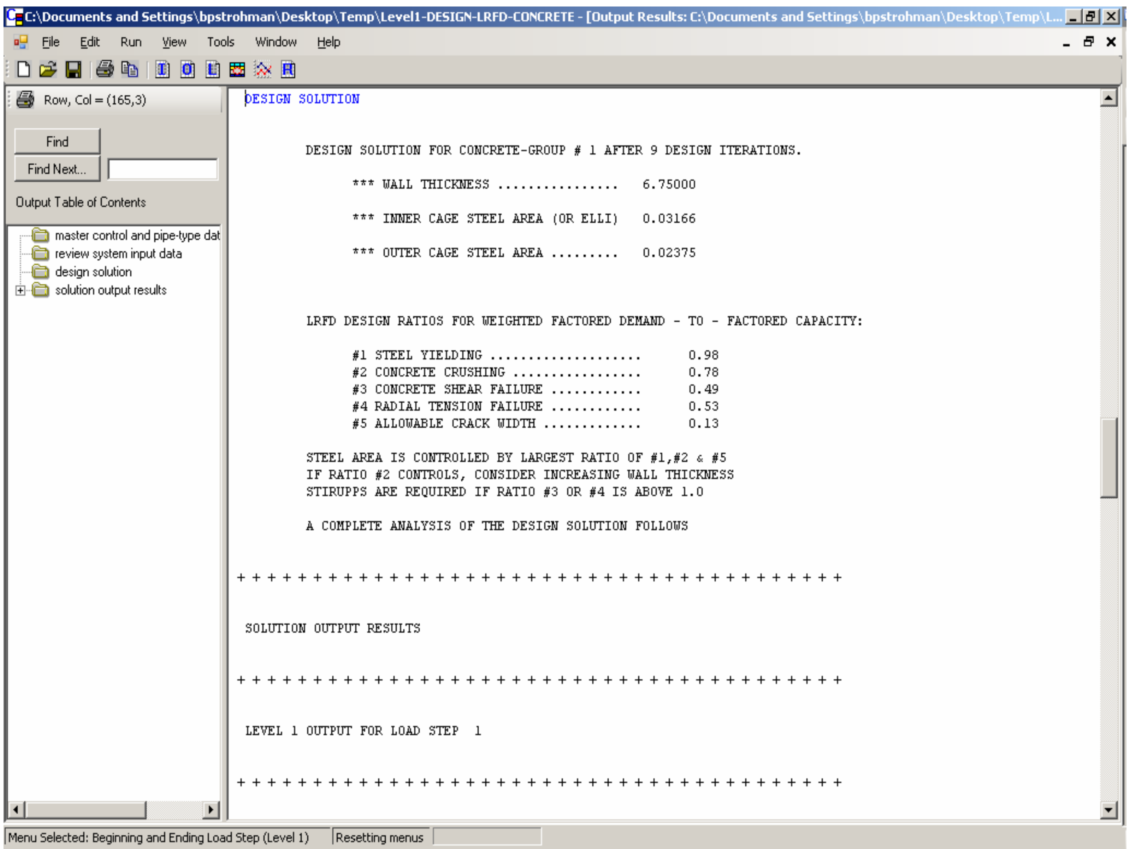


Figure 2-12 – Design Solution

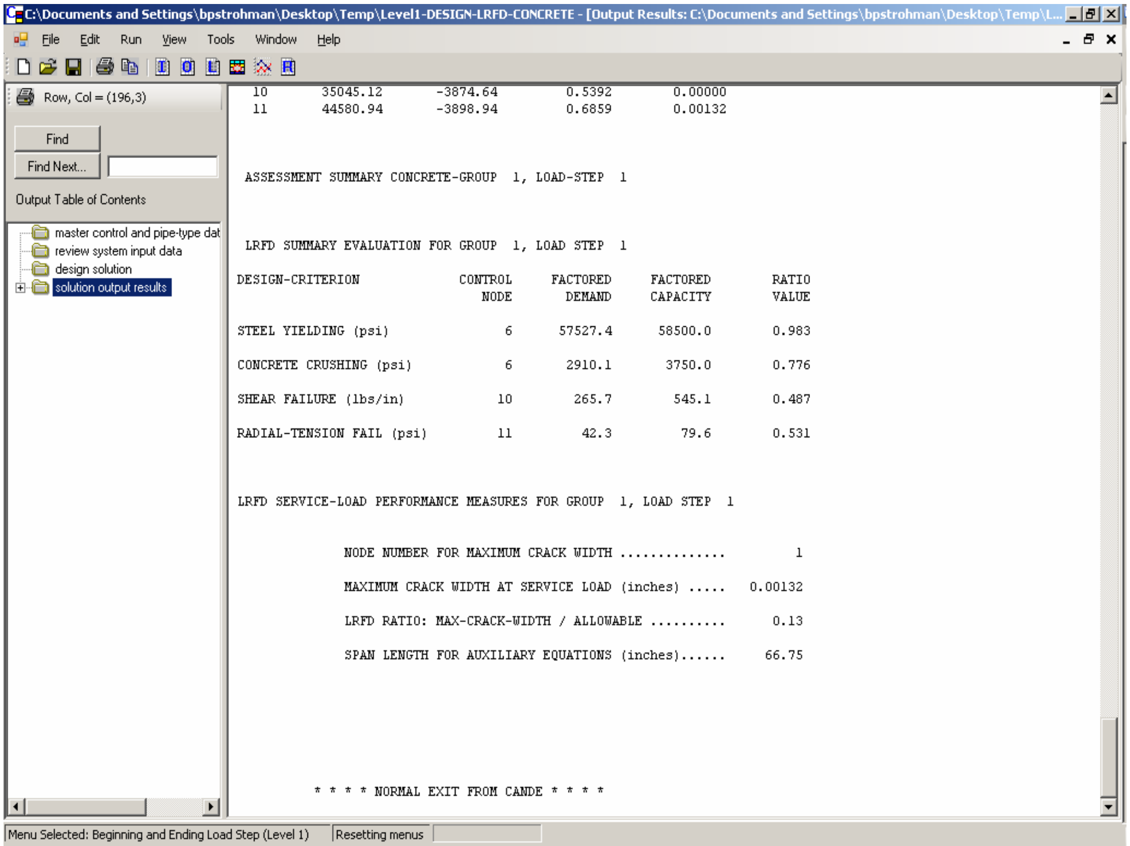


Figure 2-13 – Design Assessment Summary – Load Step 1

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 3

May 2008

This page intentionally left blank

3. CANDE TEST PROBLEM 3

3.1 Problem Definition

Design a 36 in. outside diameter smooth wall HDPE plastic pipe with 40 ft of fill over the top of the pipe using LRFD design. The problem is shown schematically in Figure 3-1. The design will be with Level 1, which is based on the Burns and Richard elasticity solution. The desired result is the wall thickness.

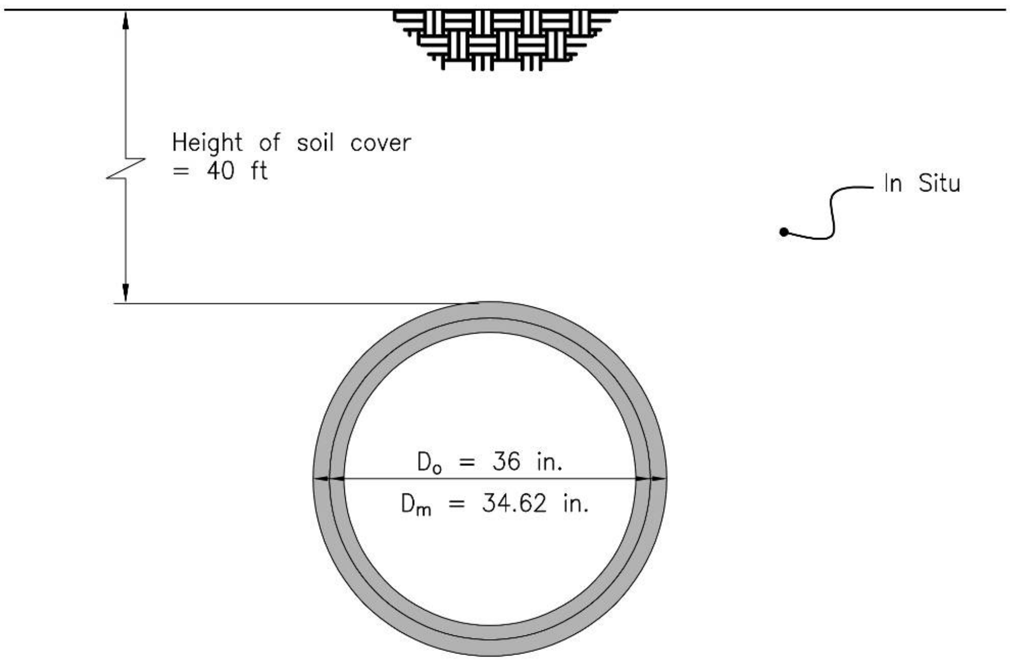


Figure 3-1 Details of Problem 3 – Level 1, infinite elastic soil

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Design

Method of analysis/design - LRFD

Solution level - Elasticity (Level 1)

Pipe material type - Plastic

Wall section type - Smooth

Load duration - Long term

Analysis mode - Small deformation

Young's modulus for short term loading - 80,000 $lb/in.^2$

Ultimate stress limit for short term loading - 3,000 $lb/in.^2$

Young's modulus for long term loading - 22,000 $lb/in.^2$

Ultimate stress limit for long term loading - 1,600 $lb/in.^2$

Poisson's ratio - 0.4

Average diameter of pipe - Set pipe diameter to centroid of pipe wall. Assume pipe is DR 26. D_m = 34.62 in.

Soil density of backfill material - 120 lb/ft^3

Number of load steps - Assume 1 load increment of 40 ft

Interface - Fully bonded

Soil height - 40 ft – Input for Solution Level 1 assumes height above crown

Soil parameters - Elastic soil model, Solution Level 1 is suitable for homogenous soil conditions only. Assume dense coarse-grained, Young's modulus = 4,000 psi and Poisson's ratio = 0.25.

LRFD load factor - 1.95

Load modifier - 1.05 (non-redundant for earth load)

3.2 Creating the CANDE Input Document

Figures 3-2 through 3-3 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking "Next" on Screen 2, CANDE will display a screen indicating that an input path is initiated – click "Finish." The user is prompted for a filename and directory.

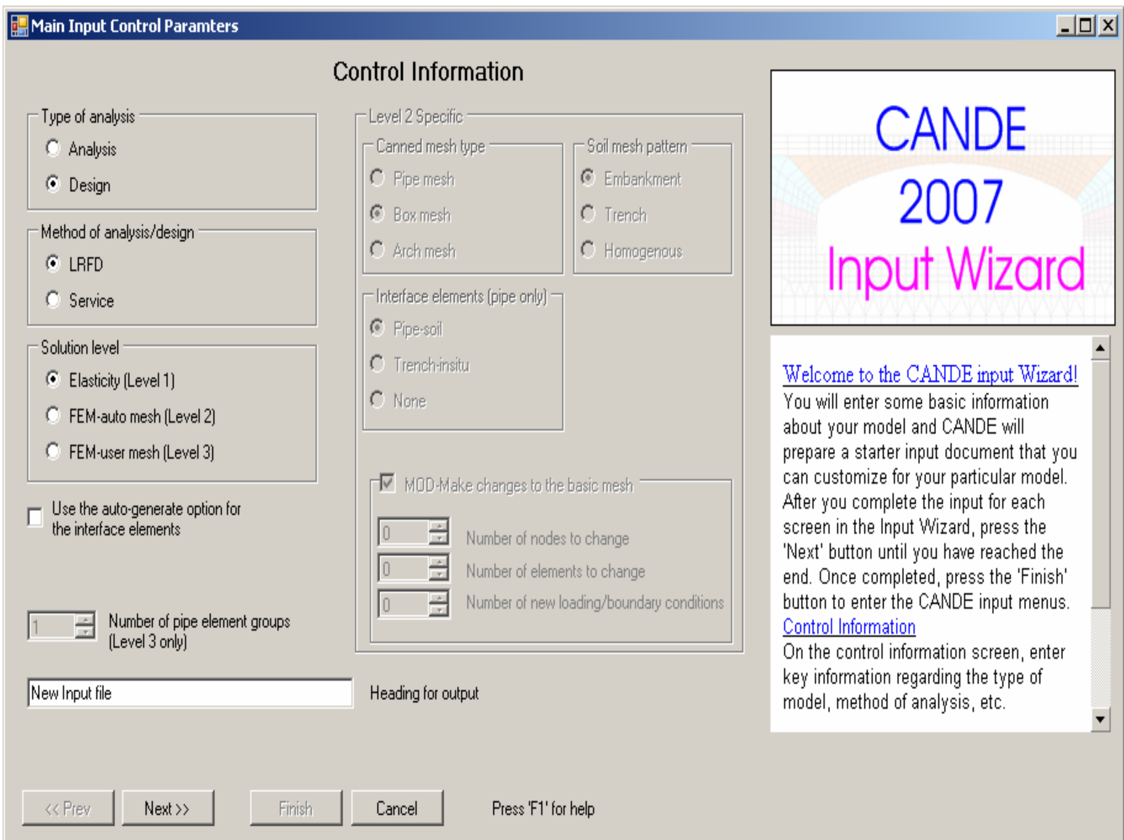


Figure 3-2 – Input Wizard, Screen 1

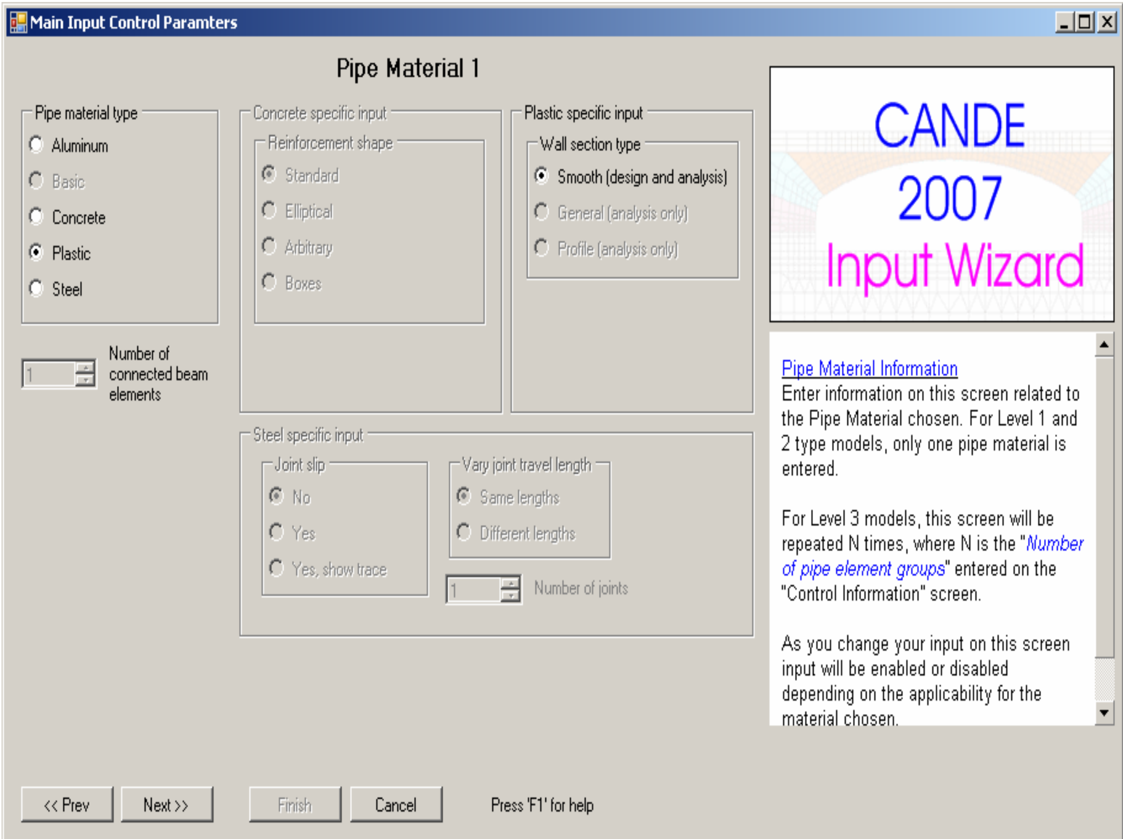


Figure 3-3 – Input Wizard, Screen 2

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 3-4. Enter an appropriate heading for output and click “Accept Input.”

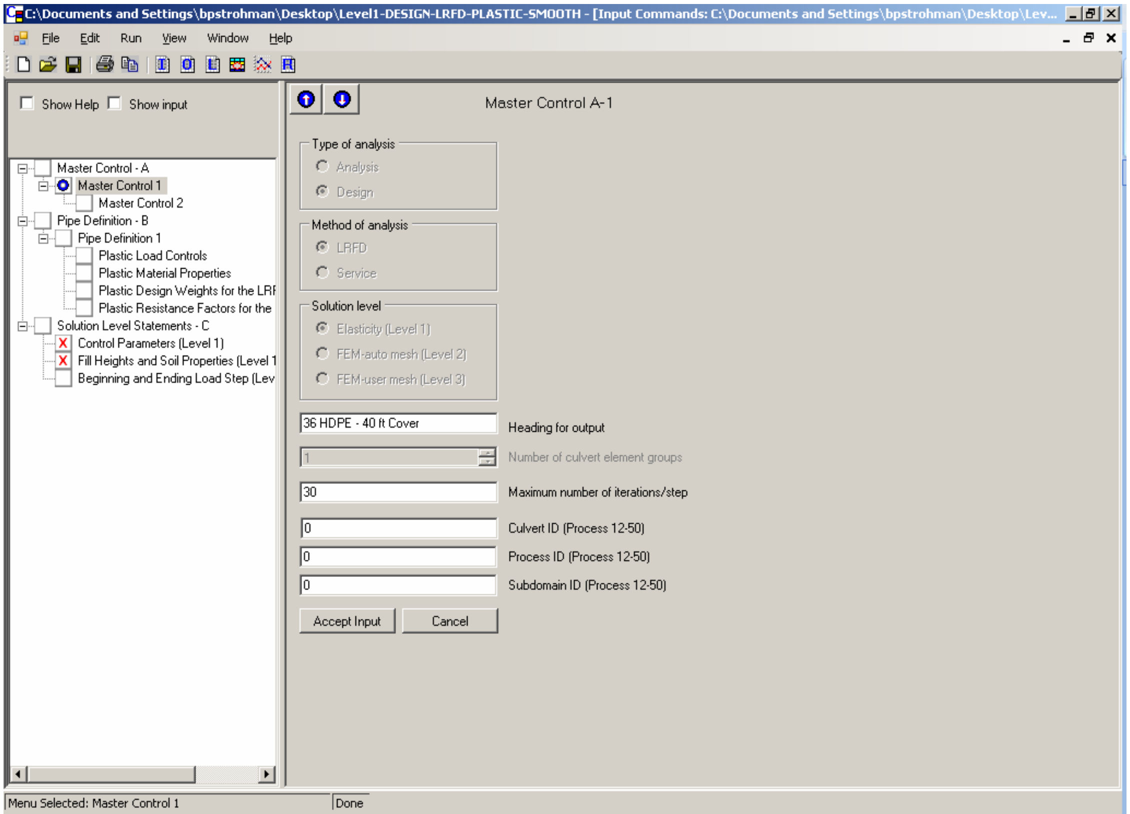


Figure 3-4 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 3-4 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 3-5 through 3-9 show the completed input for the screens requiring data for the tutorial. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

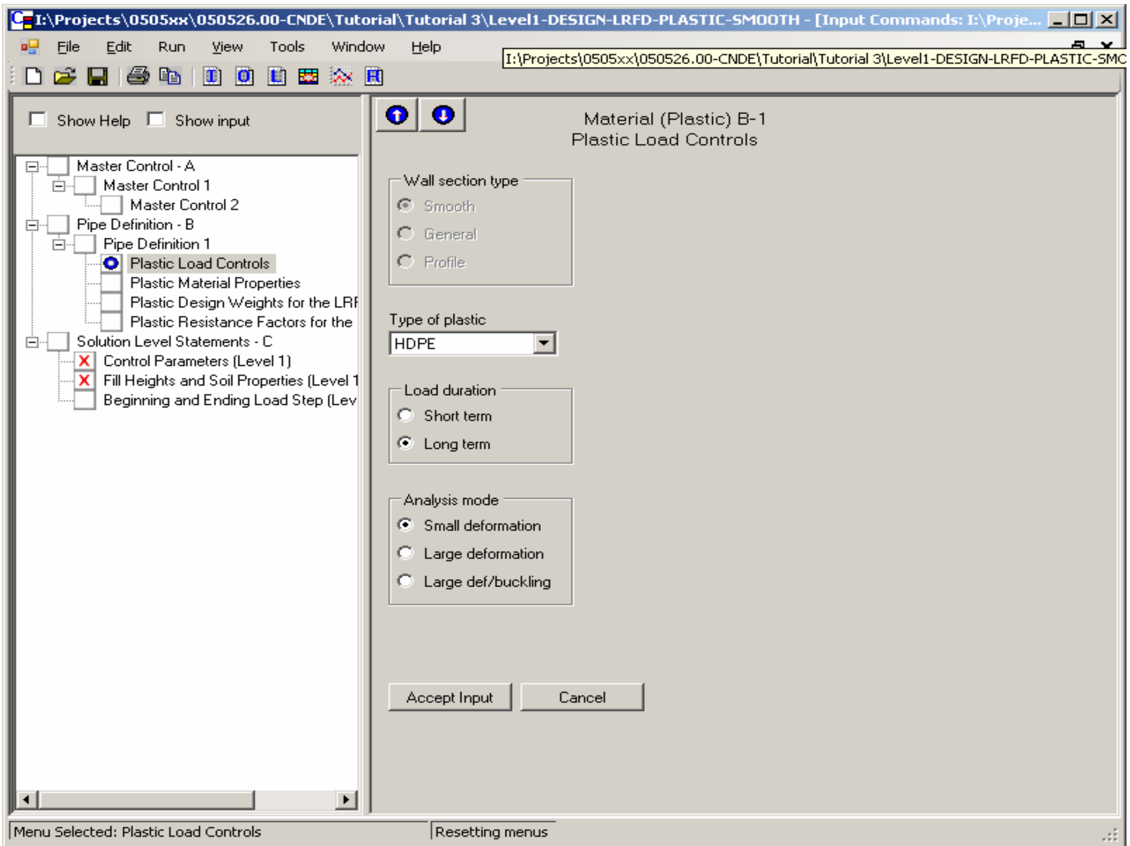


Figure 3-5 – Input Screen B-1

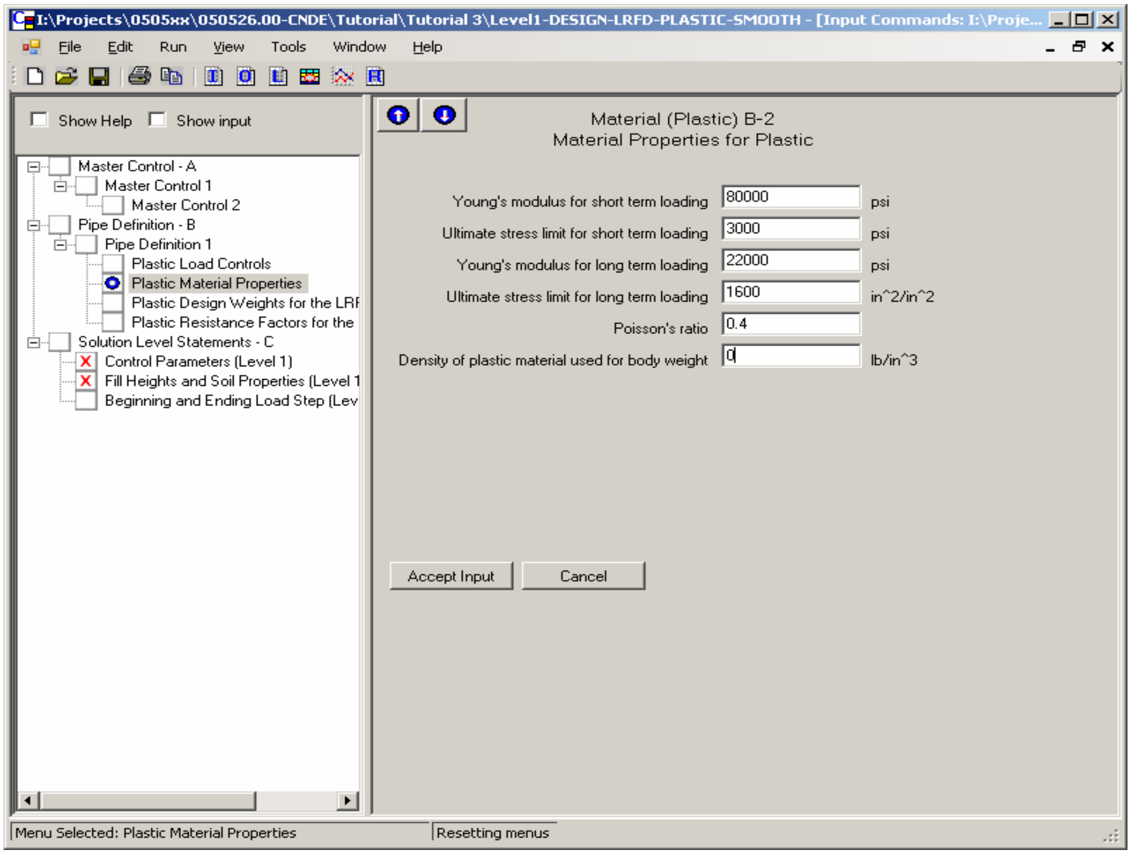


Figure 3-6 – Input Screen B-2

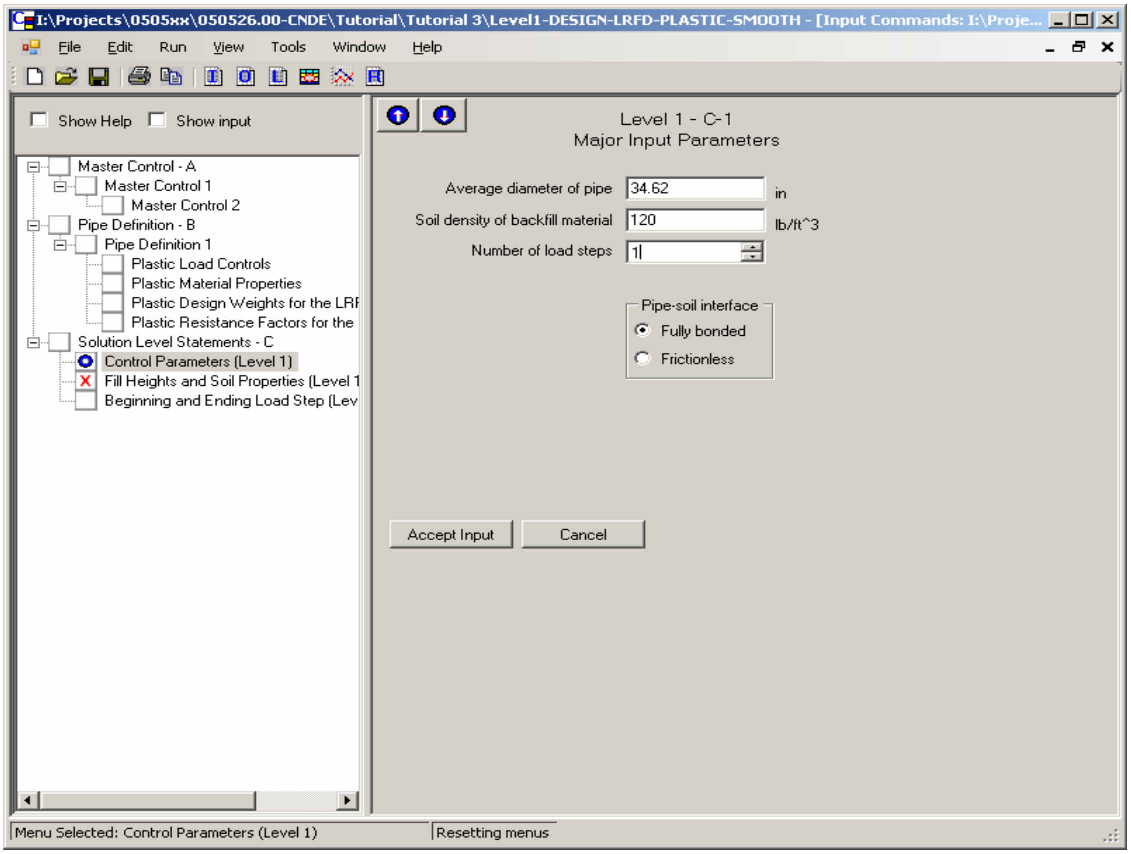


Figure 3-7 – Input Screen C-1

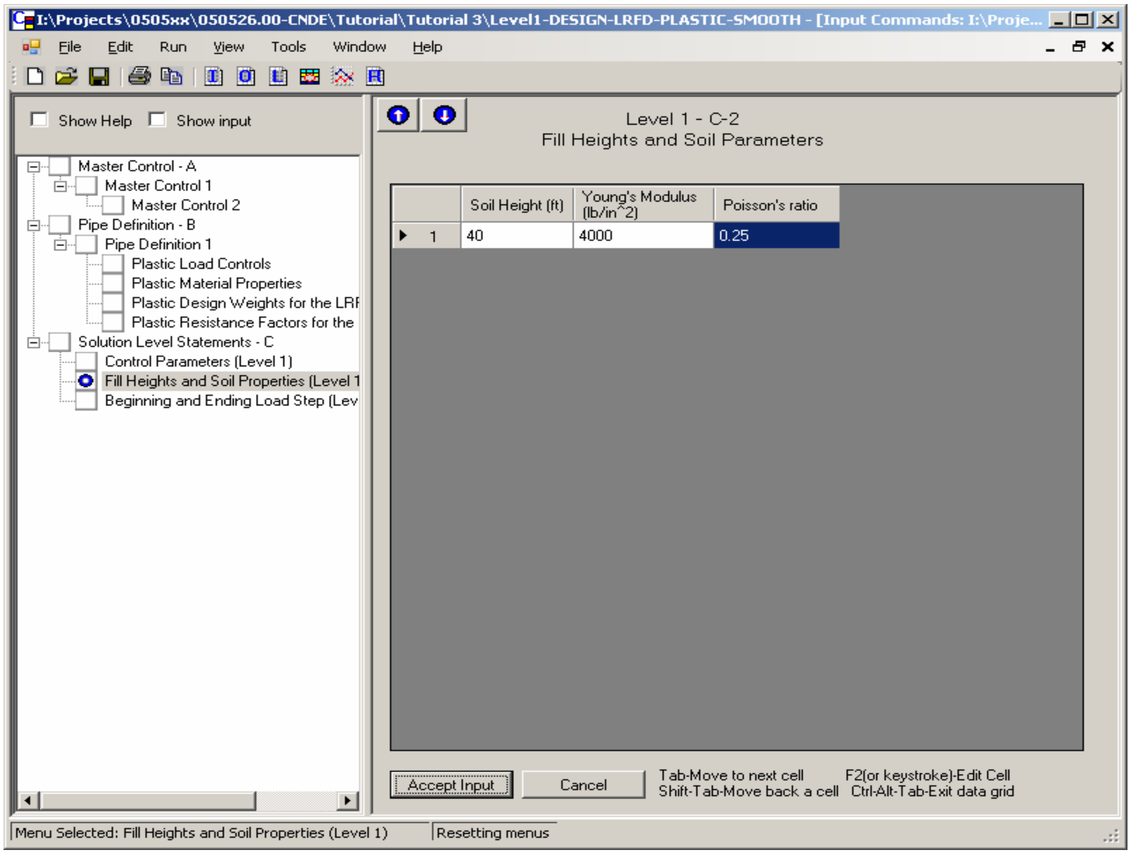


Figure 3-8 – Input Screen C-2

Under LRFD load factor in Figure 3-9 enter the combined value of the load factor and load modifier, i.e. – 1.95 * 1.05 = 2.05

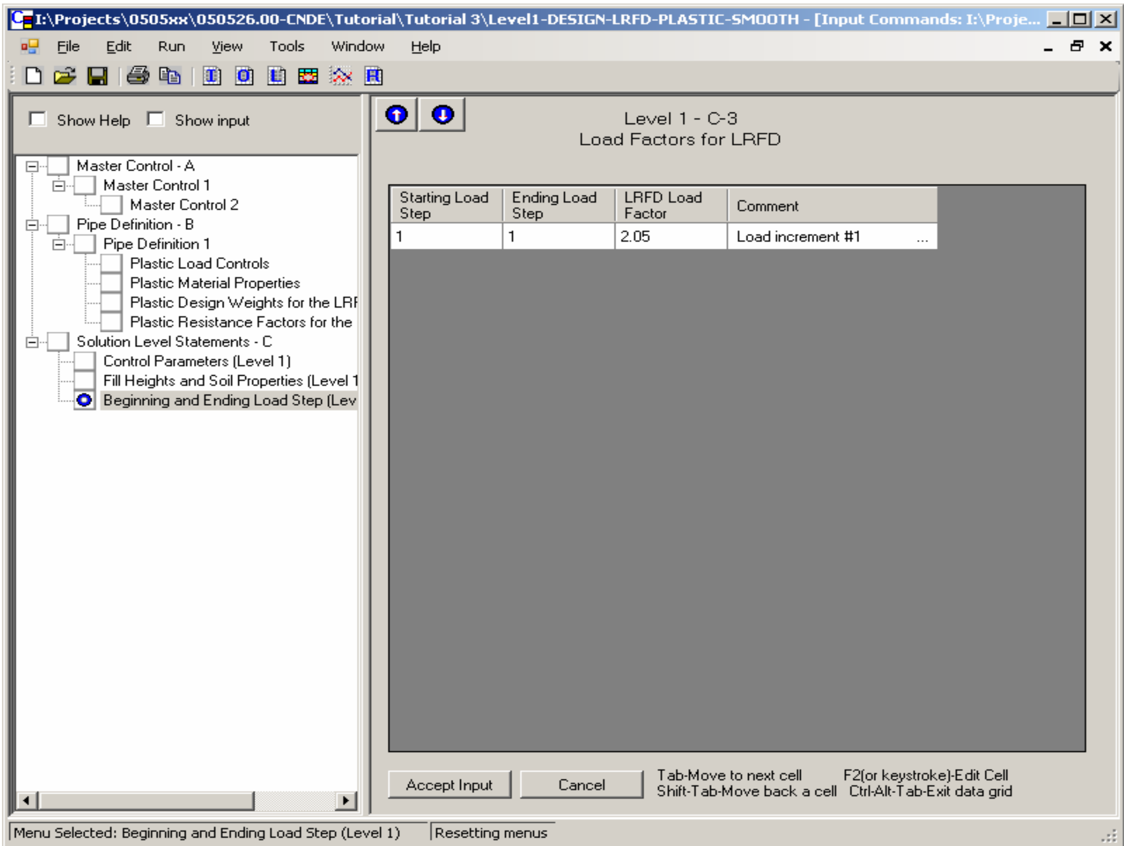


Figure 3-9 – Input Screen C-3

When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow the user to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

3.3 Reviewing and Interpreting the Output

Now proceed to check the output file. The user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data, the design solution (Figure 3-10), or the design assessment (Figure 3-11) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 3-11 shows the final assessment printed at the end of the file.

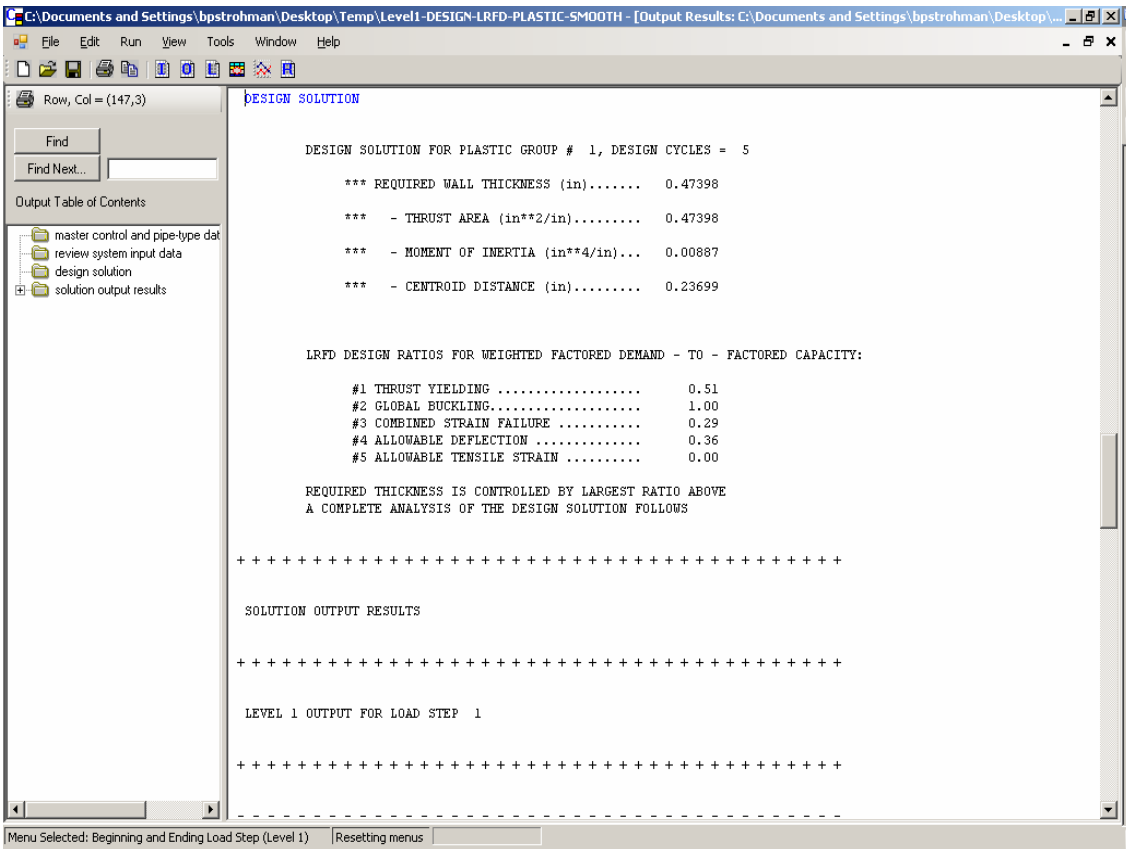


Figure 3-10 – Design Solution

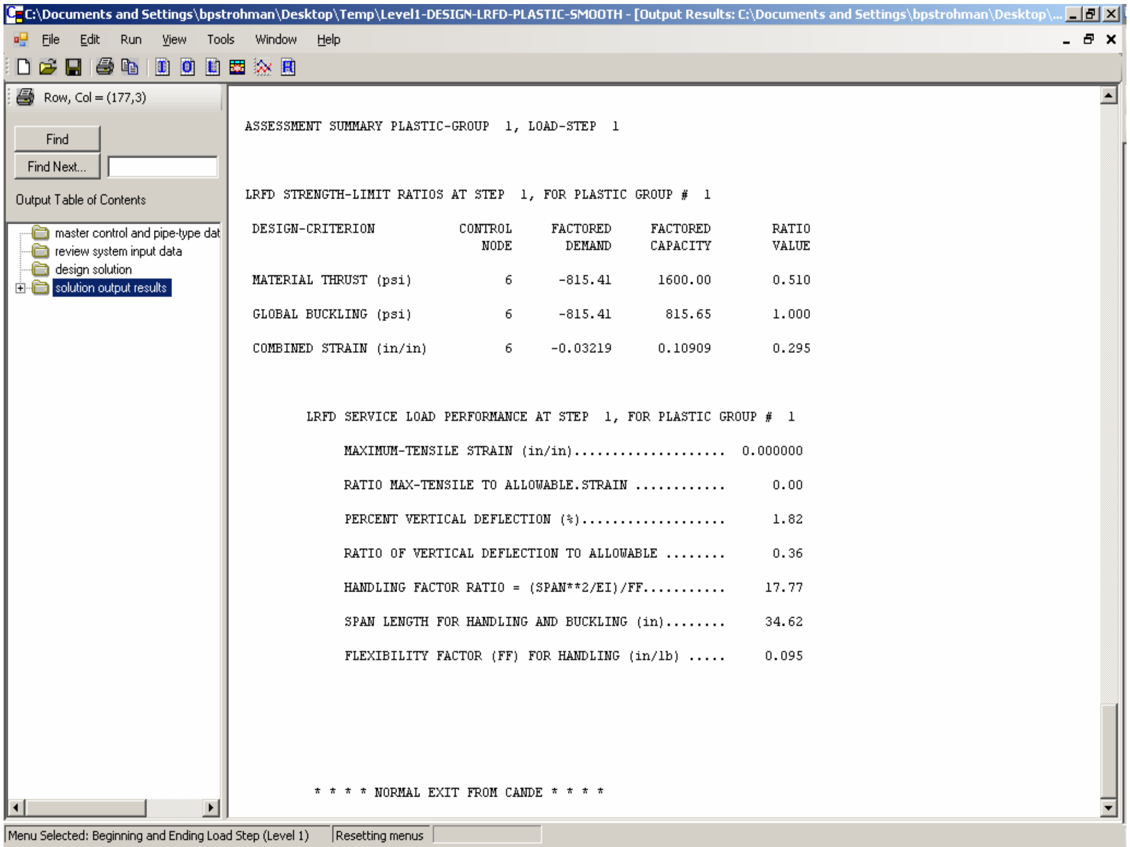


Figure 3-11 – Design Assessment Summary – Load Step 1

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 4

May 2008

This page intentionally left blank

4. CANDE TEST PROBLEM 4

4.1 Problem Definition

Design a 60 in. inside diameter corrugated aluminum pipe with 30 ft of fill over the top of the pipe using Working Stress (service) design. The problem is shown schematically in Figure 4-1. The design will be with Level 2, using an automated finite element pipe mesh for a trench installation having no interface elements. The desired result is the corrugation size and thickness.

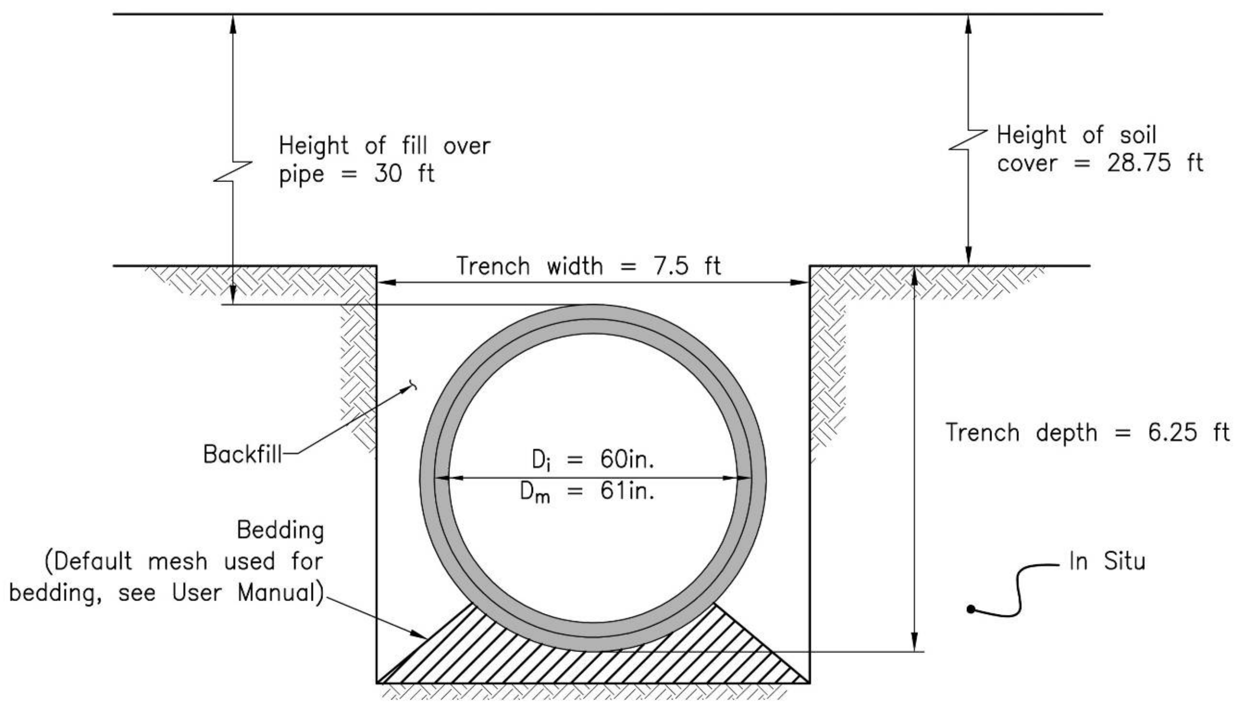


Figure 4-1 Details of Problem 4

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Design

Method of analysis/design - Service

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Pipe mesh

Soil mesh pattern - Trench

Interface elements (pipe only) - None

Pipe material type - Aluminum

Soil parameters - Canned Isotropic-Linear Elastic soil models for each of the following materials:

In situ soil - Hard clay – Young's modulus = 6,000 psi, Poisson's ratio = 0.35

Bedding soil - Lightly compacted sand – Young's modulus = 2,600 psi, Poisson's ratio = 0.19

Backfill soil - Lightly compacted sand – Young's modulus = 2,600 psi, Poisson's ratio = 0.19

Overfill soil - Lightly compacted silt – Young's modulus = 800 psi, Poisson's ratio = 0.23

Soil density - 120 lb/ft^3 for all soils

Material behavior - Bilinear stress/strain

Analysis mode - Small deformation

Average diameter of pipe - Set pipe diameter to centroid of pipe wall. Assume design will be met with a 1 in. deep corrugation, therefore set to nominal diameter + 1 in. = 61 in.

Ratio of horizontal to vertical diameter - 1.0

Height of soil cover - For trench installations CANDE calculates height of soil cover from the top of the trench (See *User Manual*, Chapter 5, C-2). To achieve a total height of fill of 30 ft over the top of the pipe, set the height of soil cover to 28.75 ft (See Figure 4-1).

Density of soil above truncated mesh - 120 lb/ft^3.

Number of construction steps - See *User Manual*, Chapter 5, C-3 – The default mesh provides 5 construction steps to the top of the mesh, which provides approximately 1.5 pipe diameters of fill over the pipe. For the remaining depth of fill, use 5 steps, making a total of 10 construction steps.

Trench depth - Depth from bottom of pipe to top of trench (See *User Manual*, Chapter 5, C-4). Use maximum of 2.5 pipe diameters = 6.25 ft per Figure 4-1.

Trench width - 7.5 ft

4.2 Creating the CANDE Input Document

Figures 4-2 through 4-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 3, CANDE will display a screen indicating that an input path is initiated - click “Finish.” The user is prompted for a filename and directory.

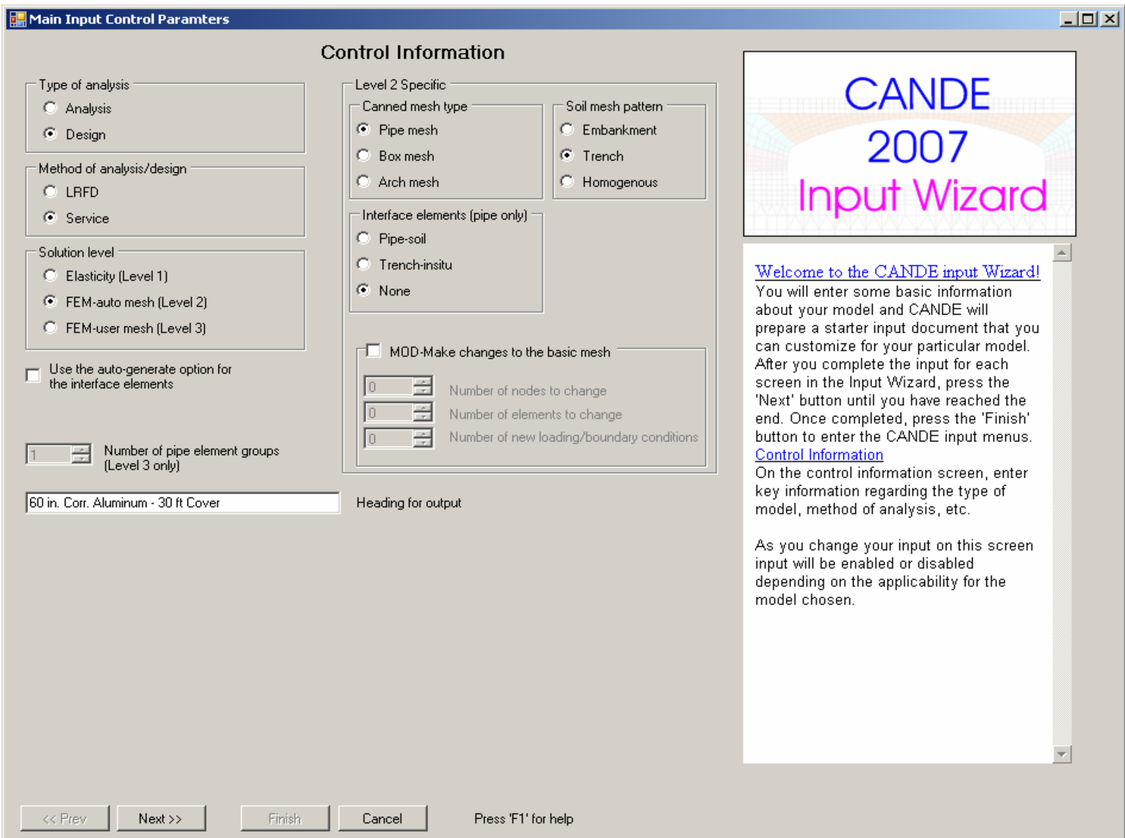


Figure 4-2 – Input Wizard, Screen 1

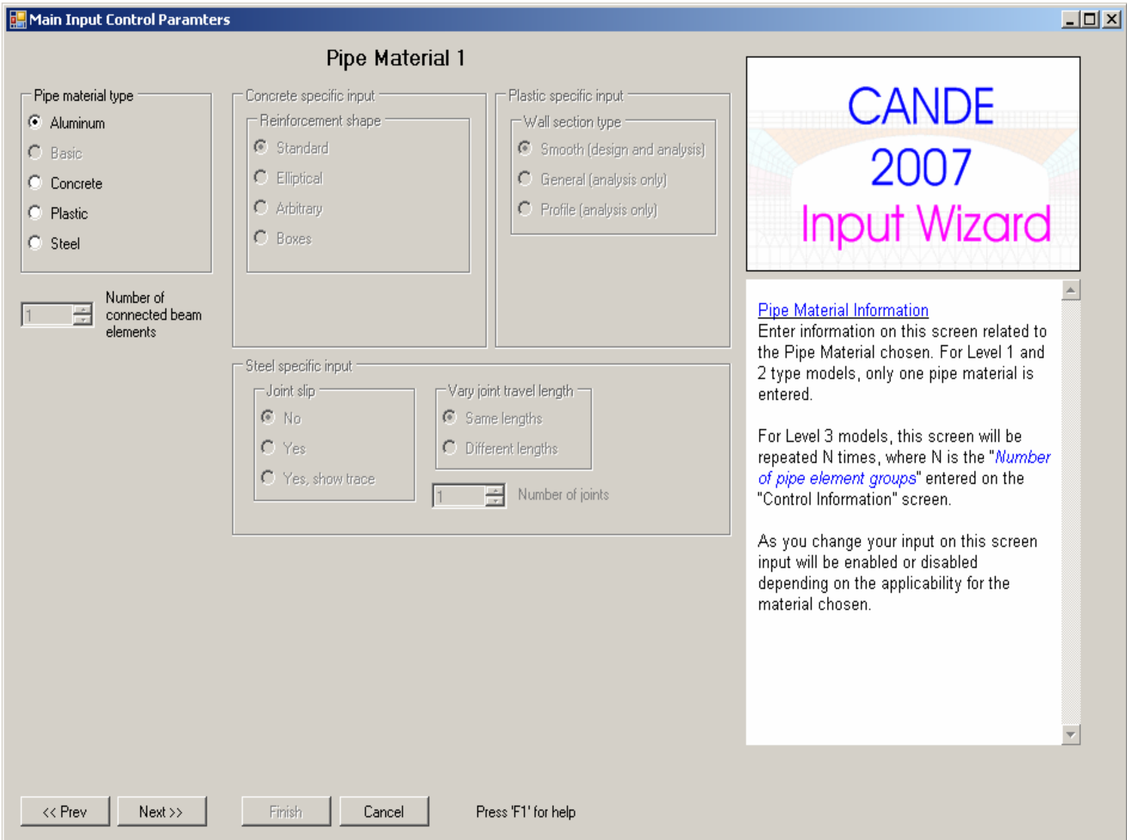


Figure 4-3 – Input Wizard, Screen 2

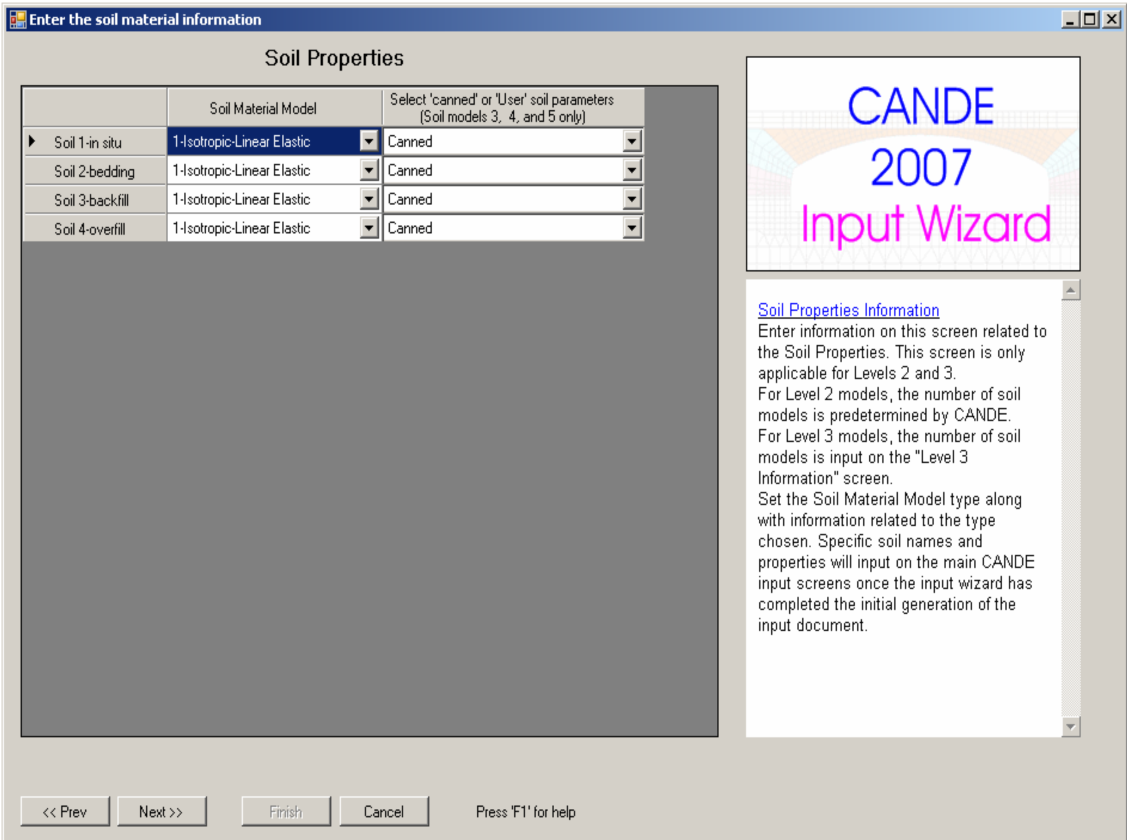


Figure 4-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 4-5. Enter an appropriate heading for output and click "Accept Input."

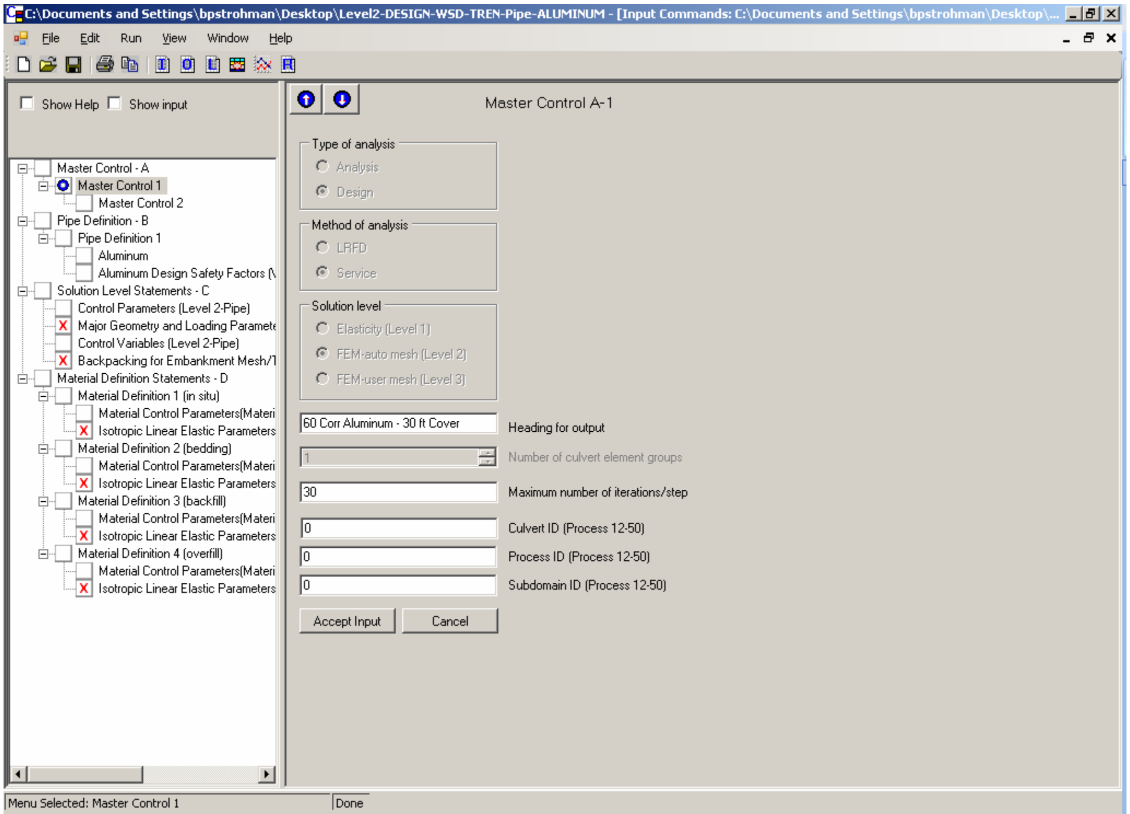


Figure 4-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 4-5 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 4-6 through 4-10 show the completed input for the screens requiring data for the tutorial, except that only one material definition screen is shown. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

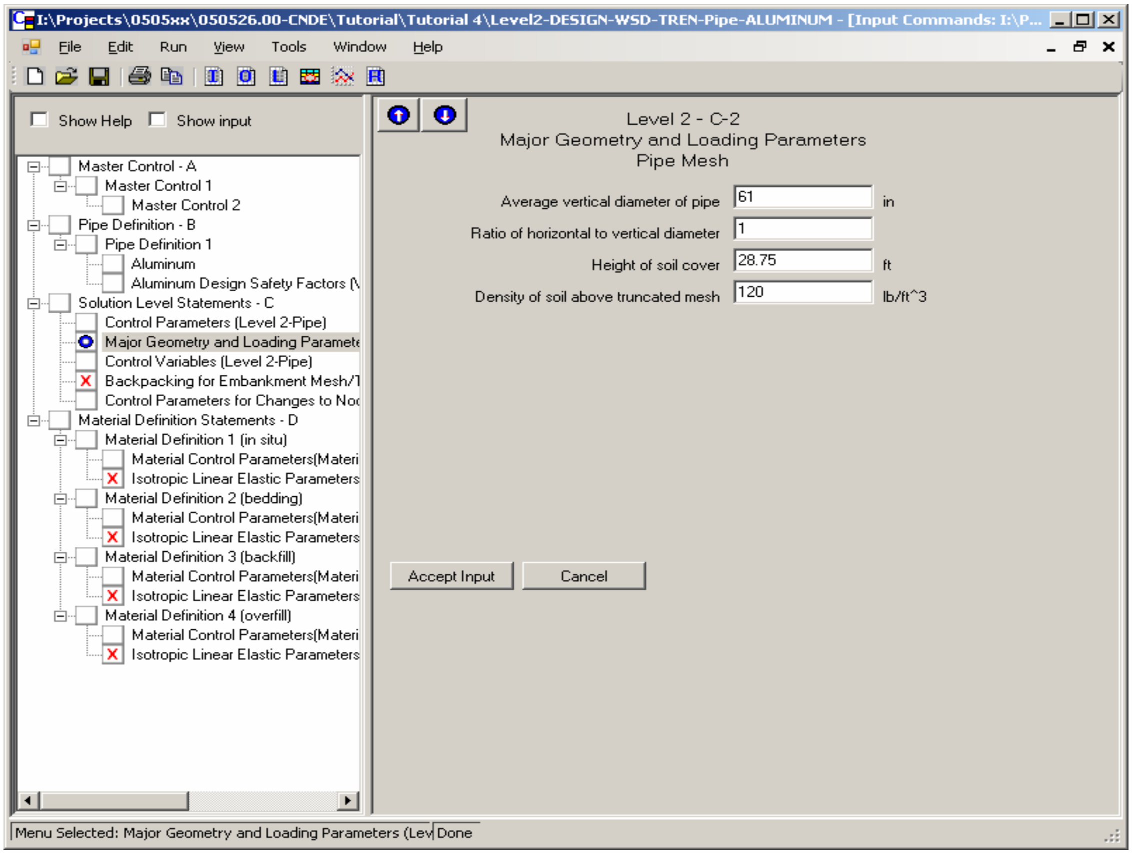


Figure 4-6 – Input Screen C-2

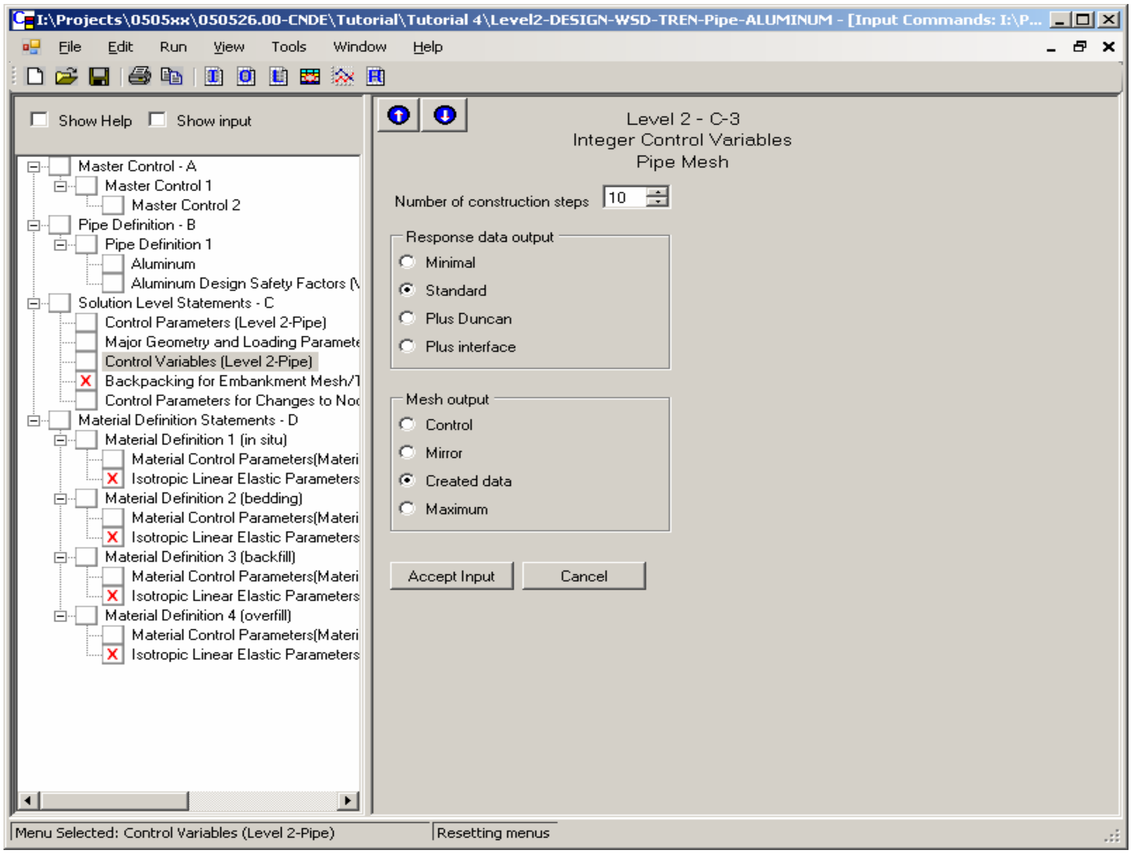


Figure 4-7 – Input Screen C-3

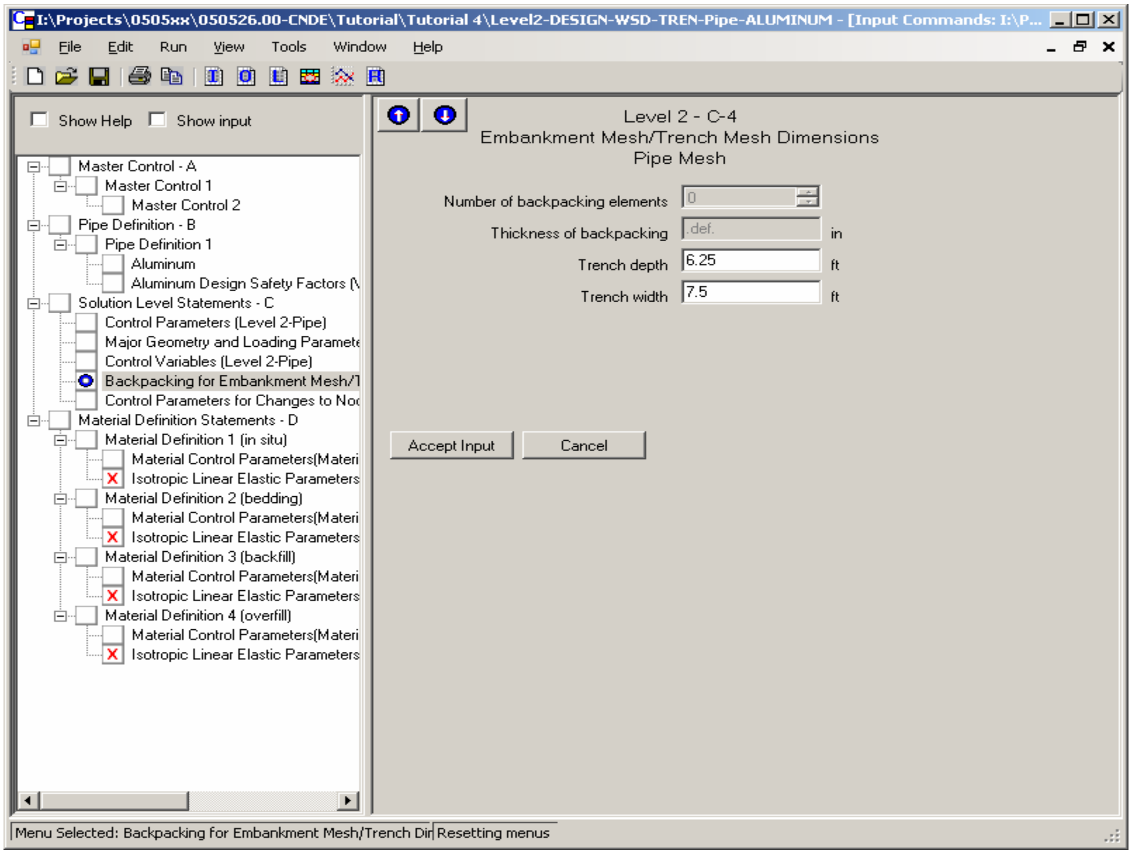


Figure 4-8 – Input Screen C-4

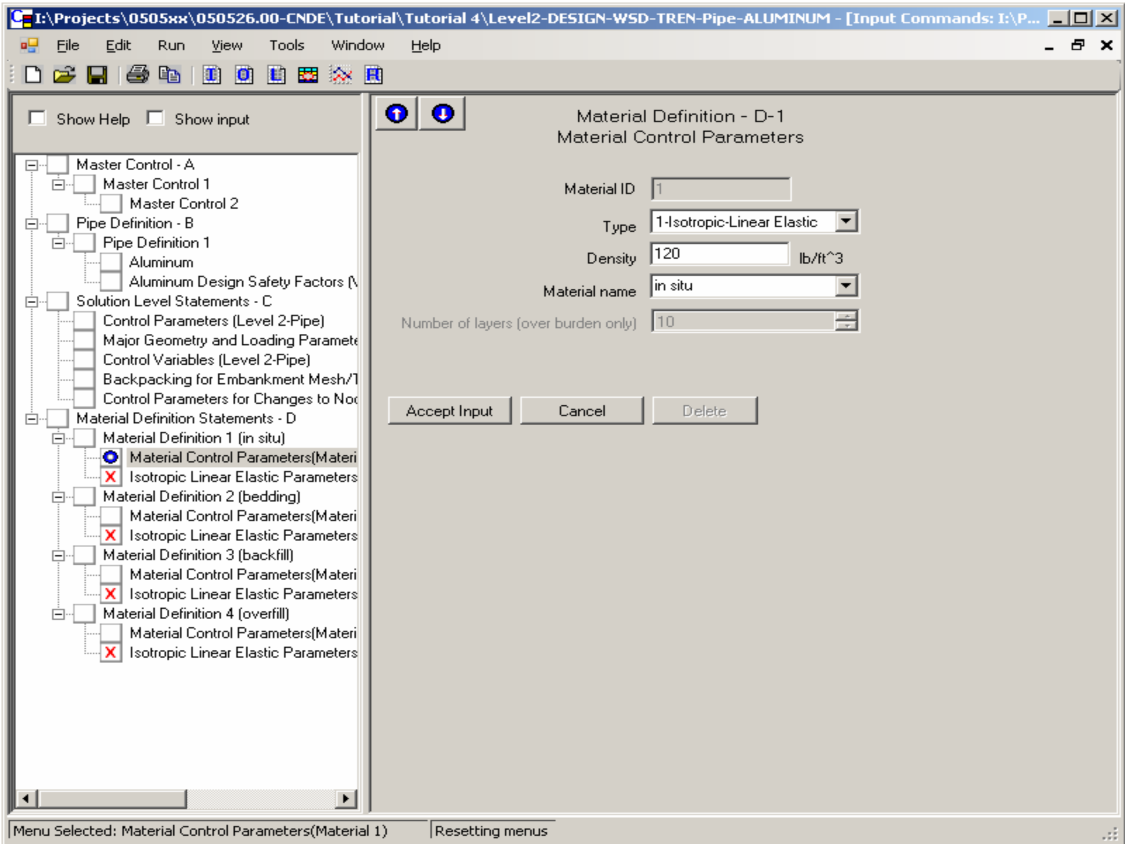


Figure 4-9 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

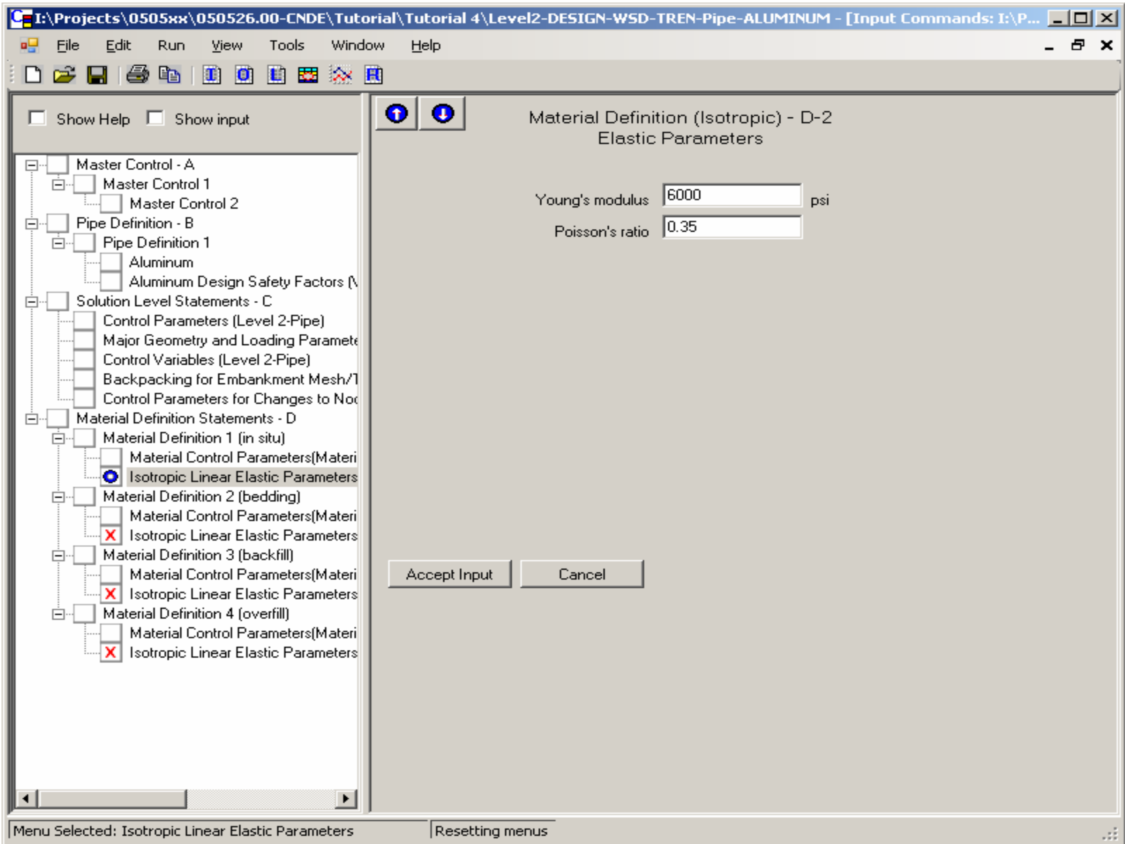


Figure 4-10 – Input Screen D-2 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

When all input is complete and saved, click "Run" and "CANDE-2007" on the main toolbar to execute the program. This will open the "Running CANDE" window which will allow you to monitor the run as it progresses. When the program is complete, a check box "CANDE Analysis Complete" window will appear. Click "OK" and check if the "Running CANDE" window states "Normal Exit From CANDE." If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the "Close" button in the "Running CANDE" window to return to the user interface.

4.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the “View” menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The “Plotting Parameters” button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click “View” and then “Mesh Plot” to open the mesh plot. Open the plotting parameters menu and click the check box “Show constr. increment numbers.” Click “OK,” set Load Increment to 5 to show the entire mesh (the remaining load is placed above the mesh – see *User Manual)* and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 4-11.

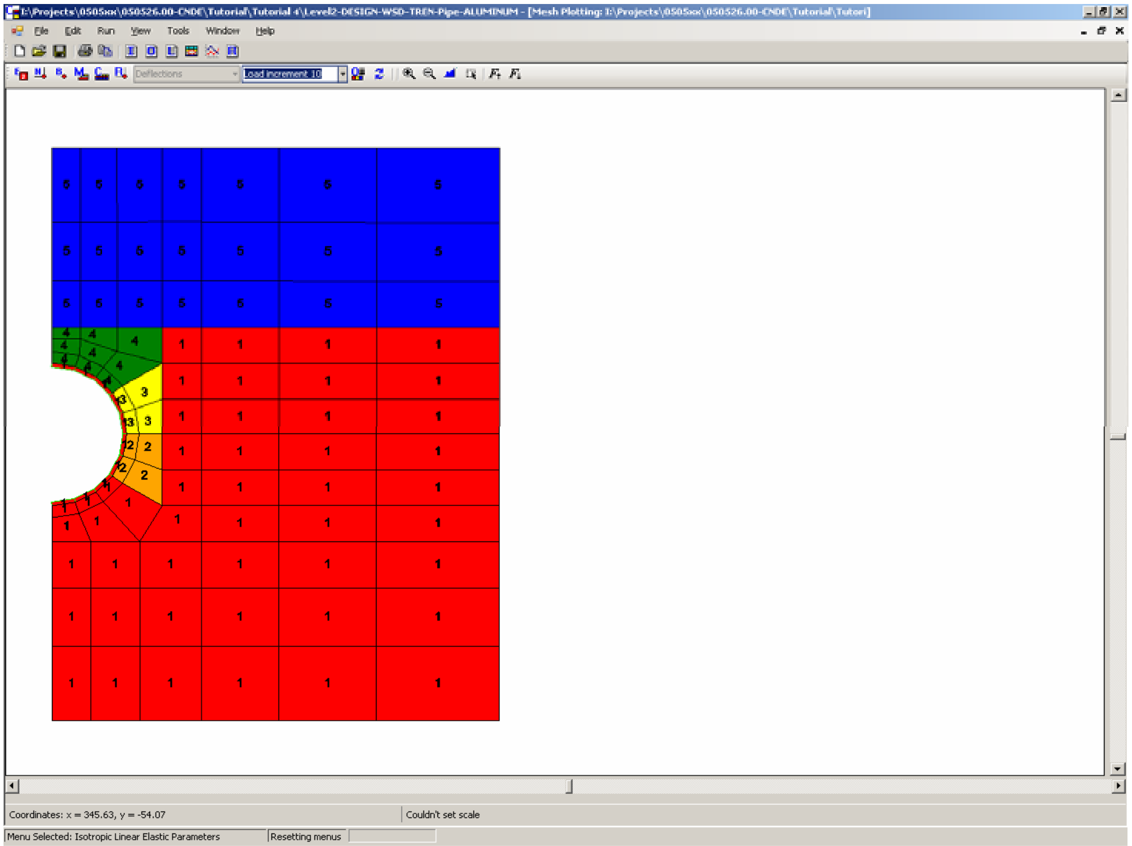


Figure 4-11 – Mesh Plot for Load Steps 1 to 5

To view deflections, open the plotting parameters window and set the deflection magnification factor to 10, click the "Ignore deflections/by load incr." check box and set the Load Increment to 1 (see Figure 4-12).

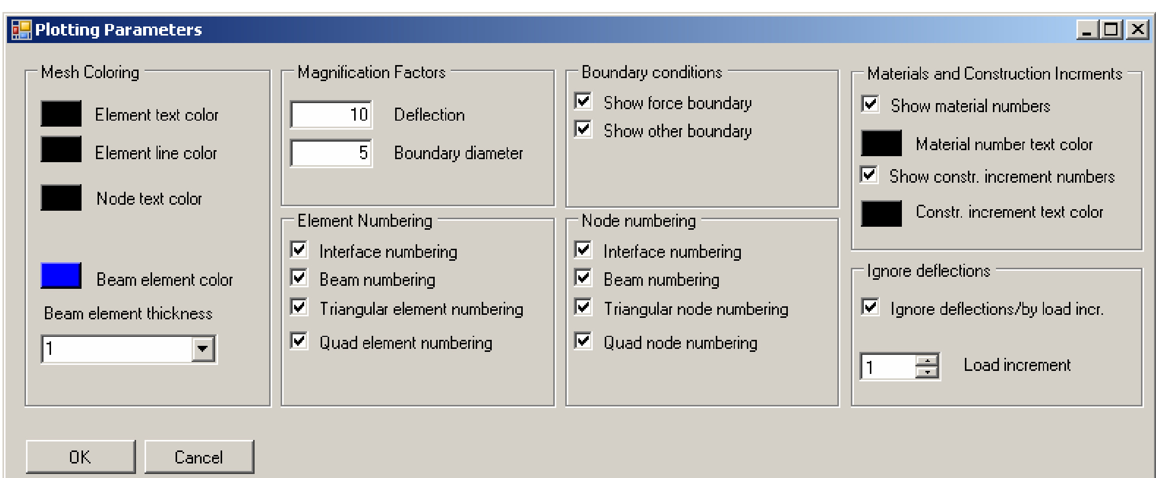


Figure 4-12 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Deflections. Note that if the Load Increment is set to 1, the deflections due to Load Increment 1 are shown, but when the Load Increment is set to 2, the Increment 1 deflections are ignored. Set the increment to 10 and the screen should look like Figure 4-13. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

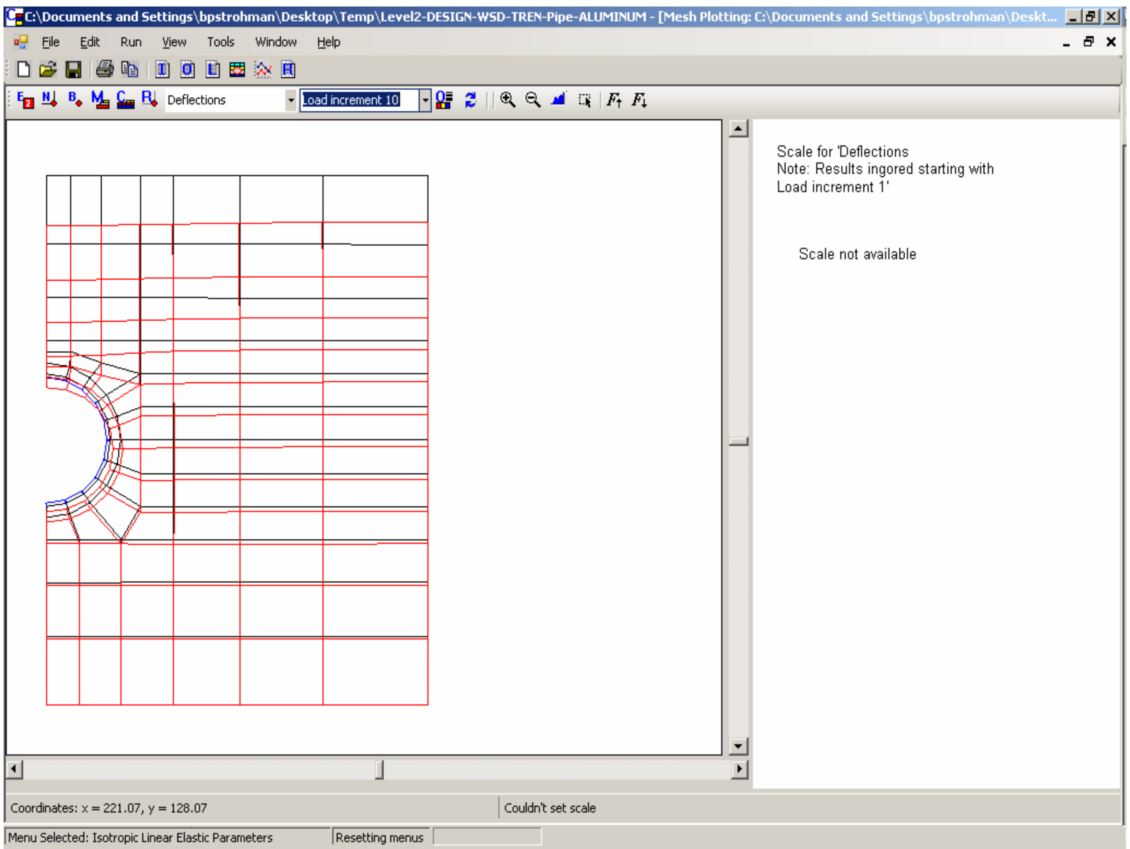


Figure 4-13 – Deflections, Excluding Those Due to Self Weight of In Situ Soil

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 4-14 compares the bending moments after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 4, 6, and 10, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Bending moment” and the screen should appear as Figure 4-14.

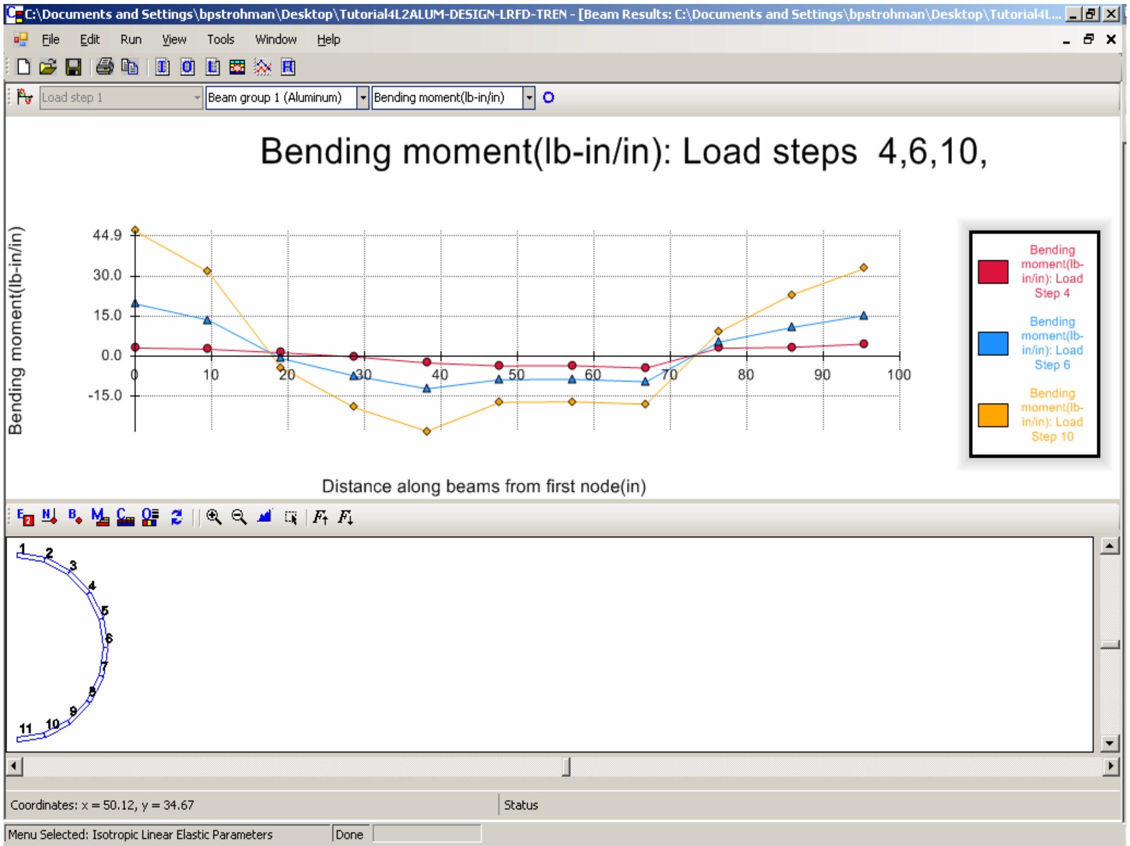


Figure 4-14 – Bending Moments for Load Steps 4, 6, and 10

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data, the design solution (Figure 4-15), or the design assessment (Figure 4-16) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 4-16 shows the final assessment printed at the end of the file.

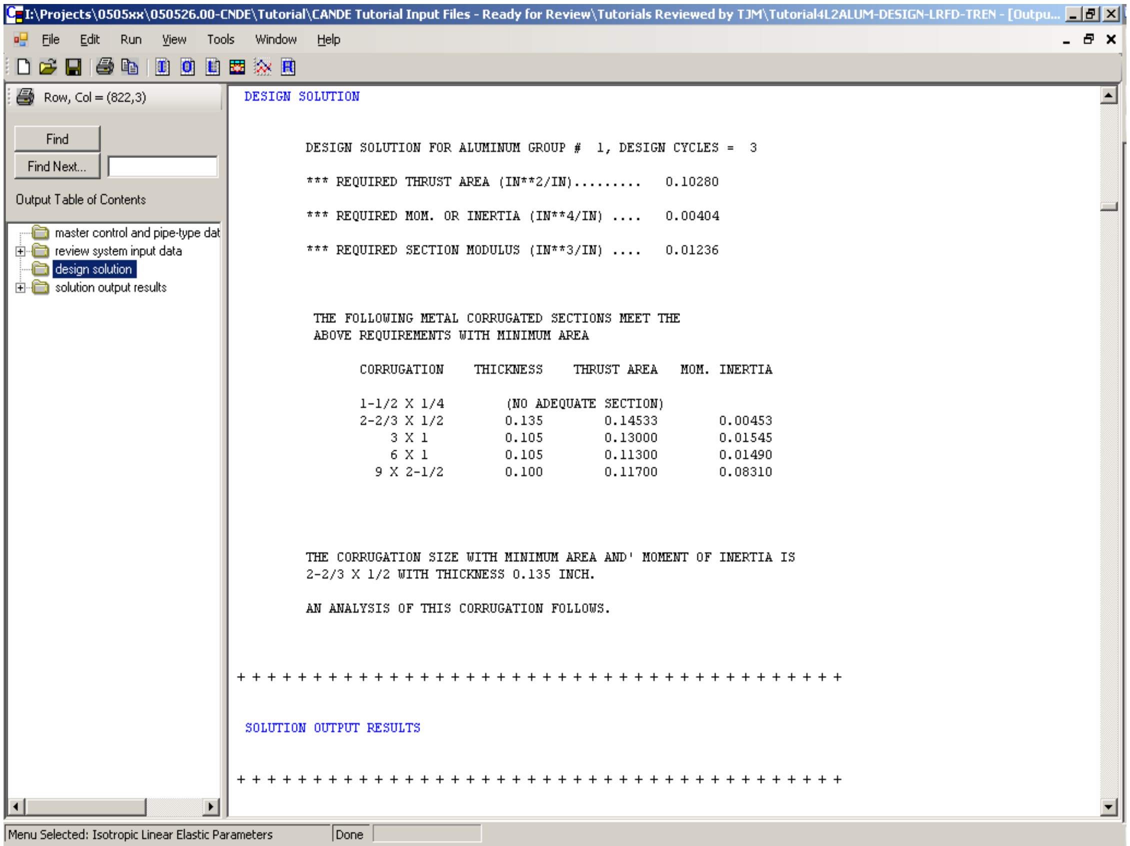


Figure 4-15 – Design Solution

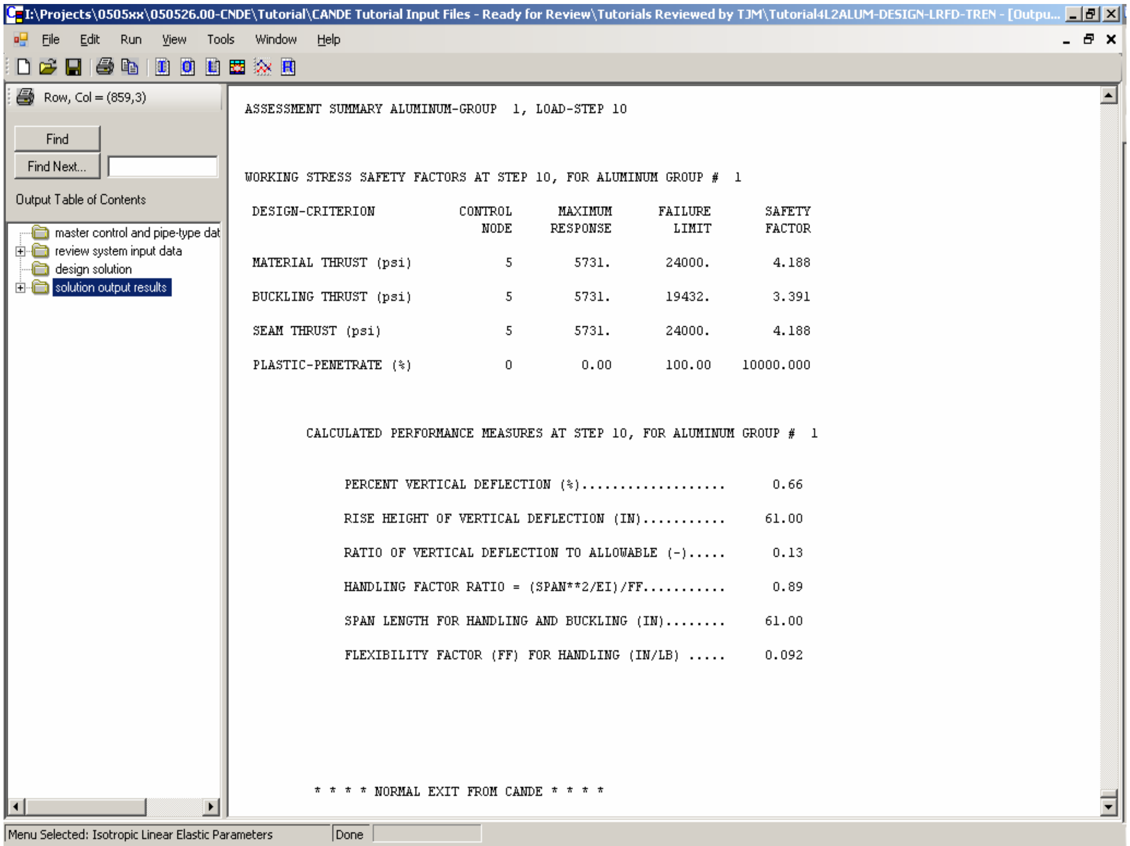


Figure 4-16 – Design Assessment Summary – Load Step 10

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 4-17 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 4-18. The three tabs shown in Figures 4-17 and 4-18 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.
- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.
- Report Preview – displays the desired report.

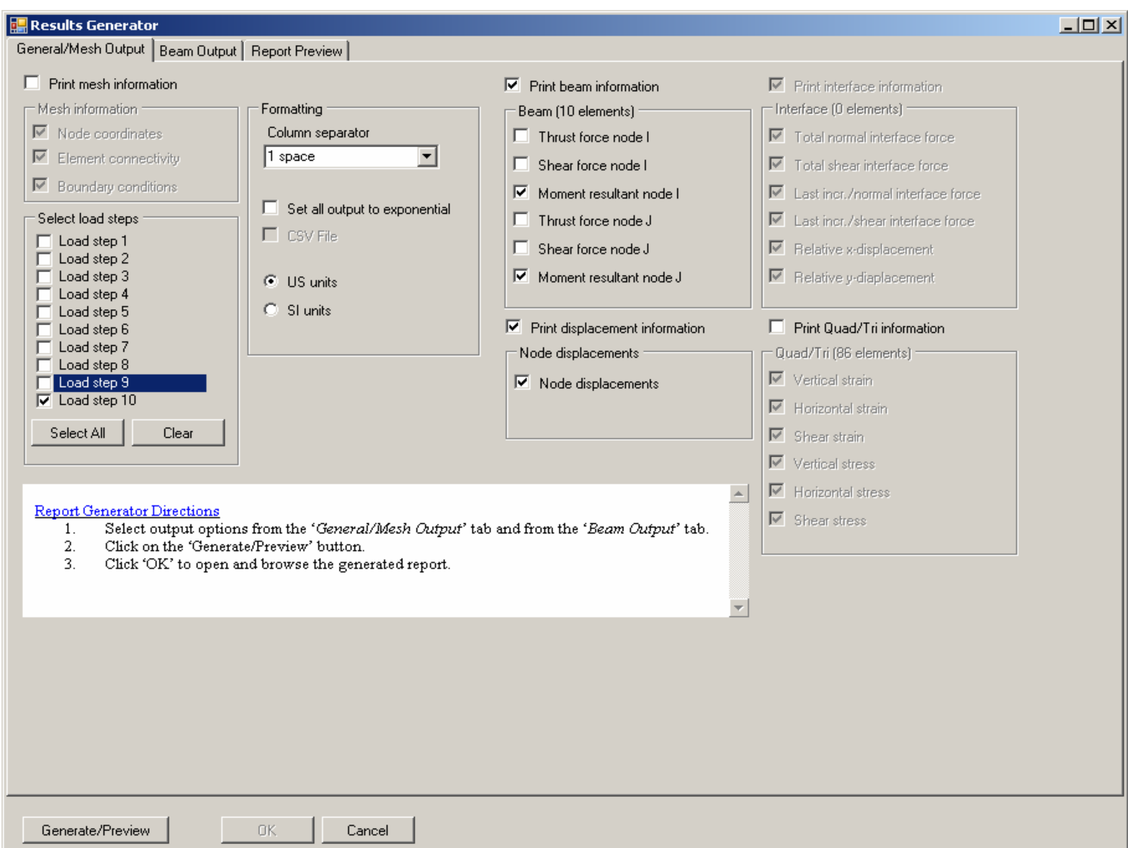


Figure 4-17 – Results Generator Input Screen – Load Step 10 Moments and Deflections

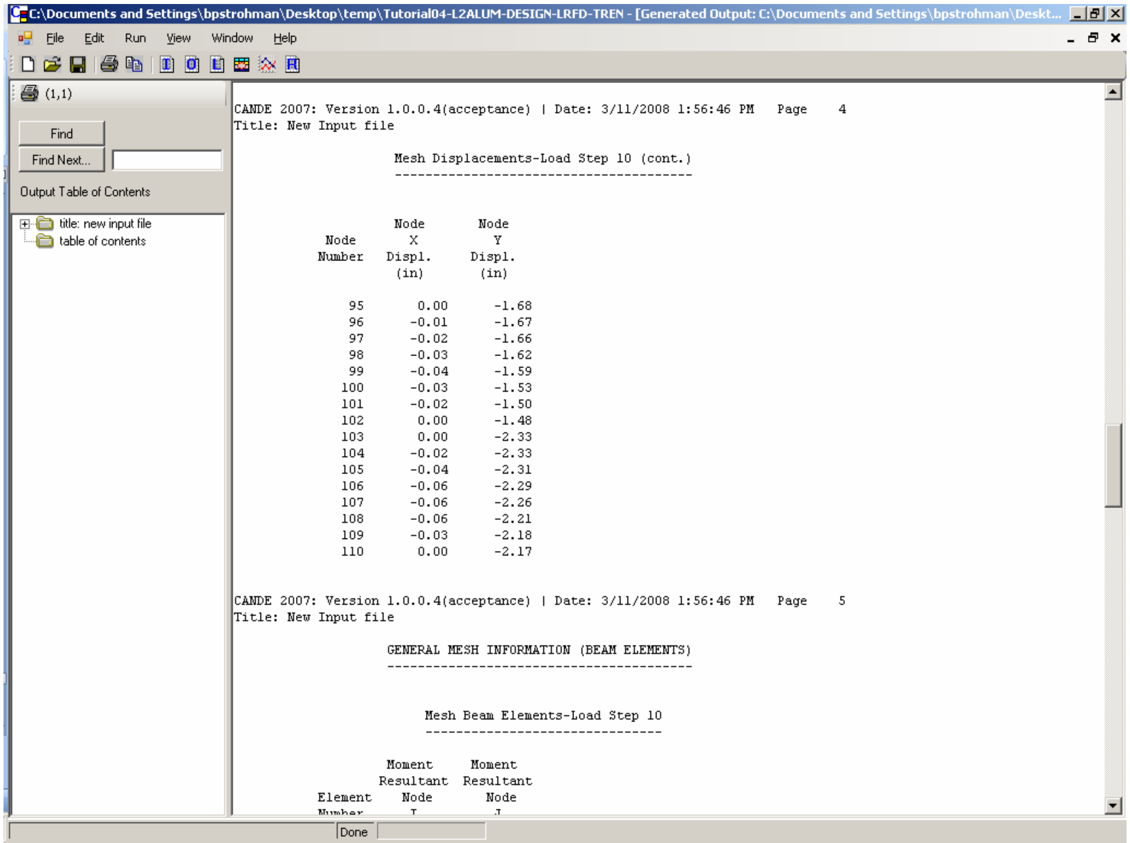


Figure 4-18 – Results Generator – Load Step 10 Deflections

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 5

May 2008

5. CANDE TEST PROBLEM 5

5.1 Problem Definition

Analyze a 36 in. outside diameter smooth wall HDPE plastic pipe with 40 ft of fill over the top of the pipe using Working Stress (service) analysis. The problem is shown schematically in Figure 5-1. The analysis will be with Level 2, using an automated finite element pipe mesh for an embankment installation having no interface elements.

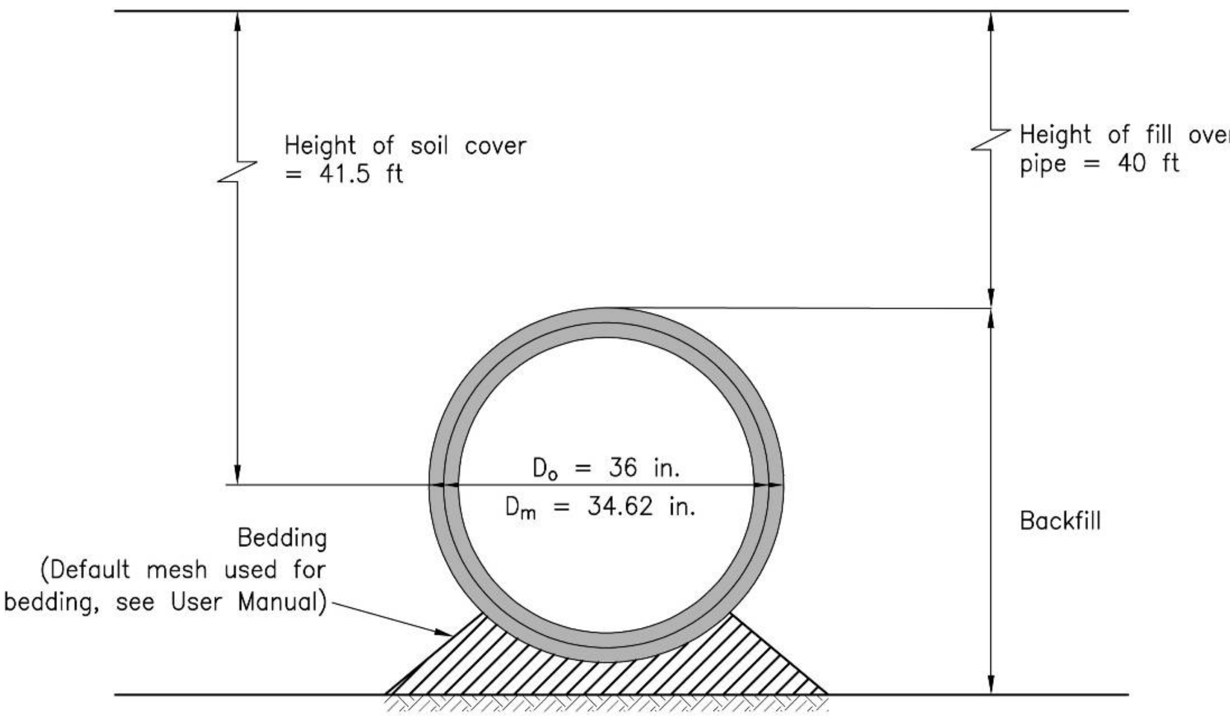


Figure 5-1 Details of Problem 5

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - Service

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Pipe mesh

Soil mesh pattern - Embankment

Interface elements (pipe only) - None

Pipe material type - Plastic

Wall section type - Smooth

Soil parameters - Canned Isotropic-Linear Elastic soil models for each of the following materials:

In situ soil - Hard clay – Young's modulus = 6,000 psi, Poisson's ratio = 0.35

Bedding soil - Lightly compacted sand – Young's Modulus = 2,600 psi, Poisson's ratio = 0.19

Backfill soil - Lightly compacted sand – Young's modulus = 2,600 psi, Poisson's ratio = 0.19

Backpack soil - Lightly compacted silt – Young's modulus = 800 psi, Poisson's ratio = 0.23

Soil density - 120 lb/ft^3 for all soils

Load duration - Short term

Analysis mode - Large deformation/buckling

Young's modulus for short term loading - 80,000 lb/in.^{2}

Ultimate stress limit for short term loading - 1,600 lb/in.^{2}

Poisson's ratio - 0.4

Density of plastic material used for body weight - 0.034 lb/in.^{3}

Total height of wall cross section - 1.38 in. Assume pipe is DR 26.

Average diameter of pipe - Set pipe diameter to centroid of pipe wall. D_m = 34.62 in.

Ratio of horizontal to vertical diameter - 1.0

Height of soil cover - For embankment installations CANDE calculates height of soil cover from the springline of the pipe (see *User Manual*, Chapter 5, C-2). To achieve a total height of fill of 40 ft over the top of the pipe, set the depth to 40 ft + mean pipe radius = 41.5 ft (See Figure 5-1).

Density of soil above truncated mesh - 120 lb/ft^3

Number of construction steps - See *User Manual*, Chapter 5, C-3 – The default mesh provides 5 construction steps to the top of the mesh, which provides approximately 1.5 pipe diameters of fill over the pipe. For the remaining depth of fill, use 5 steps, making a total of 10 construction steps.

5.2 Creating the CANDE Input Document

Figures 5-2 through 5-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking "Next" on Screen 3, CANDE will display a screen indicating that an input path is initiated - click "Finish." The user is prompted for a filename and directory.

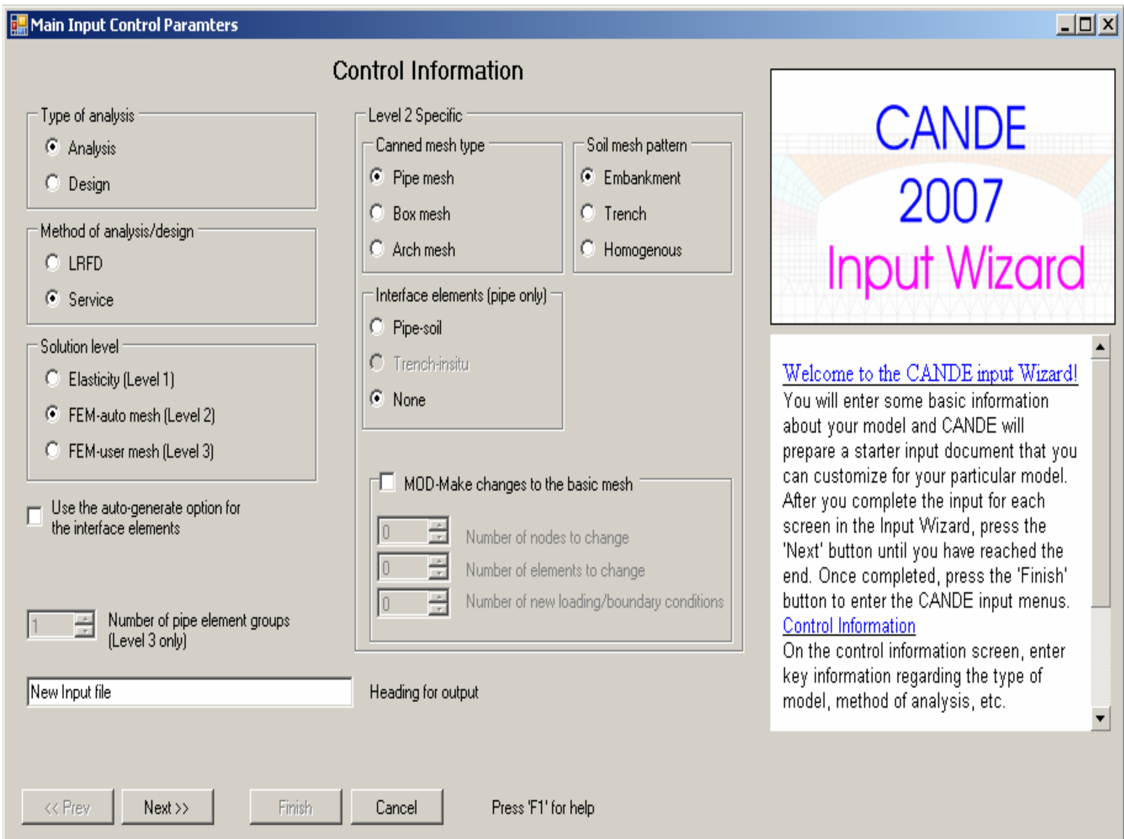


Figure 5-2 – Input Wizard, Screen 1

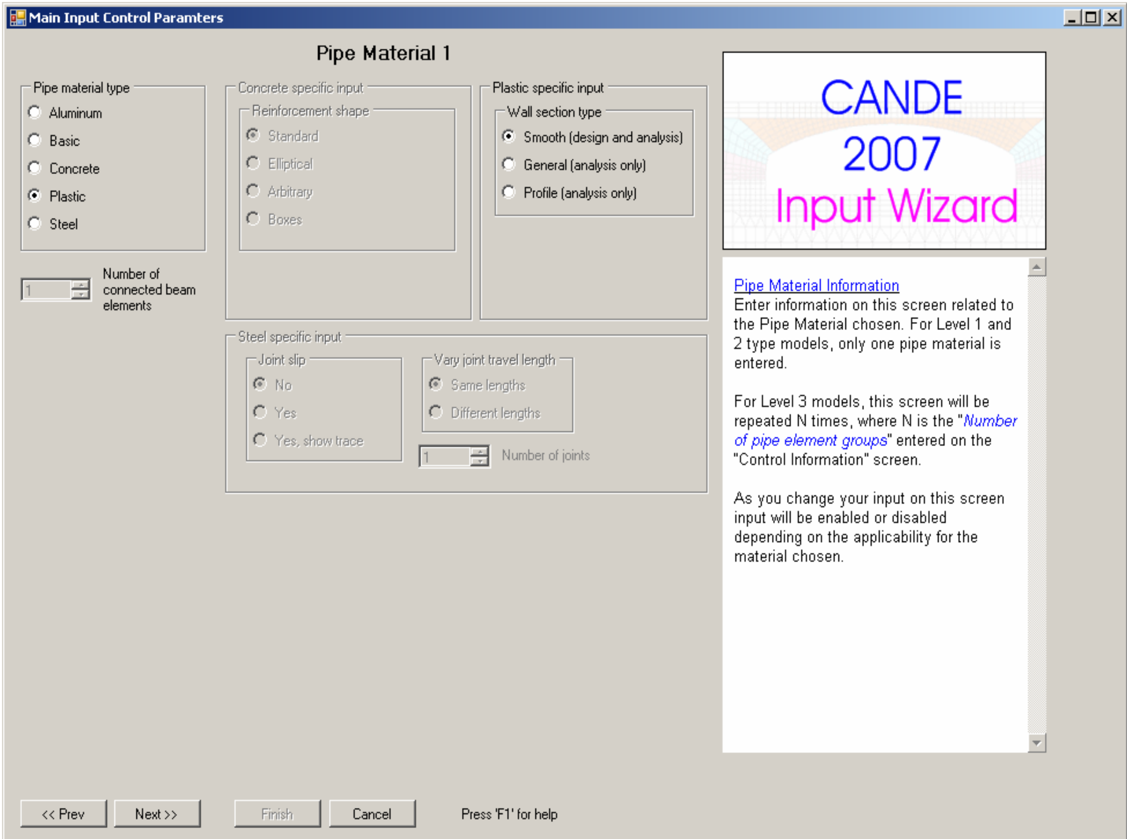


Figure 5-3 – Input Wizard, Screen 2

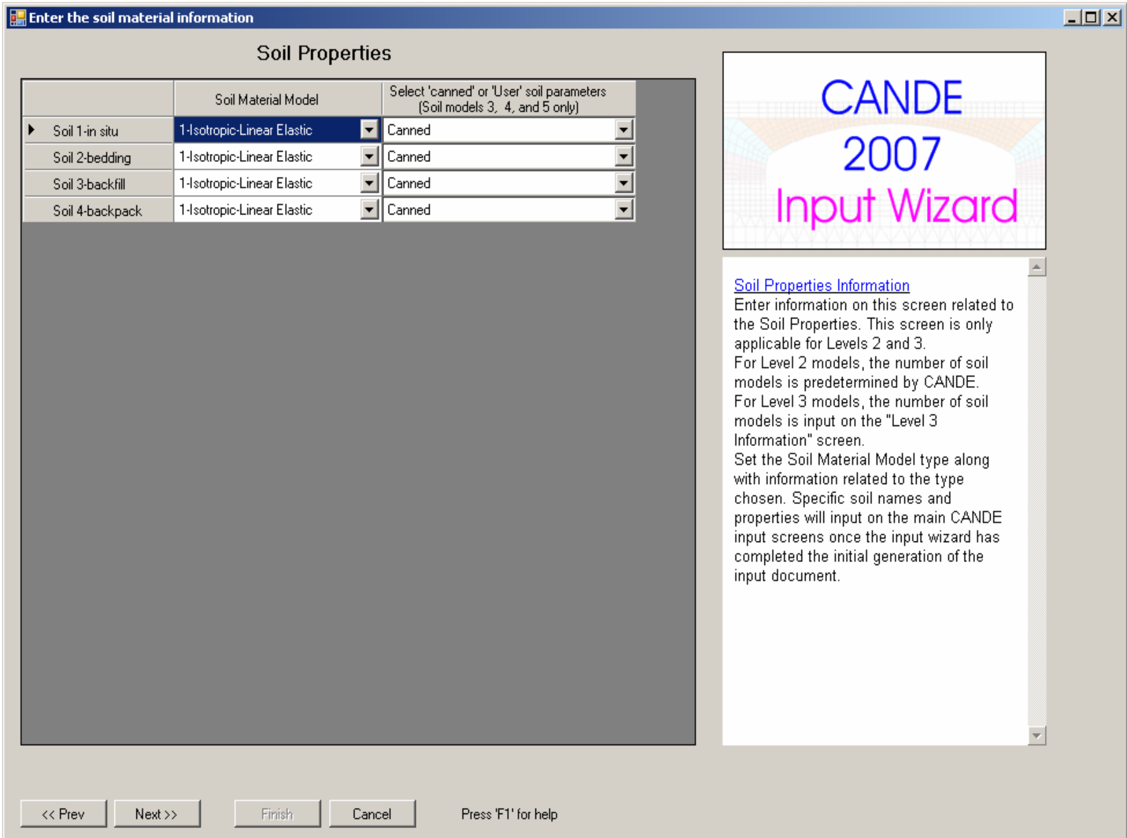


Figure 5-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 5-5. Enter an appropriate heading for output and click "Accept Input."

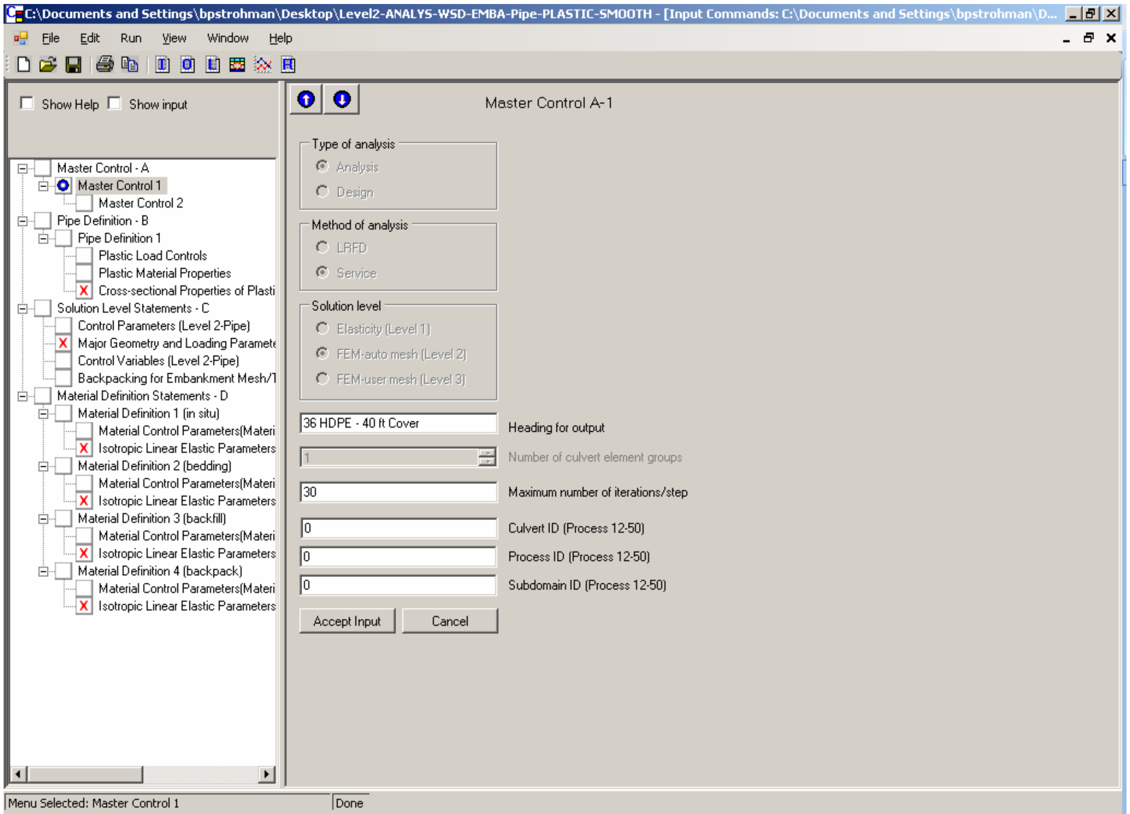


Figure 5-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 5-5 shows “X” for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 5-6 through 5-12 show the completed input for the screens requiring data for the tutorial, except that only one material definition screen is shown. Input the remaining definition screens. As input is modified, a ‘blue circle’ icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click “Accept Input” and the blue circle in the control panel will disappear indicating all fields have data.

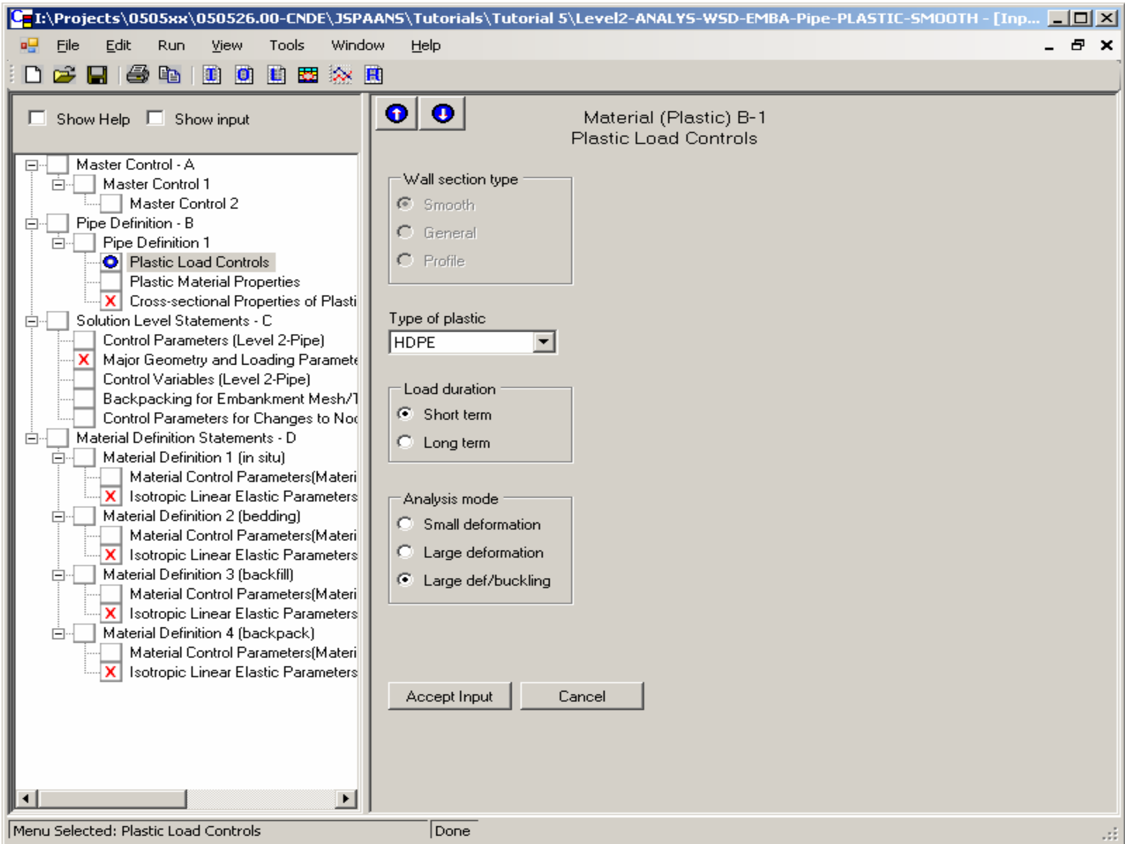


Figure 5-6 – Input Screen B-1

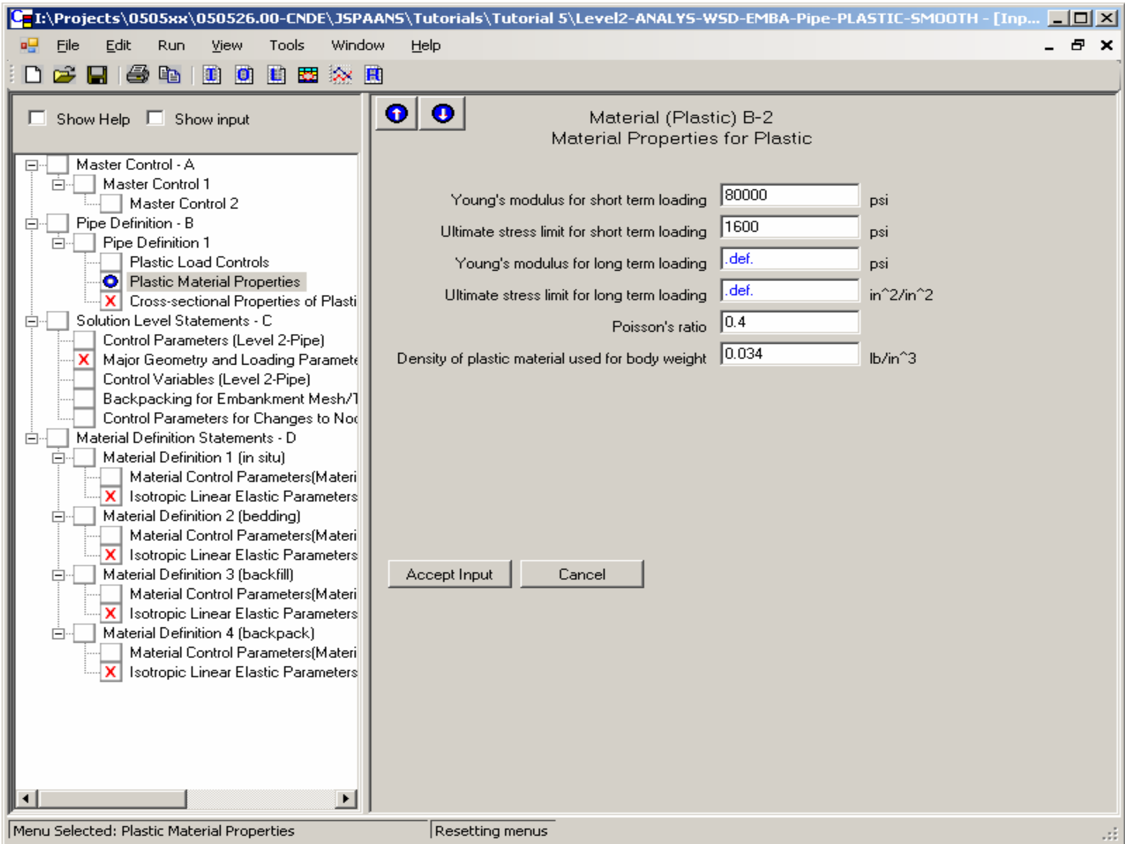


Figure 5-7 – Input Screen B-2

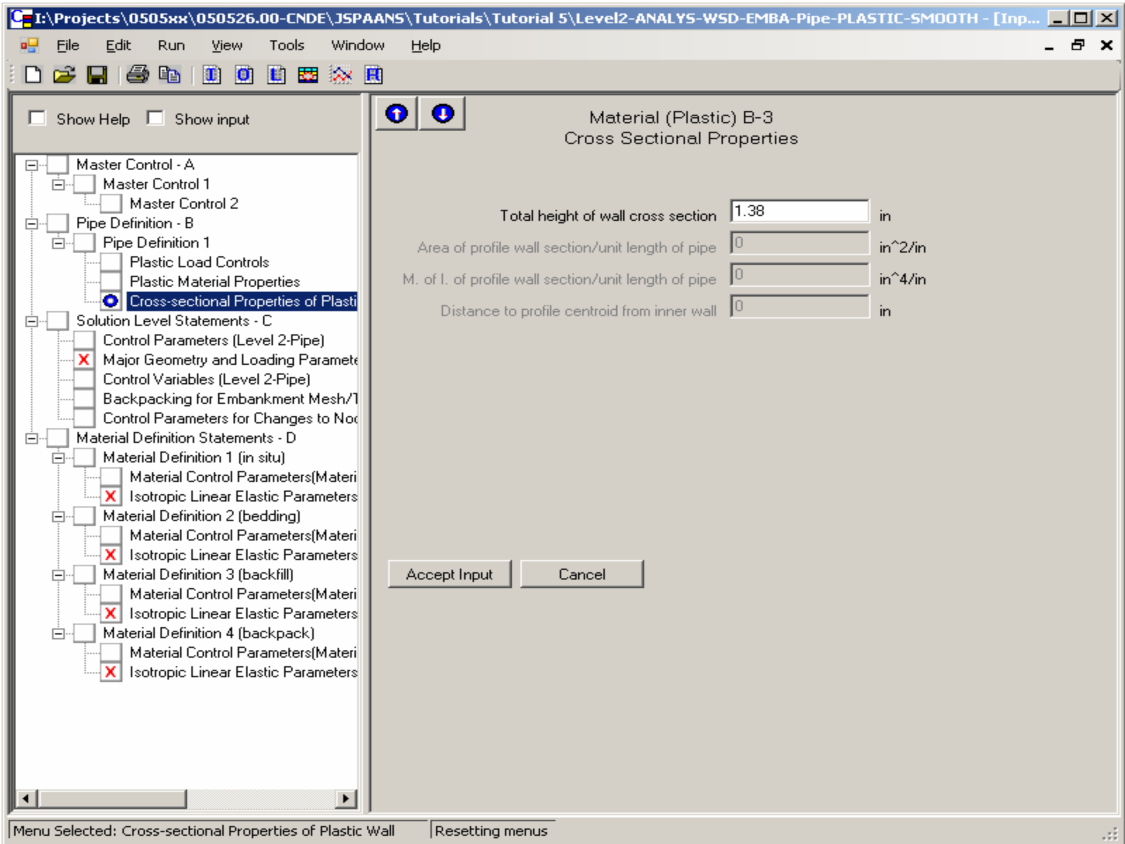


Figure 5-8 – Input Screen B-3

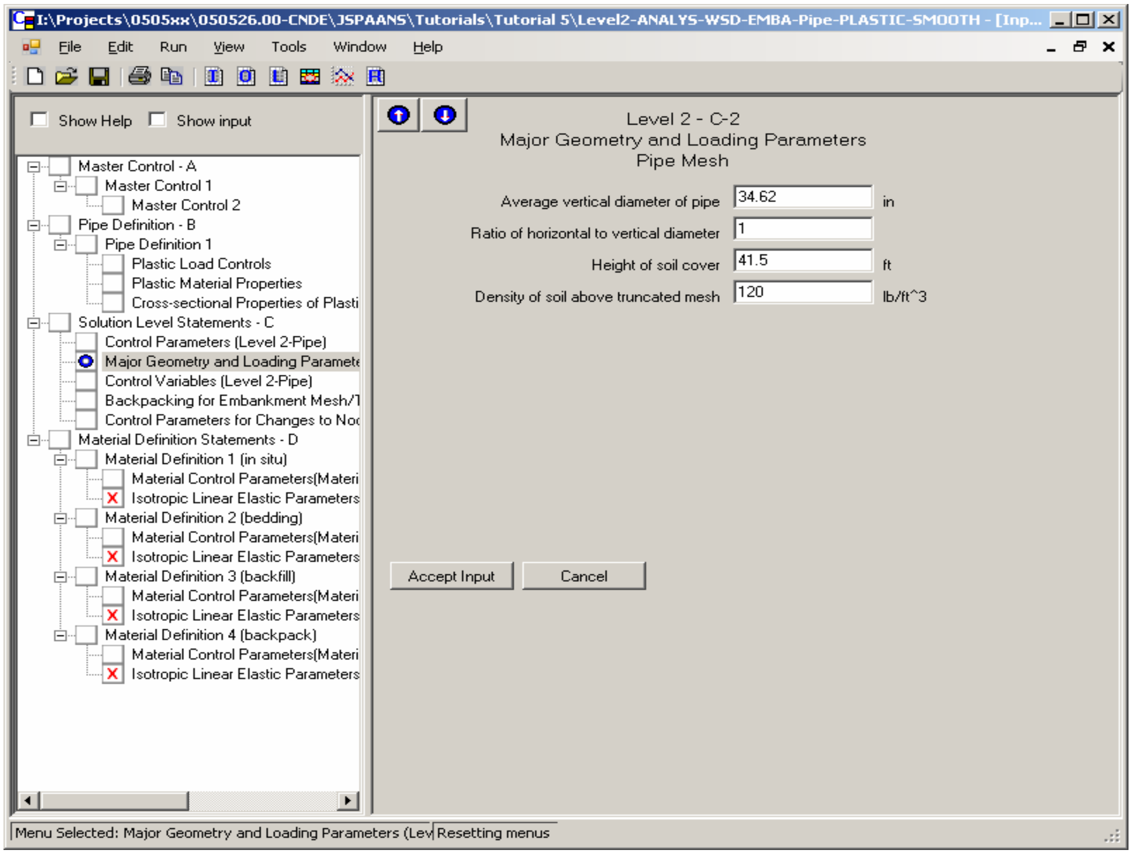


Figure 5-9 – Input Screen C-2

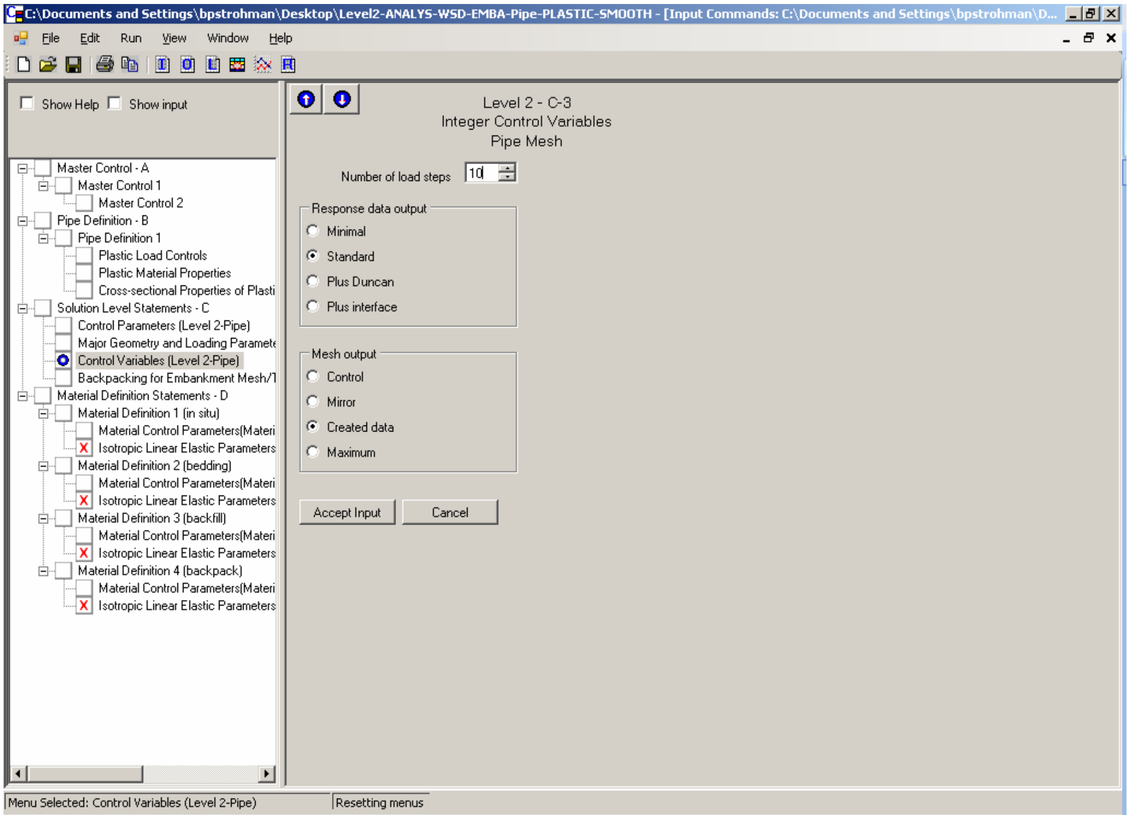


Figure 5-10 – Input Screen C-3

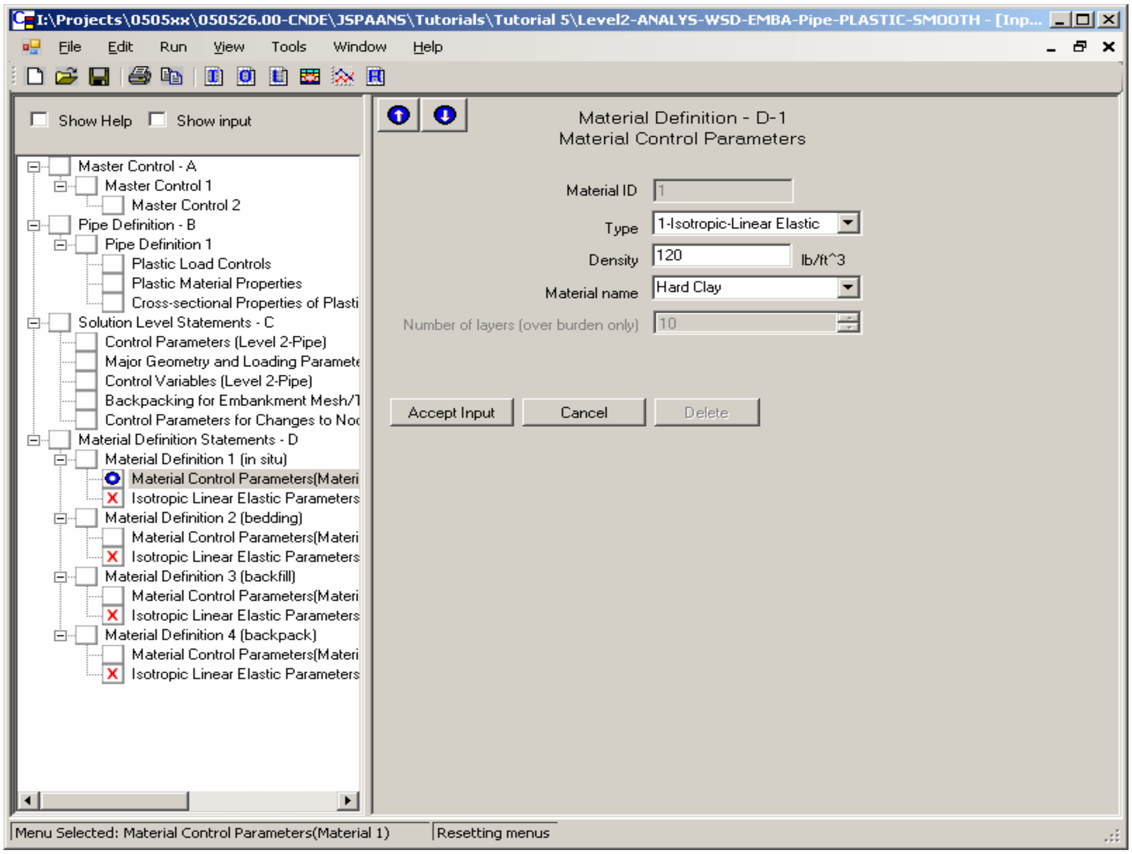


Figure 5-11 – Input Screen D-1 for Material 1 – In Situ Soil

(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

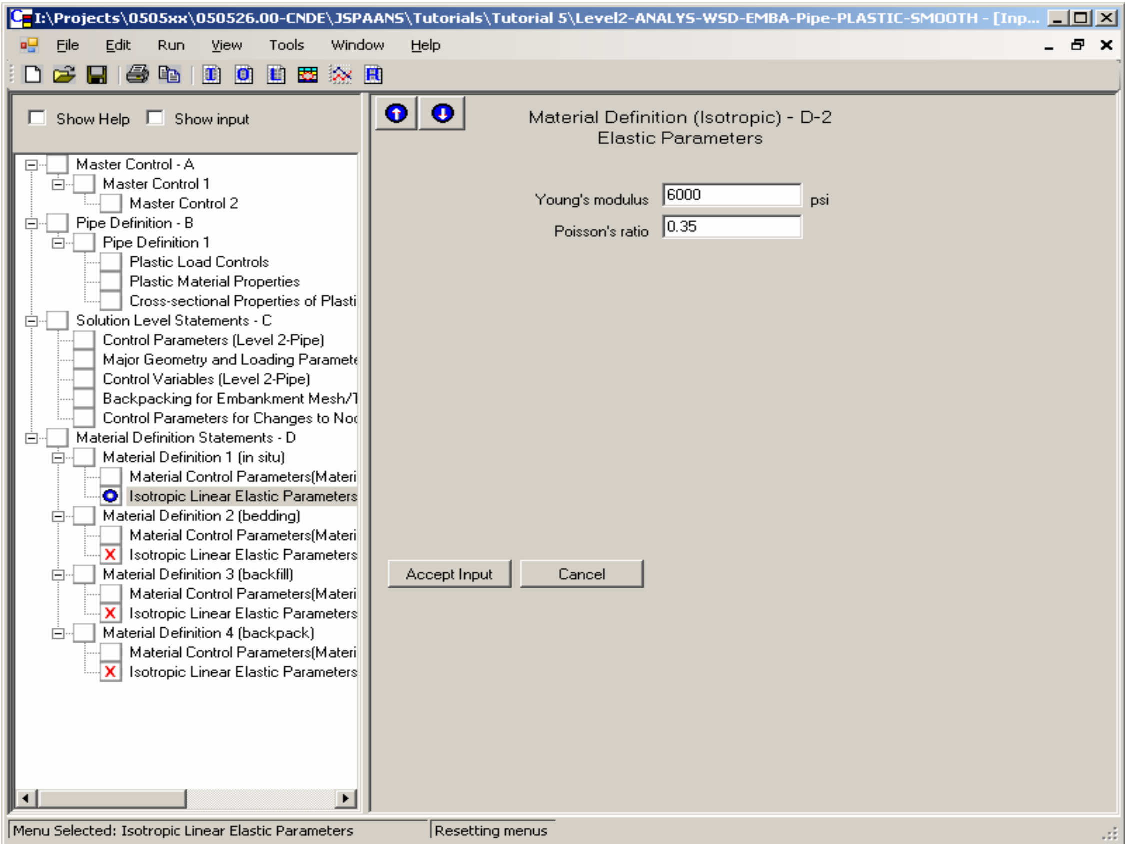


Figure 5-12 – Input Screen D-2 for Material 1 – In Situ Soil

(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

When all input is complete and saved, click "Run" and "CANDE-2007" on the main toolbar to execute the program. This will open the "Running CANDE" window which will allow you to monitor the run as it progresses. When the program is complete, a check box "CANDE Analysis Complete" window will appear. Click "OK" and check if the "Running CANDE" window states "Normal Exit From CANDE." If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the "Close" button in the "Running CANDE" window to return to the user interface.

5.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The "Plotting Parameters" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting parameters menu and click the check box "Show constr. increment numbers." Click "OK," set Load Increment to 5 to show the entire mesh (the remaining load is placed above the mesh – see *User Manual*) and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 5-13.

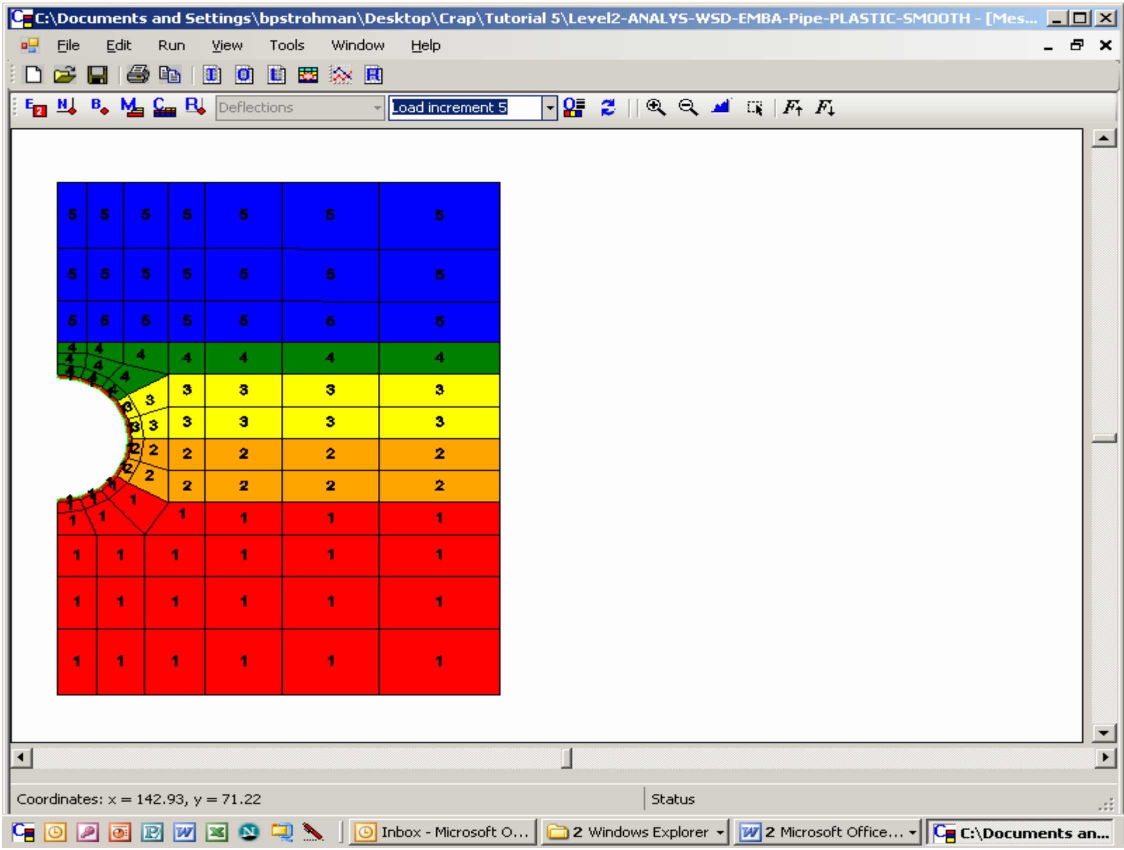


Figure 5-13 – Mesh Plot for Load Steps 1 to 5

To view horizontal strain, open the plotting parameters window and set the deflection magnification factor to 10 (see Figure 5-14).

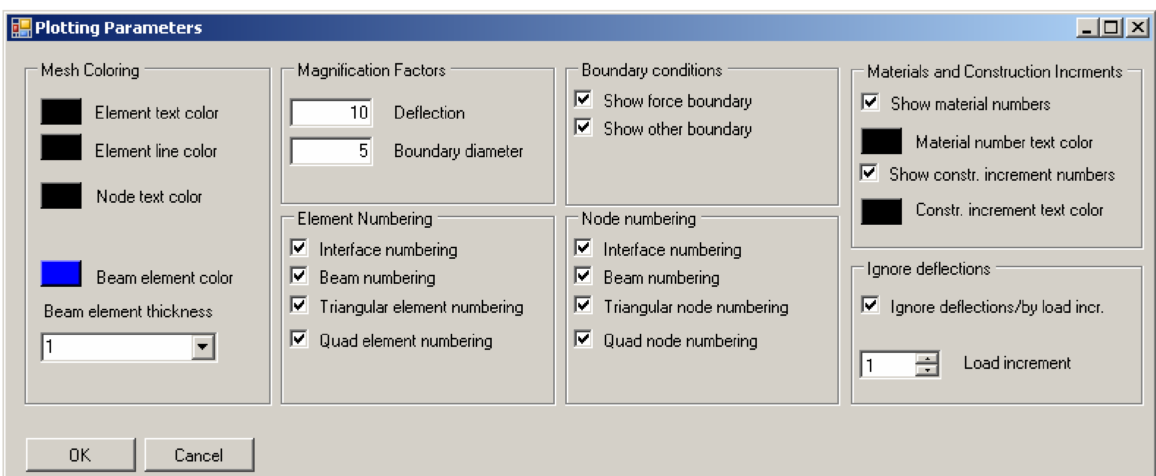


Figure 5-14 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Horizontal Strain. Set the increment to 10 and the screen should look like Figure 5-15, which also shows the deflected mesh geometry. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

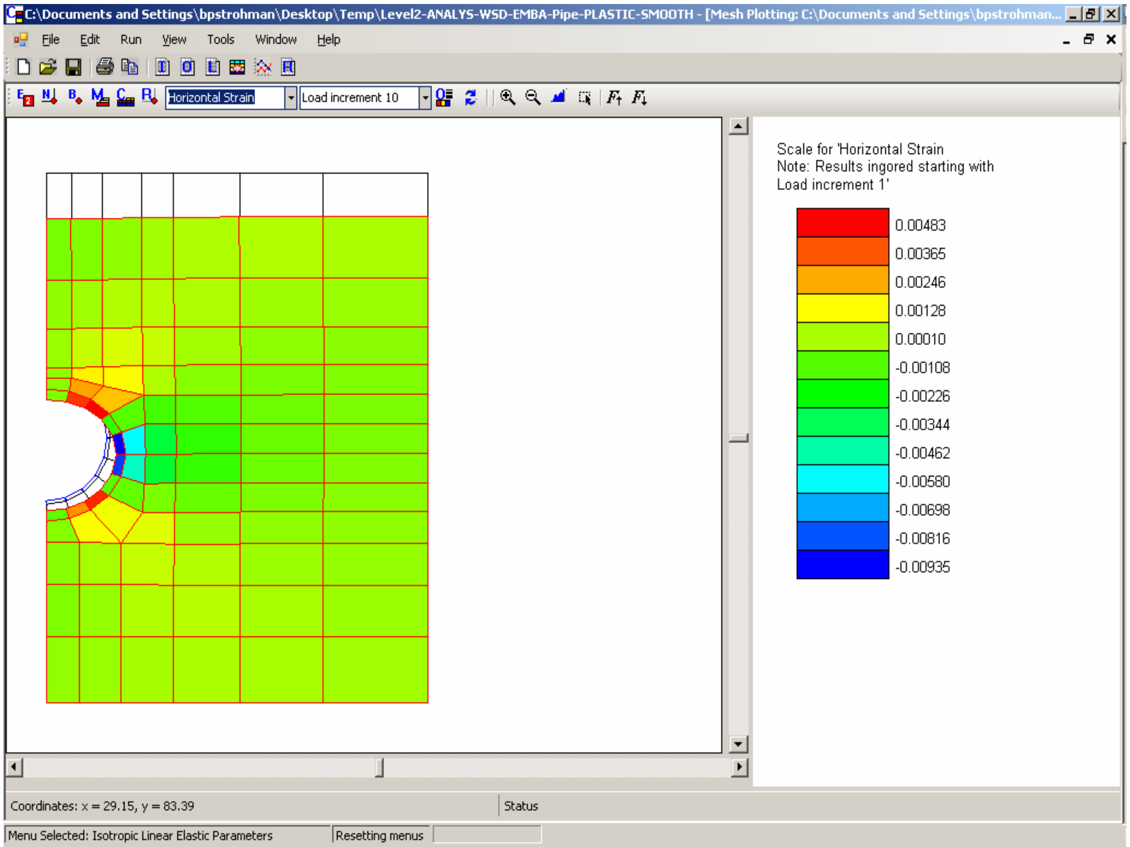


Figure 5-15 – Horizontal Strain, Load Step 10

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 5-16 compares the thrusts after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 2, 4, and 6, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Thrust Force” and the screen should appear as Figure 5-16.

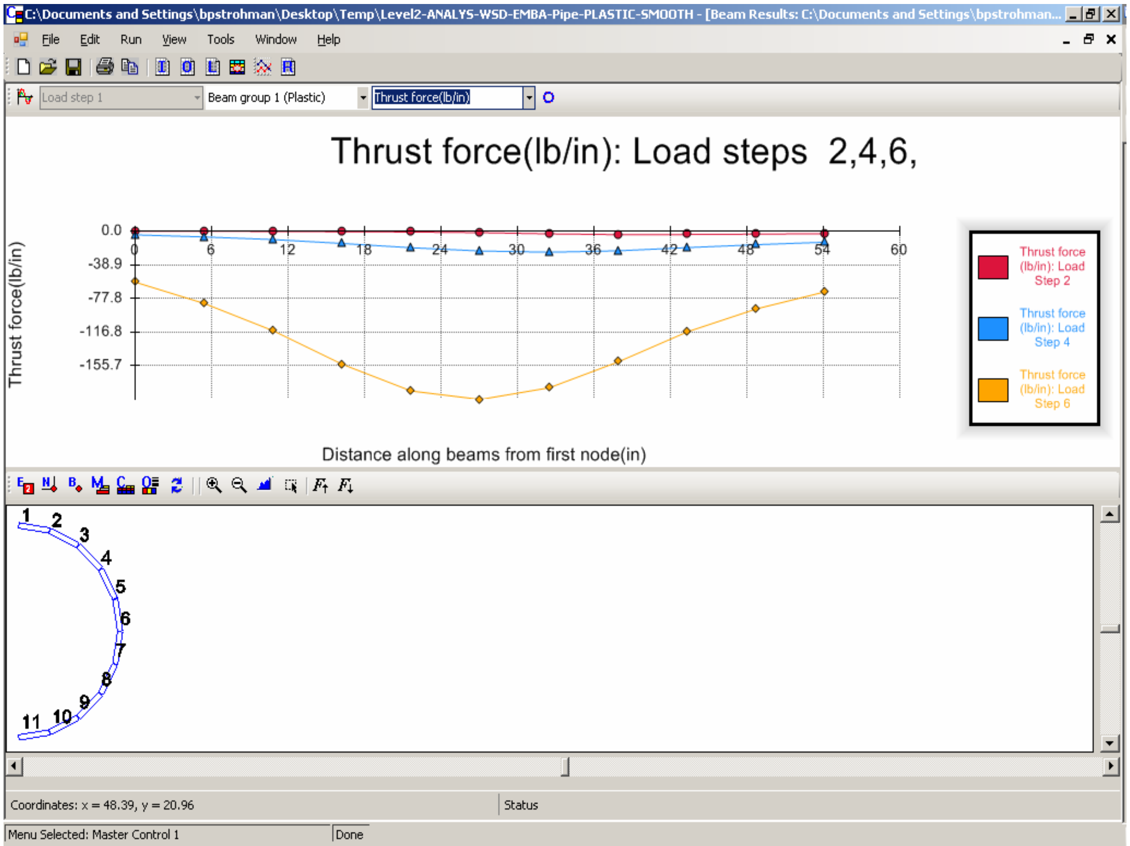


Figure 5-16 – Thrust Force for Load Steps 2, 4, and 6

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 5-17) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 5-17 shows the final assessment printed at the end of the file.

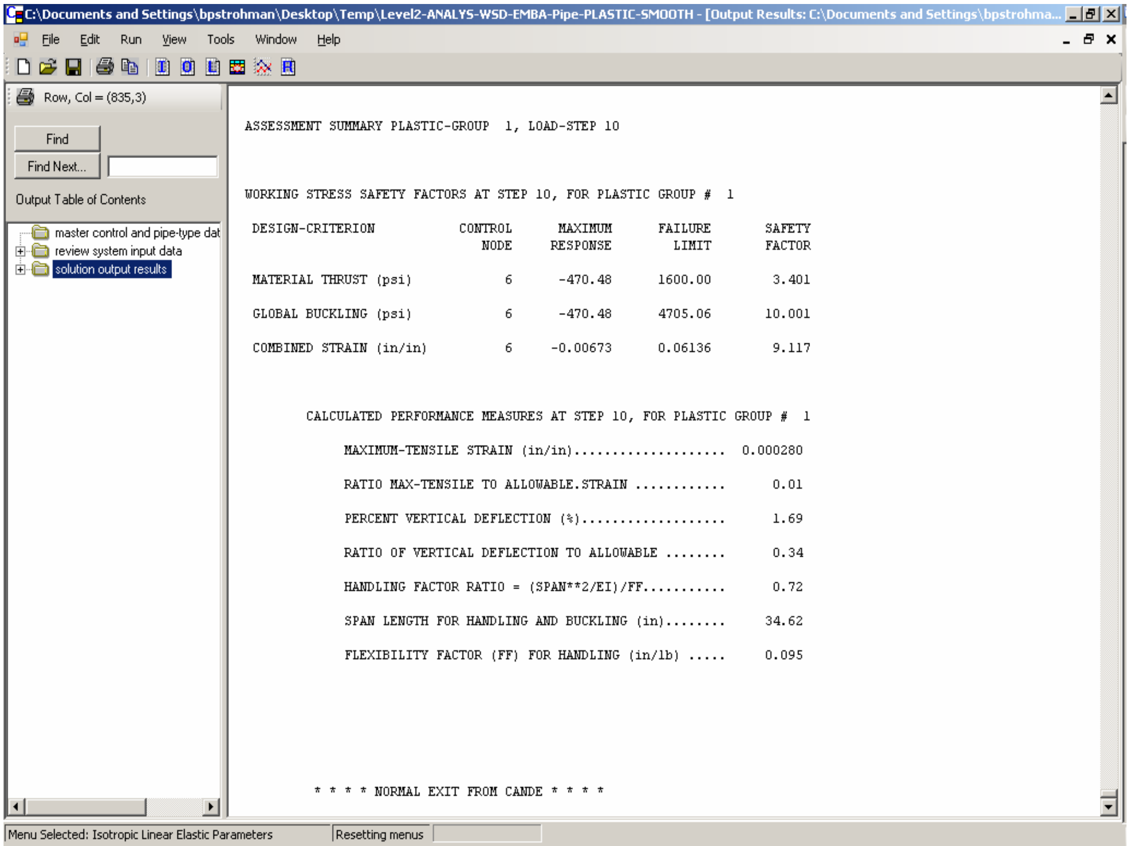


Figure 5-17 – Design Assessment Summary – Load Step 10

One additional tool is the Results Generator which can be used by selecting “View/Results Generator” or the “Start CANDE Results Generator” button. Figure 5-18 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the “Generate/Preview” button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 5-19. The three tabs shown in Figures 5-18 and 5-19 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

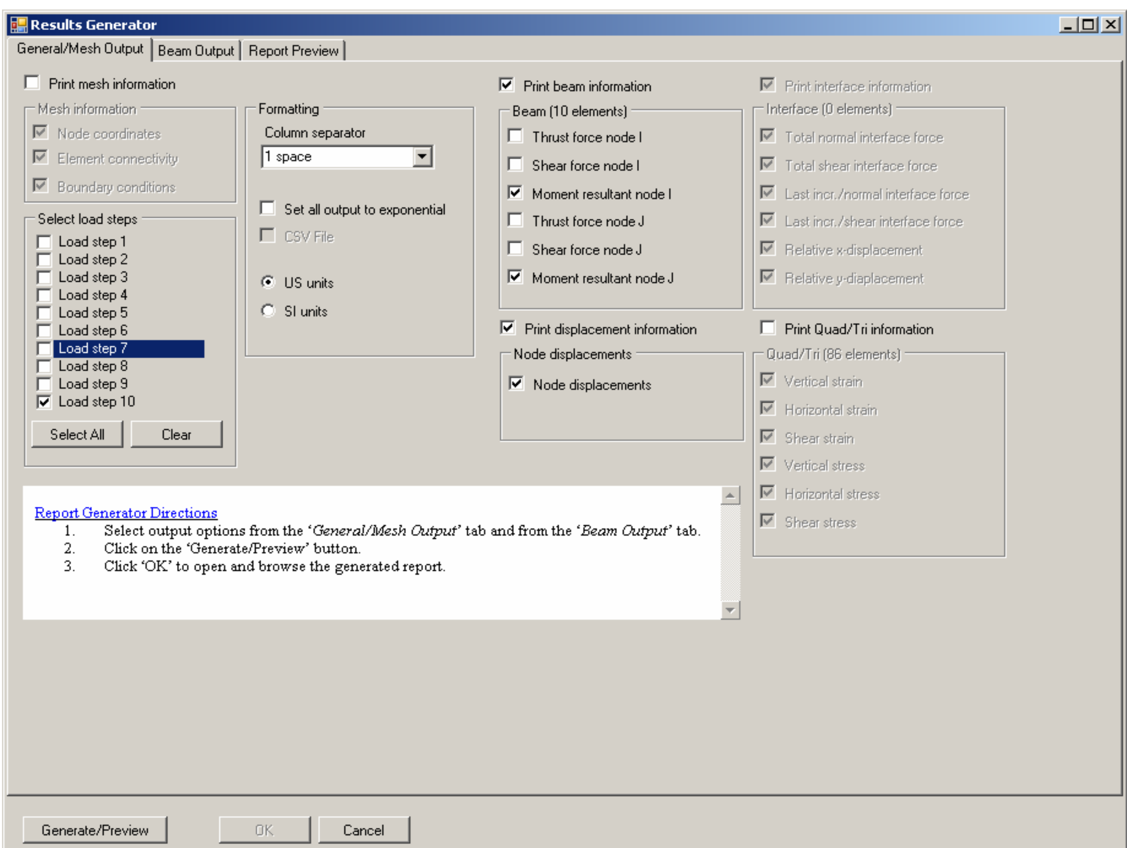
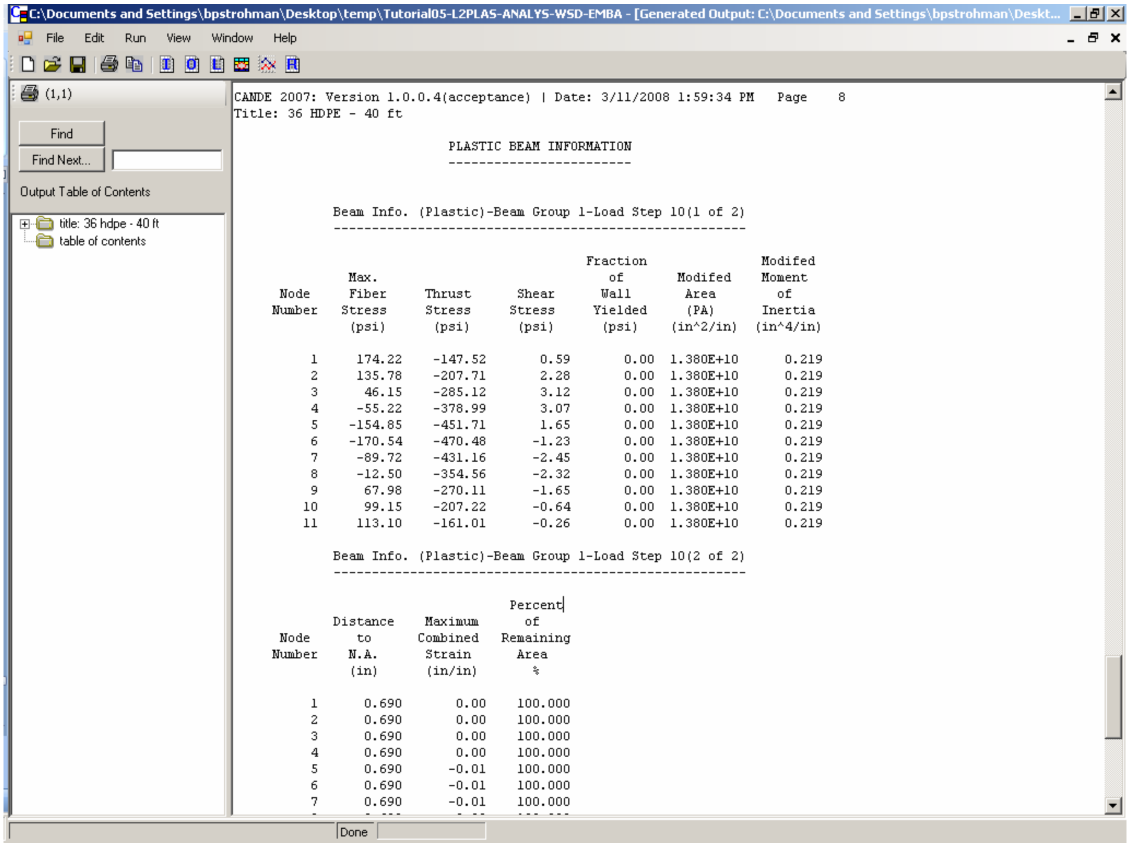


Figure 5-18 – Results Generator Input Screen – Load Step 10 Moments and Deflections



CANDE 2007: Version 1.0.0.4(acceptance) | Date: 3/11/2008 1:59:34 PM Page 8
Title: 36 HDPE - 40 ft

PLASTIC BEAM INFORMATION

Beam Info. (Plastic)-Beam Group 1-Load Step 10(1 of 2)

Node Number	Max. Fiber Stress (psi)	Thrust Stress (psi)	Shear Stress (psi)	Fraction of Wall Yielded (psi)	Modifed Area (PA) (in^2/in)	Modifed Moment of Inertia (in^4/in)
1	174.22	-147.52	0.59	0.00	1.380E+10	0.219
2	135.78	-207.71	2.28	0.00	1.380E+10	0.219
3	46.15	-285.12	3.12	0.00	1.380E+10	0.219
4	-55.22	-378.99	3.07	0.00	1.380E+10	0.219
5	-154.85	-451.71	1.65	0.00	1.380E+10	0.219
6	-170.54	-470.48	-1.23	0.00	1.380E+10	0.219
7	-89.72	-431.16	-2.45	0.00	1.380E+10	0.219
8	-12.50	-354.56	-2.32	0.00	1.380E+10	0.219
9	67.98	-270.11	-1.65	0.00	1.380E+10	0.219
10	99.15	-207.22	-0.64	0.00	1.380E+10	0.219
11	113.10	-161.01	-0.26	0.00	1.380E+10	0.219

Beam Info. (Plastic)-Beam Group 1-Load Step 10(2 of 2)

Node Number	Distance to N.A. (in)	Maximum Combined Strain (in/in)	Percent of Remaining Area %
1	0.690	0.00	100.000
2	0.690	0.00	100.000
3	0.690	0.00	100.000
4	0.690	0.00	100.000
5	0.690	-0.01	100.000
6	0.690	-0.01	100.000
7	0.690	-0.01	100.000

Figure 5-19 – Results Generator – Load Step 10 Beam Stresses

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 6

May 2008

This page intentionally left blank

6. CANDE TEST PROBLEM 6

6.1 Problem Definition

Analyze a 237-inch span (90-inch rise) reinforced concrete arch supported on spread footings with 2 ft of fill over the top of the arch, using LRFD analysis. The problem is shown schematically in Figure 6-1. The analysis will be with Level 2, using an automated finite element arch mesh for a trench installation having interface elements. The automated finite element mesh will be modified using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch. Additionally, the live load rating procedure will be demonstrated using CANDE output.

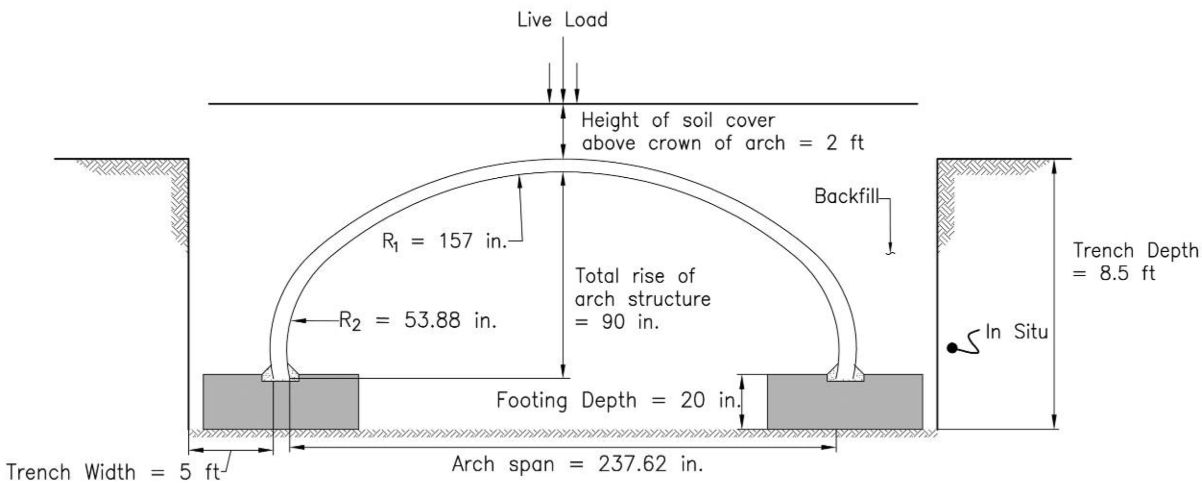


Figure 6-1 Details of Problem 6

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - LRFD

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Arch mesh

Soil mesh pattern - Trench

MOD-Make changes to the basic mesh - check on (problem requires using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch).

Number of new loading/boundary conditions to be added - 8 (for Live Load calculation see Figure 6-14.)

Pipe material type - Concrete

Reinforcement shape - Standard

Soil parameters - Canned Isotropic-Linear Elastic soil models except as noted:

In situ soil - Linear Elastic - Young's modulus = 5,000 psi, Poisson's ratio = 0.35

Footing - Linear Elastic - Young's modulus = 3,600,000 psi, Poisson's ratio = 0.17

Backfill soil - Canned Duncan/Selig - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils and 150 lb/ft^3 for the footing

Compressive strength of concrete (f'c) - 5,000 lb/in.^{2}

Shear strength equation - Pipes/arches (AASHTO 12.10.4.2.5)

Concrete strain at tension rupture - 0.00001 in./in.

Note: 0.00001 in./in. used to achieve convergence.

Compressive strain at the initial strength limit - 0.002 in./in.

Unit weight of concrete for body weight - 150 lb/ft^3

Crack width model - Heger-McGrath (AASHTO 12.10.4.2.4d)

Analysis mode - Small deformation

Yield stress of reinforcing steel - 65,000 lb/in.^{2}

Young's modulus of reinforcing steel - 29,000,000 lb/in.^{2}

Poisson's ratio of reinforcing steel - 0.3

Inner surface spacing between rows of rebar - 2 in.

Outer surface spacing between rows of rebar - 2 in.

Number of inner cage layers of reinforcement - 1

Number of outer cage layers of reinforcement - 1

Type of reinforcement - Welded wire fabric

Nonlinear behavior selection - Option 3 plus steel yielding behavior.

Concrete wall thickness - 16 in.

Steel cage area 1- 0.022 in.^{2}/in.

Steel cage area 2- 0.022 in.^{2}/in.

Concrete cover to c.l. of cage 1 - 1.25 in.

Concrete cover to c.l. of cage 2 - 1.25 in.

Number of construction steps - See *User Manual*, Chapter 5, C-2 – The default mesh provides 8 construction steps to the top of the trench. Use an additional 4 steps for the live load (applying load in steps results in fewer convergence issues), making a total of 12 construction steps.

Height of soil cover above crown of arch - 2 ft

Density of soil above truncated mesh - 120 lb/ft^3

Trench depth - 8.5 ft

Trench width - 5 ft

Total rise of arch structure - 90 in.

One-half of arch span at footing level - 118.81 in.

Footing depth - 20 in.

Outside footing width - 30 in.

Inside footing width - 30 in.

Radius of top arch (segment 1) - R1 - 157 in.

Angle for R1 segment - 40 degrees

Radius of second segment - R2 - 53.88 in.

Angle for R2 segment - 62.87 degrees

LRFD load factors - 1.35 for load steps 1 through 8 and 1.75 for load steps 9 through 12

** The 1.2 multiple presence factor and 1.2475 impact factor are part of the service load and thus are included in the live load calculation).

Load modifier – 1.05 (non-redundant for earth load)

6.2 Creating the CANDE Input Document

Figures 6-2 through 6-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 3, CANDE will display a screen indicating that an input path is initiated – click “Finish.” The user is prompted for a filename and directory.

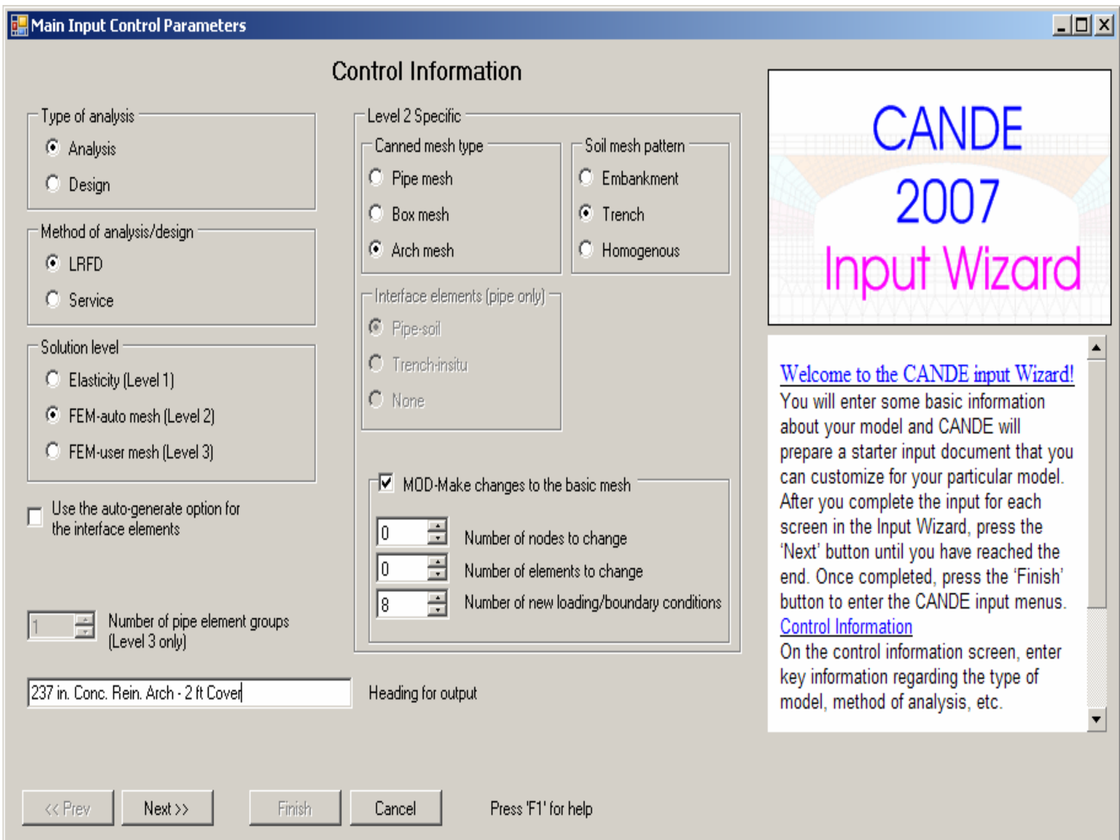


Figure 6-2 – Input Wizard, Screen 1

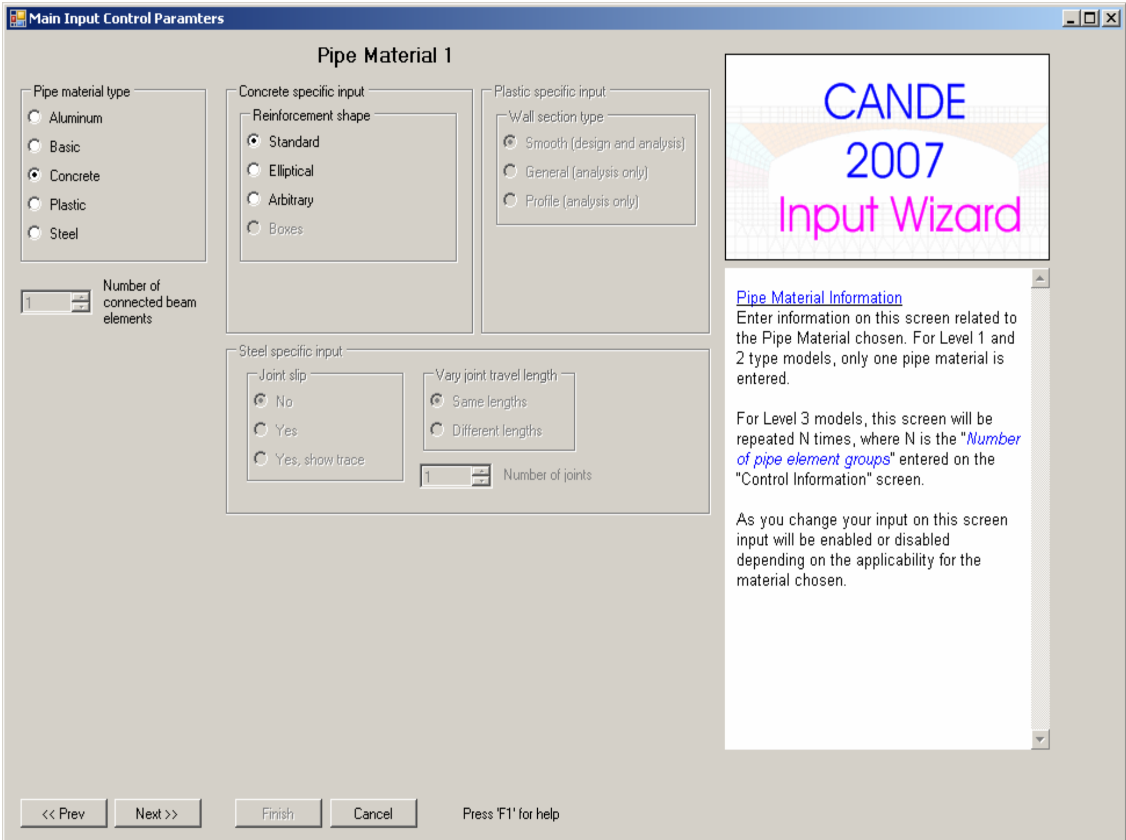


Figure 6-3 – Input Wizard, Screen 2

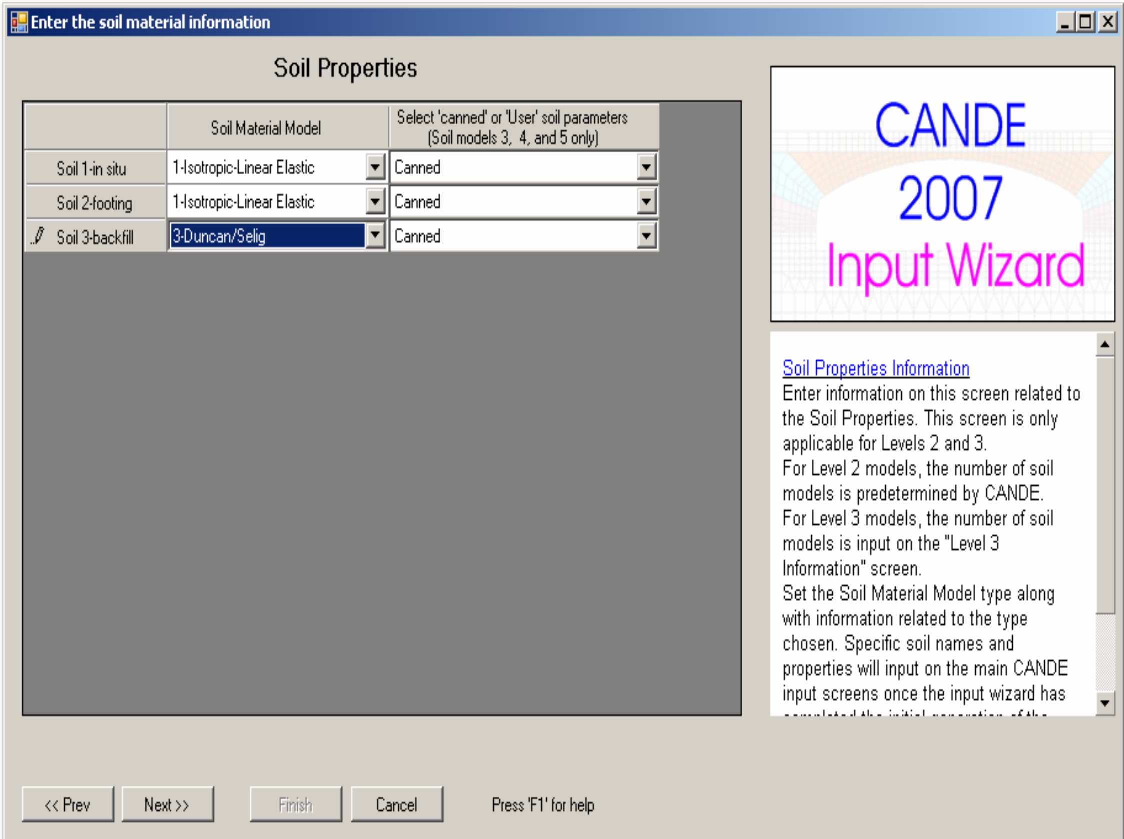


Figure 6-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 6-5. Enter an appropriate heading for output and click “Accept Input.”

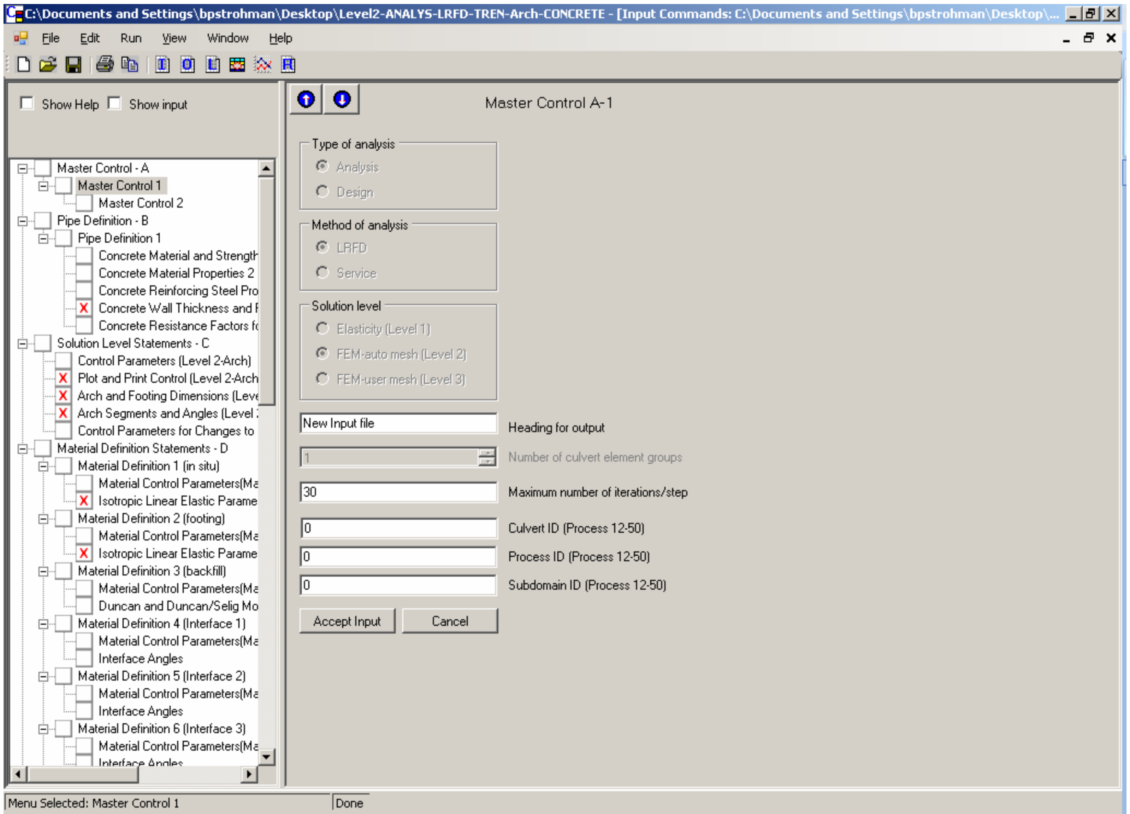


Figure 6-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 6-5 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 6-6 through 6-12 and Figures 6-16 through 6-18 show the completed input for the standard screens requiring data for the tutorial, except that only one material definition screen is shown. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data. Figures 6-13 through 6-15 show the live load calculation and use of the Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch. Note that the interface properties will be modified in the "file/open text input" mode after all of the other material properties and problem definitions have been defined (see Figures 16-19 through 16-21).

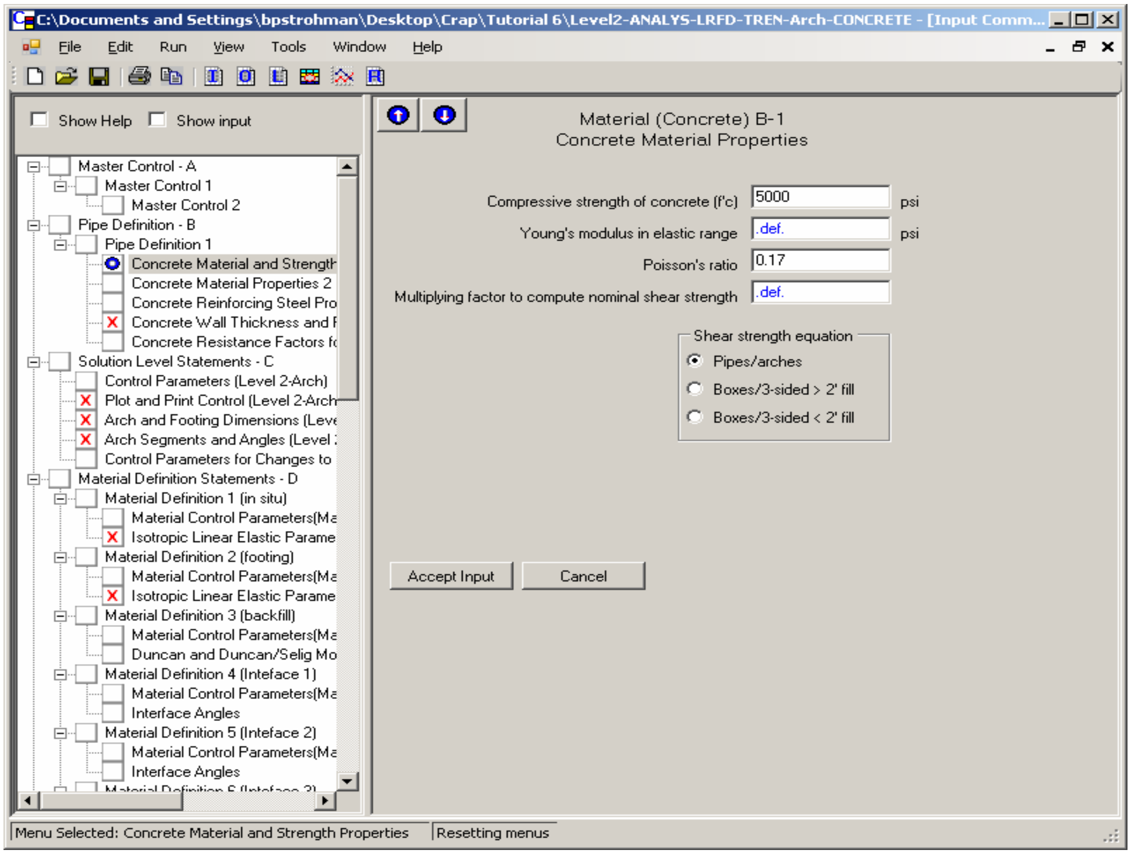


Figure 6-6 – Input Screen B-1

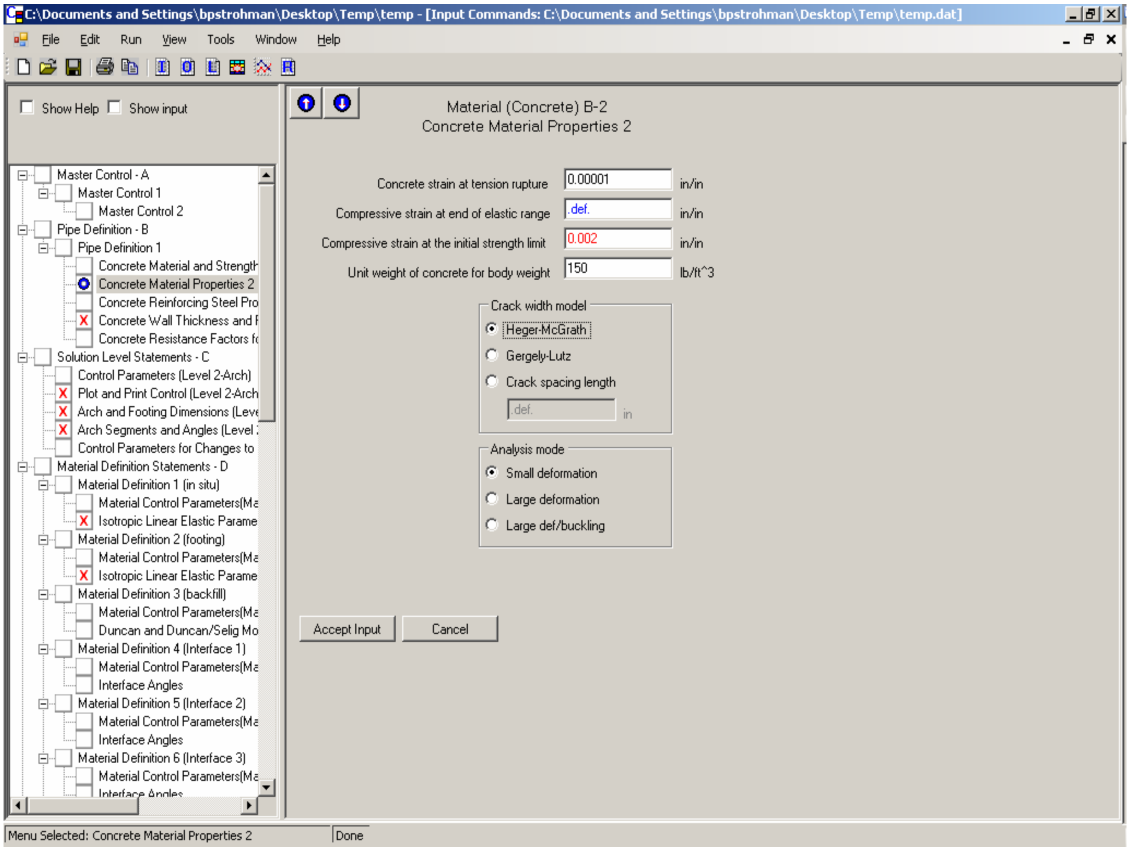


Figure 6-7 – Input Screen B-2

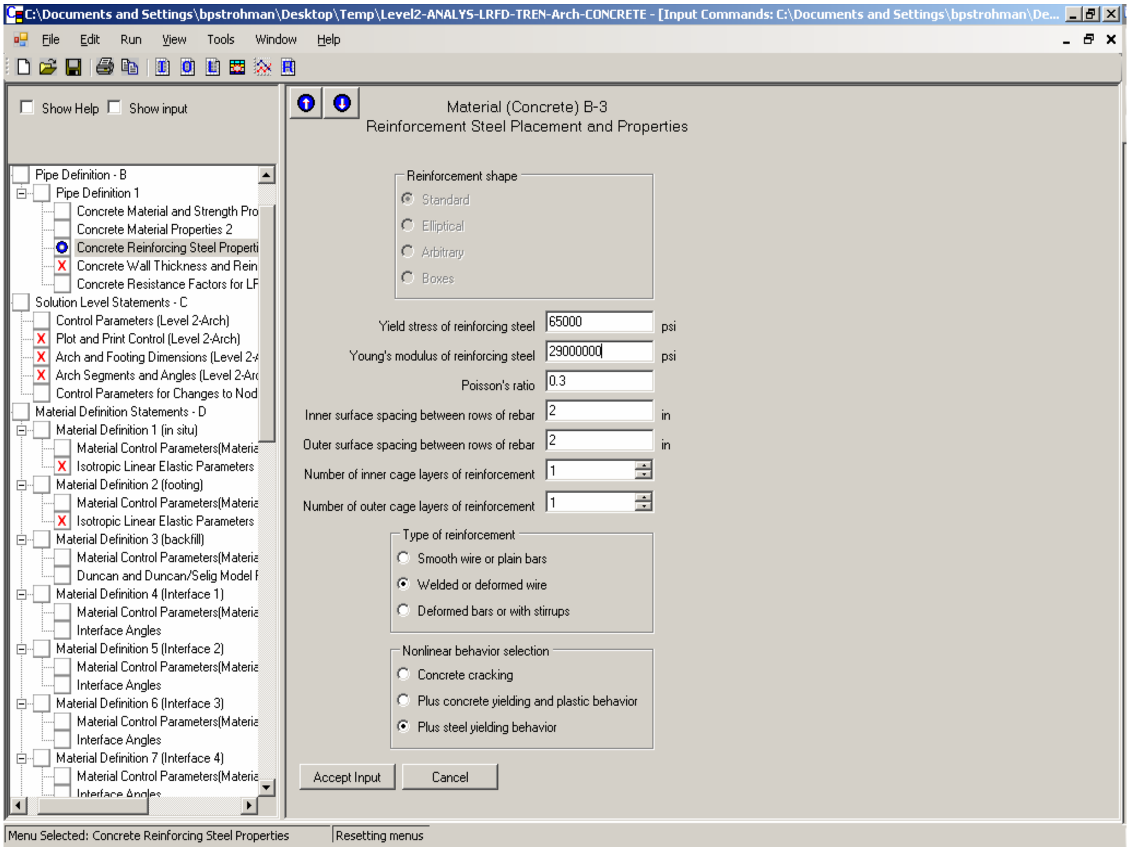


Figure 6-8 – Input Screen B-3

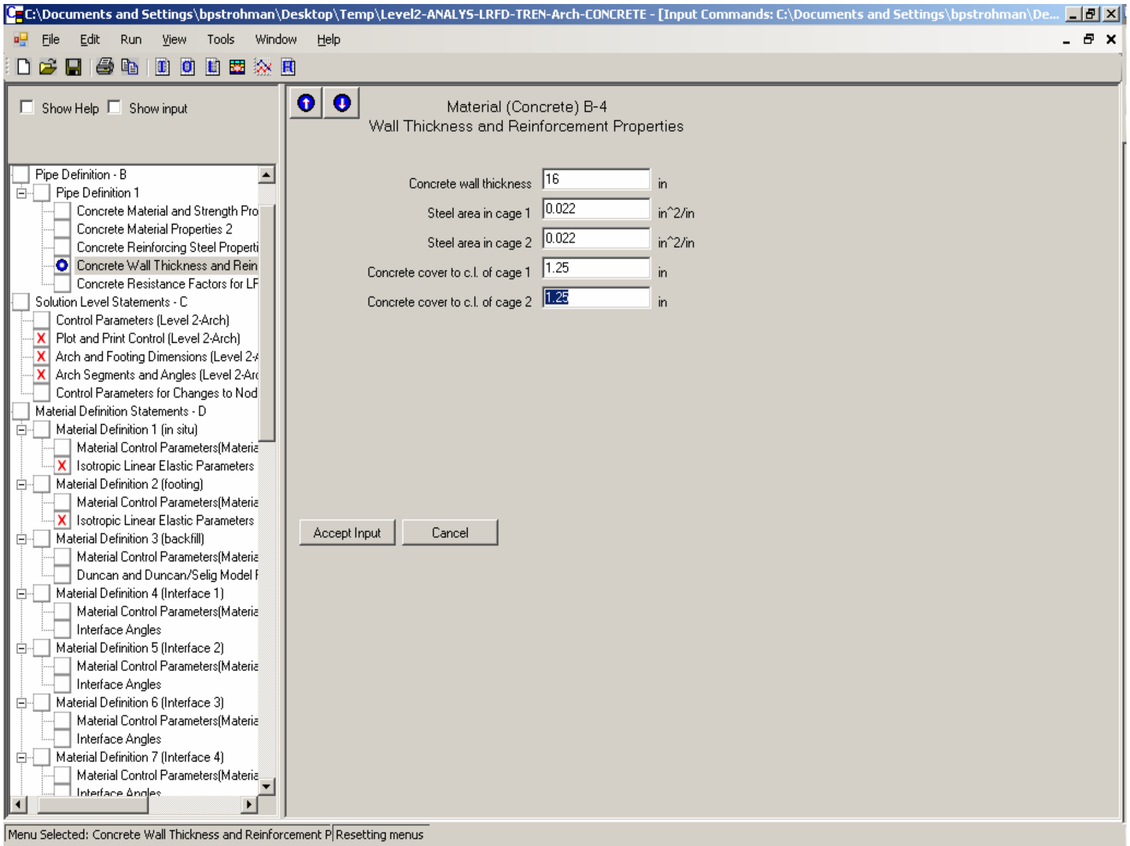


Figure 6-9 – Input Screen B-4

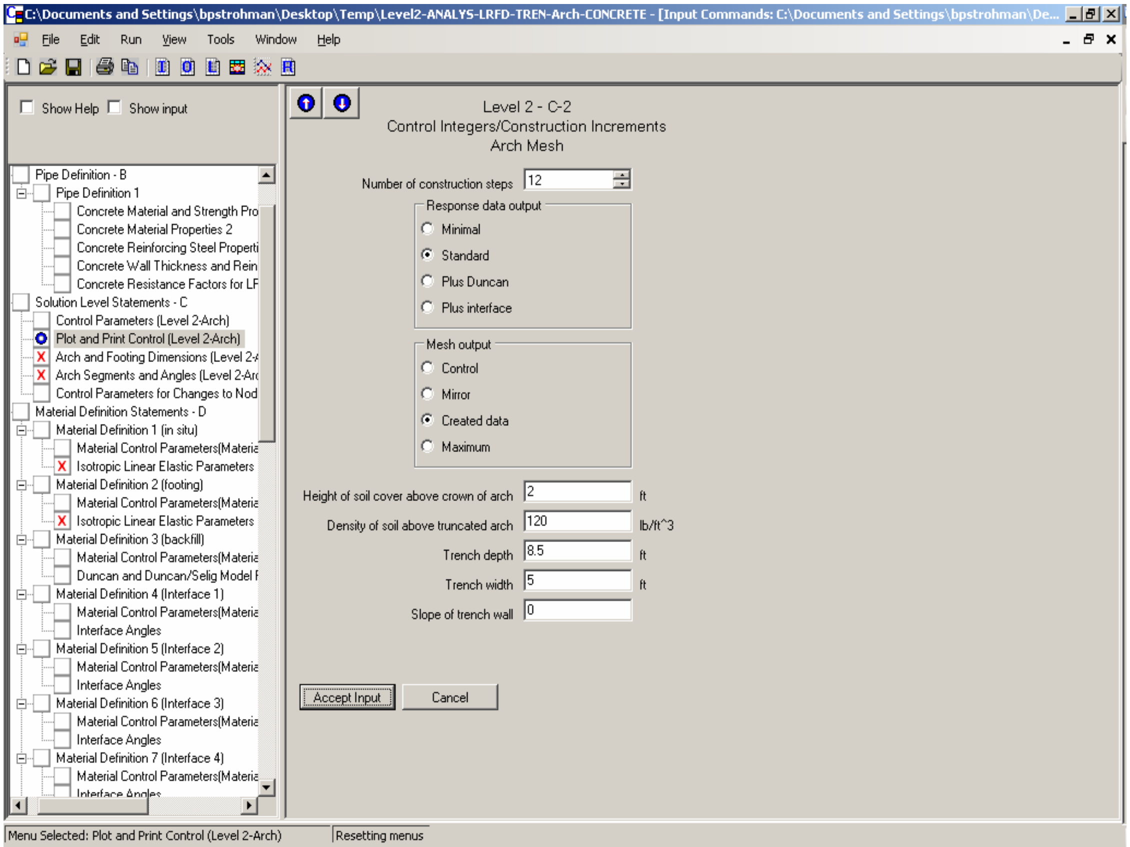


Figure 6-10 – Input Screen C-2

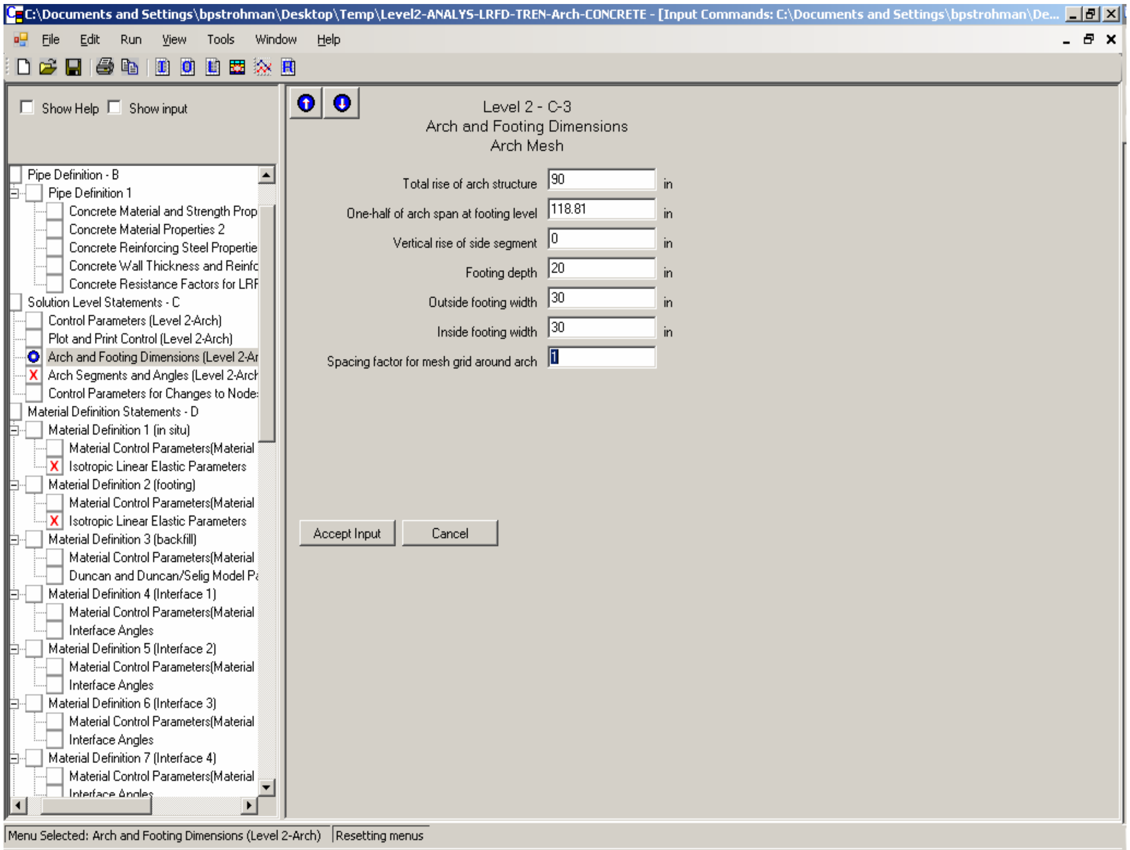


Figure 6-11 – Input Screen C-3

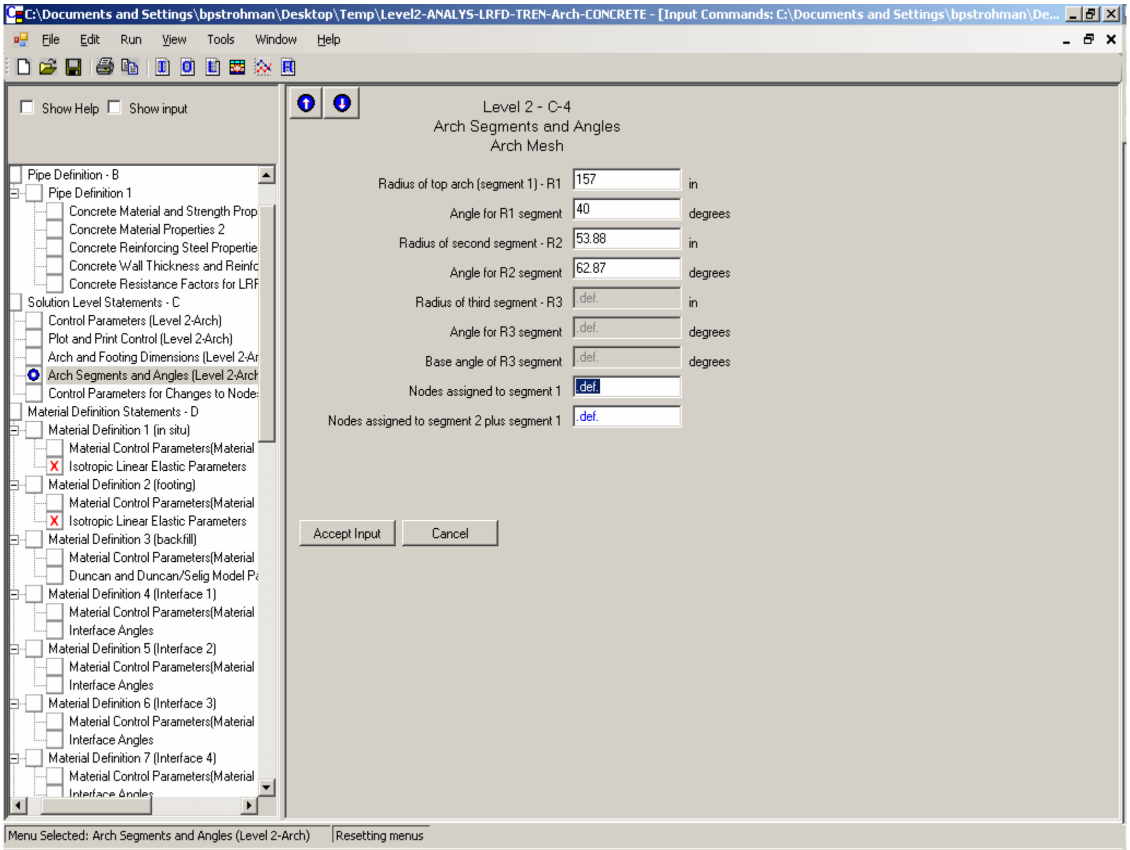


Figure 6-12 – Input Screen C-4

The automated finite element mesh will be modified using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch. Figures 6-13 and 6-15 show the completed input screens for the Level 2-extended. Figure 6-14 shows the calculation of the two dimensional out-of-plane live load through soil for use in the CANDE finite element model for a Design Truck (HS-20) at the ground surface 2 ft above the crown of the arch traveling parallel to the span of the culvert.

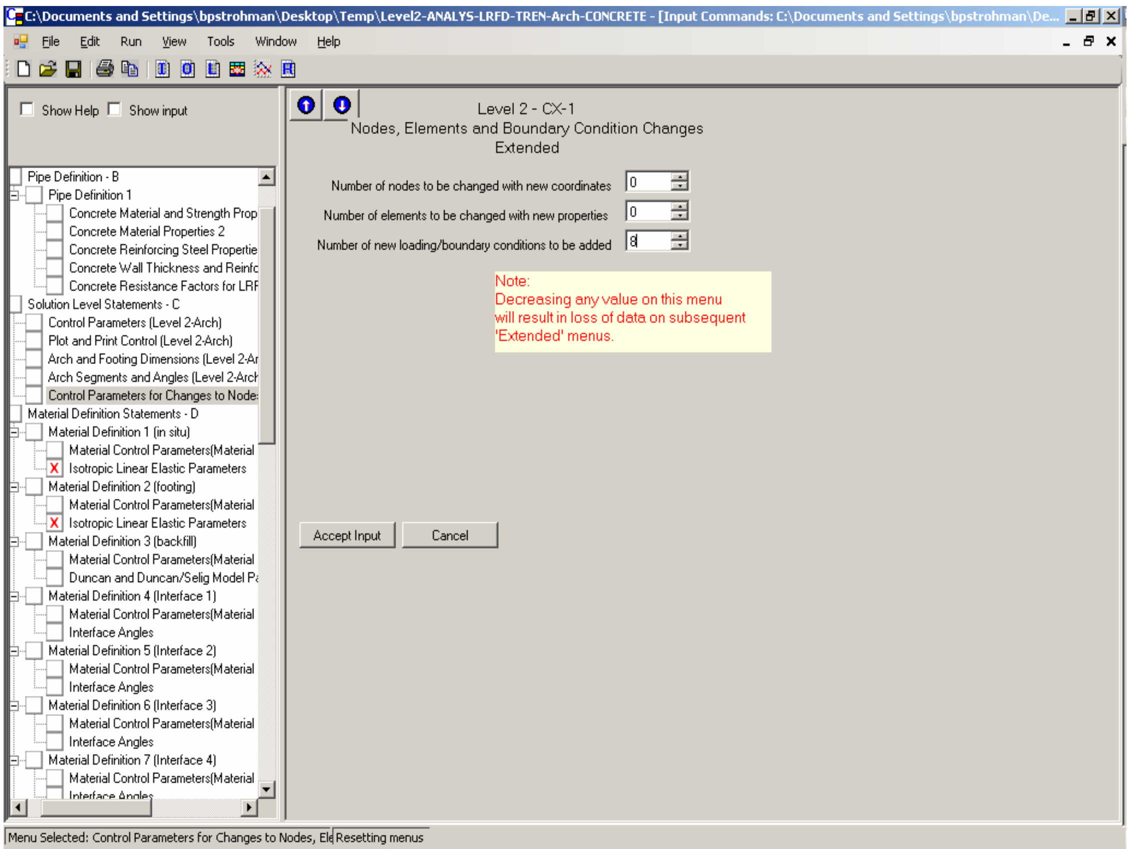


Figure 6-13 – Input Screen CX-1

Live Load Calculation

DISTRIBUTION OF LIVE LOAD THROUGH EARTH FILLS - AASHTO LRFD

Purpose:
This sheet demonstrates calculating the two-dimensional out-of-plane distribution of live load through soil for use with CANDE FEM for a Design Truck (HS-20) traveling parallel to the span of the culvert. AASHTO LRFD code references are shown in parenthesis.

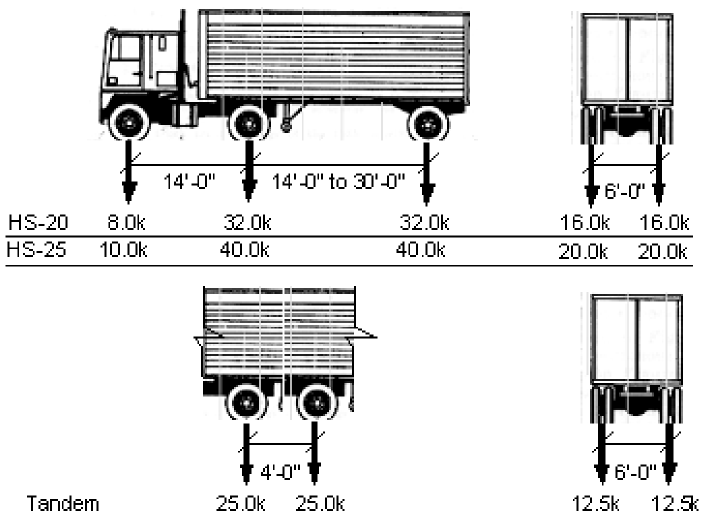


Figure 1. AASHTO Design Truck (HS-20), Design Truck plus 25% (HS-25), and Design Tandem

NOTES

1. Input values shown in grey.
2. := indicates definition of a variable.
3. = prints a previously defined variable.

GENERAL PROPERTIES

Depth of Fill/ Cover.. $H_E := 2ft$

Bridge Dimensions:

Bridge Span.. $Span := 19ft + 9.625in$

Design Truck:

Axle Load.. $P_{axle} := 32kips$

Tire Load... $P_{tire} := \frac{P_{axle}}{2}$

Axle Width (distance between tires)...................... $w_{axle} := 6ft$

Tire Contact Area (LRFD 3.6.1.2.5)

Contact Width.. $w_{tire} := 20in$

Contact Length... $L_{tire} := 10in$

DESIGN FACTORS

Live Load Distribution Factor (LRFD 3.6.1.2.6)...................... $LLDF := 1.15$ (Select Granular Backfill)

Multiple Presence of Live Load (LRFD 3.6.1.1.2).................. $mpf := 1.20$

Live Load Factor (Service I).. $LL := 1.0$

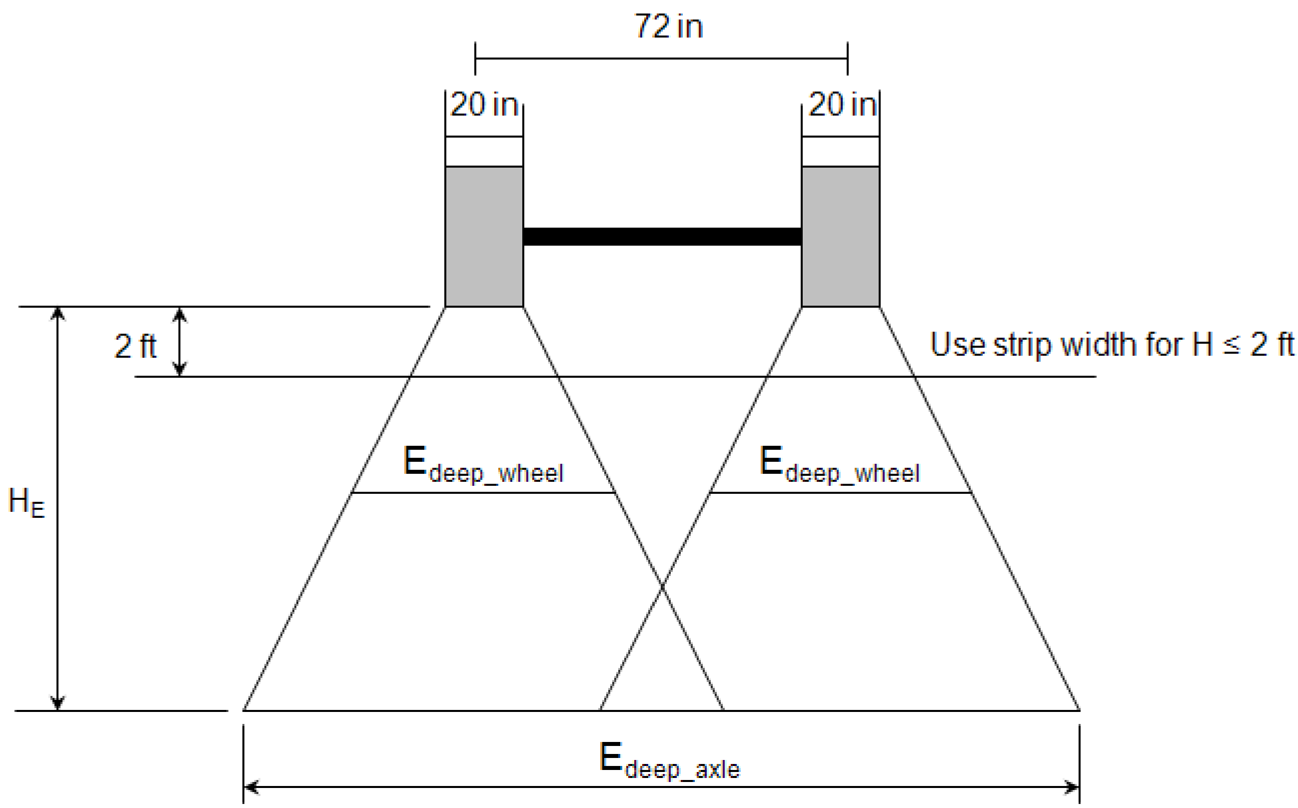
Figure 6-14 – Live Load Calculation

Impact for Buried Components (LRFD 3.6.2.2)

Dynamic Allowance.. $IM := 33 \cdot \left(1.0 - 0.125 \frac{H_E}{ft}\right)$

Impact Factor.. $I_{imp} := \max\left[1.0, \left(1 + \frac{IM}{100}\right)\right]$ $I_{imp} = 1.248$

FOR DEPTH OF FILL GREATER THAN 2FT(LRFD 3.6.1.2.6)



Wheel Load Equivalent Distribution Width............ $E_{deep\_wheel} := w_{tire} + LLDF \cdot H_E$ $E_{deep\_wheel} = 47.6 in$

Axle Load Equivalent Distribution Width................ $E_{deep\_axle} := w_{axle} + w_{tire} + LLDF \cdot H_E$ $E_{deep\_axle} = 119.6 in$

Wheel Live Load Equivalent Distribution.............
(no overlap)
$LL_{deep\_wheel} := \frac{P_{tire} \cdot I_{imp} \cdot mpf}{E_{deep\_wheel}}$ $LL_{deep\_wheel} = 503.2 \frac{lbf}{in}$

Axle Live Load Equivalent Distribution................
(overlap)
$LL_{deep\_axle} := \frac{P_{axle} \cdot I_{imp} \cdot mpf}{E_{deep\_axle}}$ $LL_{deep\_axle} = 400.5 \frac{lbf}{in}$

Determine Controlling Distribution.....................
(check wheel overlap)

$$LL_{deep} := \begin{cases} LL_{deep\_wheel} & \text{if } E_{deep\_wheel} \le \frac{E_{deep\_axle}}{2} \\ LL_{deep\_axle} & \text{otherwise} \end{cases}$$

$LL_{deep} = 503.2 \frac{lbf}{in}$

Figure 6-14 – Live Load Calculation (continued)

FOR DEPTH OF FILL LESS THAN 2FT (LRFD 4.6.2.10)

Equivalent Axle Distribution Width............................ $E_{shallow} := \left(96 + 1.44\frac{Span}{ft}\right)\cdot in$ $E_{shallow} = 124.5in$

Axle Live Load Distribution.................................... $LL_{shallow} := \frac{P_{axle}\cdot I_{imp}\cdot mpf}{E_{shallow}}$ $LL_{shallow} = 384.7\frac{lbf}{in}$

TWO-DIMENSIONAL LIVE LOAD - (SERVICE I)

Two Dimensional Live .... $LL_{2D} := \begin{cases} LL_{shallow} & \text{if } H_E \le 2ft \\ LL_{deep} & \text{if } H_E > 2ft \end{cases}$ $LL_{2D} = 384.7\frac{lbf}{in}$

If Modeling Full Structure:

Number of Elements Over Wheel Length.................. $n_{elems} := 2$ (3 nodes)

use equal mesh spacing

Load per Interior Node.. $l_{int.node} := \frac{LL_{2D}}{n_{elems}}$ $l_{int.node} = 192.36\frac{lbf}{in}$

Load per Exterior Node.. $l_{ext.node} := \frac{l_{int.node}}{2}$ $l_{ext.node} = 96.18\frac{lbf}{in}$

Apply Wheel Load in Increments...........................
(fewer convergence issues) $n_{con.wheel} := 4$

Load per Interior Node per
Construction Increment... $l_{int.node\_per\_con\_incr} := \frac{LL_{2D}}{n_{con.wheel}\cdot n_{elems}}$

$l_{int.node\_per\_con\_incr} = 48.09\frac{lbf}{in}$

For this problem, the interior node is on the plane of symmetry.
Use half of the load above if modeling half the structure.

Load per Exterior Node per
Construction Increment... $l_{ext.node\_per\_con\_incr} := \frac{l_{int.node}}{2\cdot n_{con.wheel}}$

$l_{ext.node\_per\_con\_incr} = 24.05\frac{lbf}{in}$

Figure 6-14 – Live Load Calculation (continued)

Figure 6-15 demonstrates adding nodal loads to 2 nodes to represent a LRFD design truck at the ground surface 2 ft above the crown of the arch. The node numbers are identified in Chapter 5, C-4 of the *User Manual* or can be obtained by making a trial run without the applied load and then using the mesh plot.

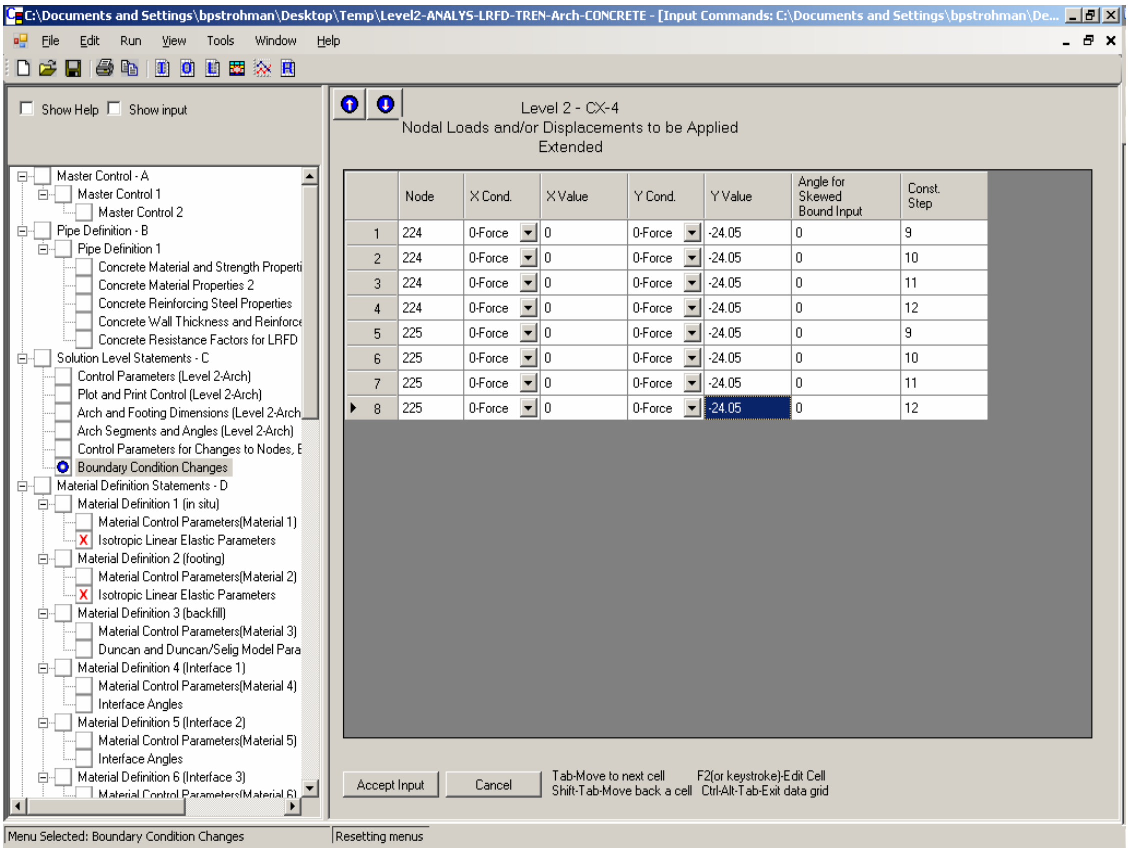


Figure 6-15 – Input Screen CX-4

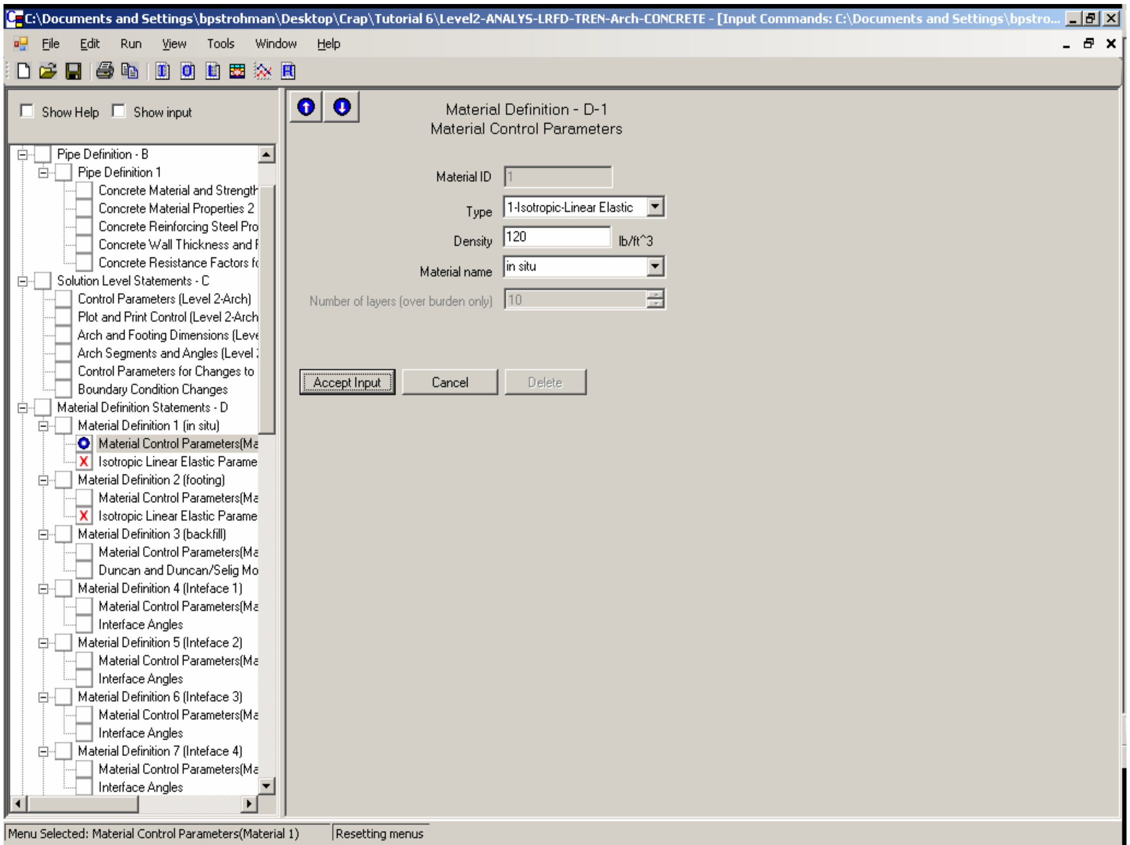


Figure 6-16 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

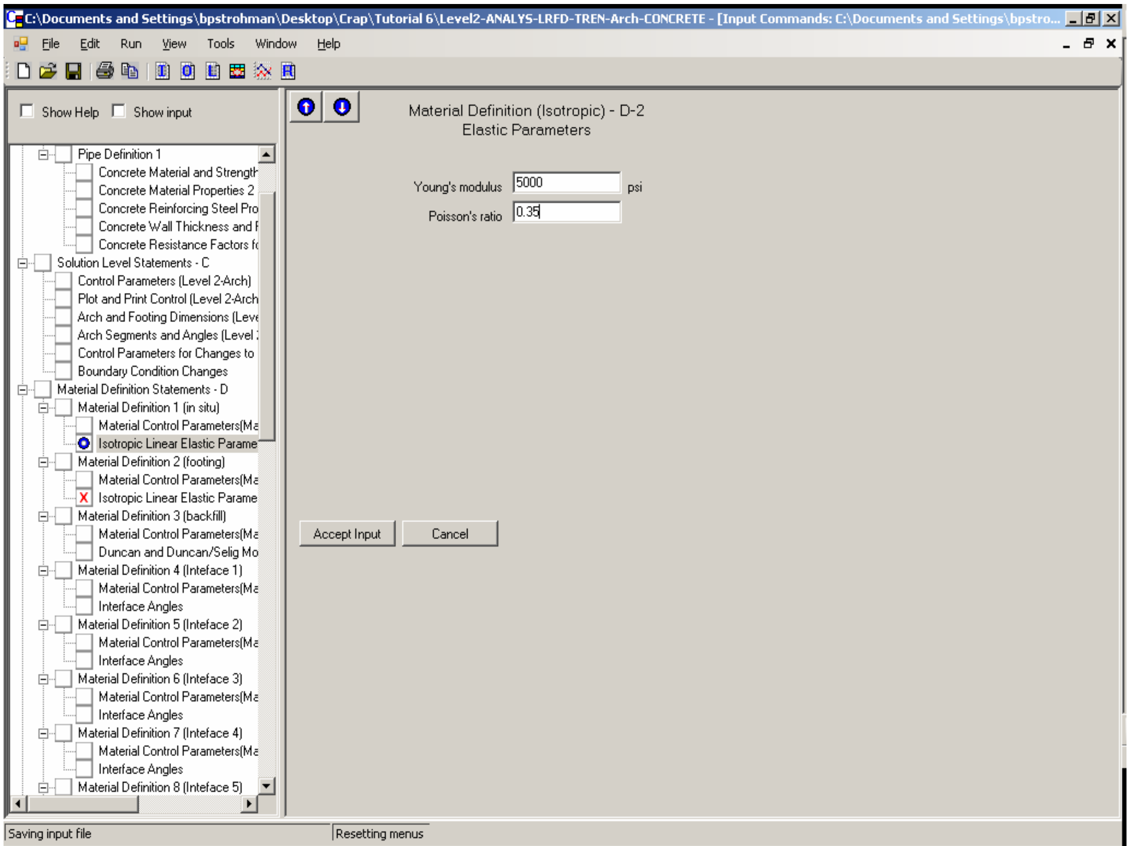


Figure 6-17 – Input Screen D-2 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

Properties for material definitions 5 through 22 (Interfaces 1-19) will not be input manually. These properties will be input using a shortcut method depicted in Figures 6-19 through 6-21. As a result, proceed to Input Screen E-1.

Under LRFD load factor in Figure 6-18 enter the combined value of the load factor and load modifier, i.e. – 1.35 * 1.05 = 1.42 for load steps 1 through 8 and 1.75 for load steps 9 through 12.

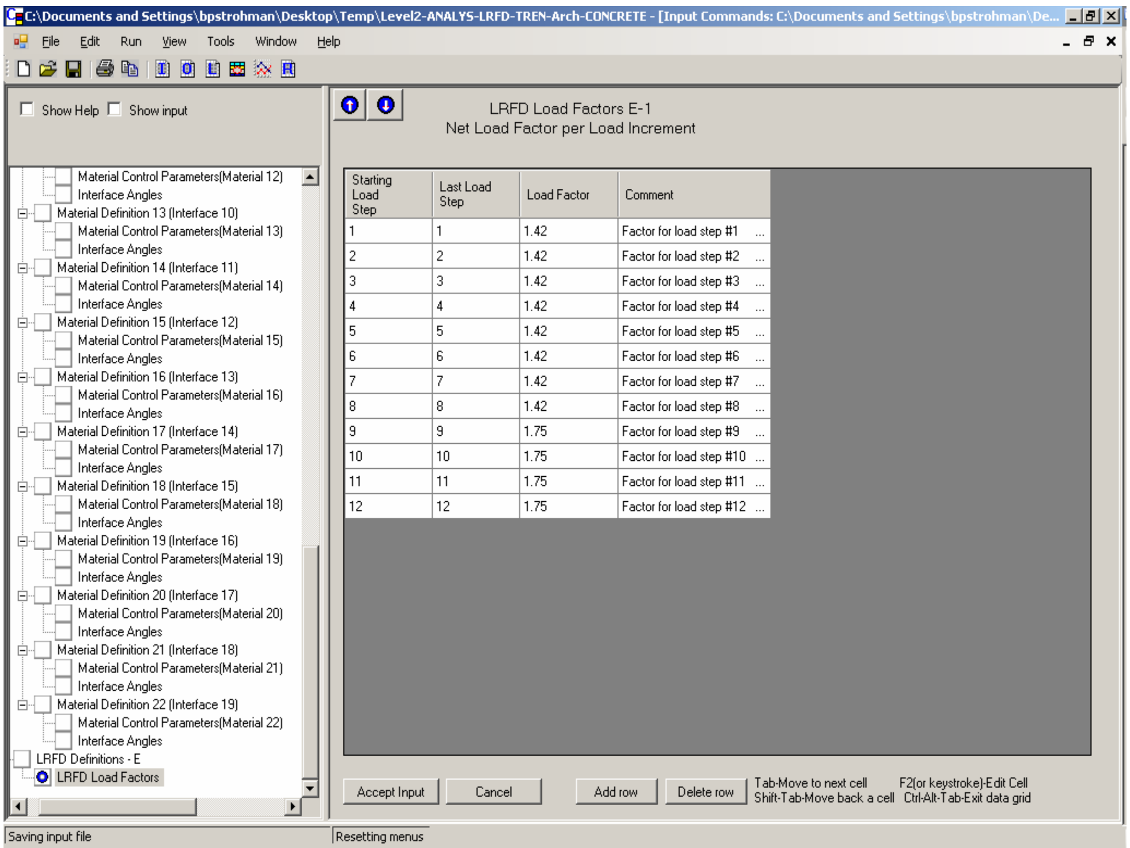


Figure 6-18 – Input Screen E-1

When input using the graphic interface is complete, save the input file. Reopen in text input mode by going to "File/open text input" to input the interface angles. The screen should look like Figure 6-19.

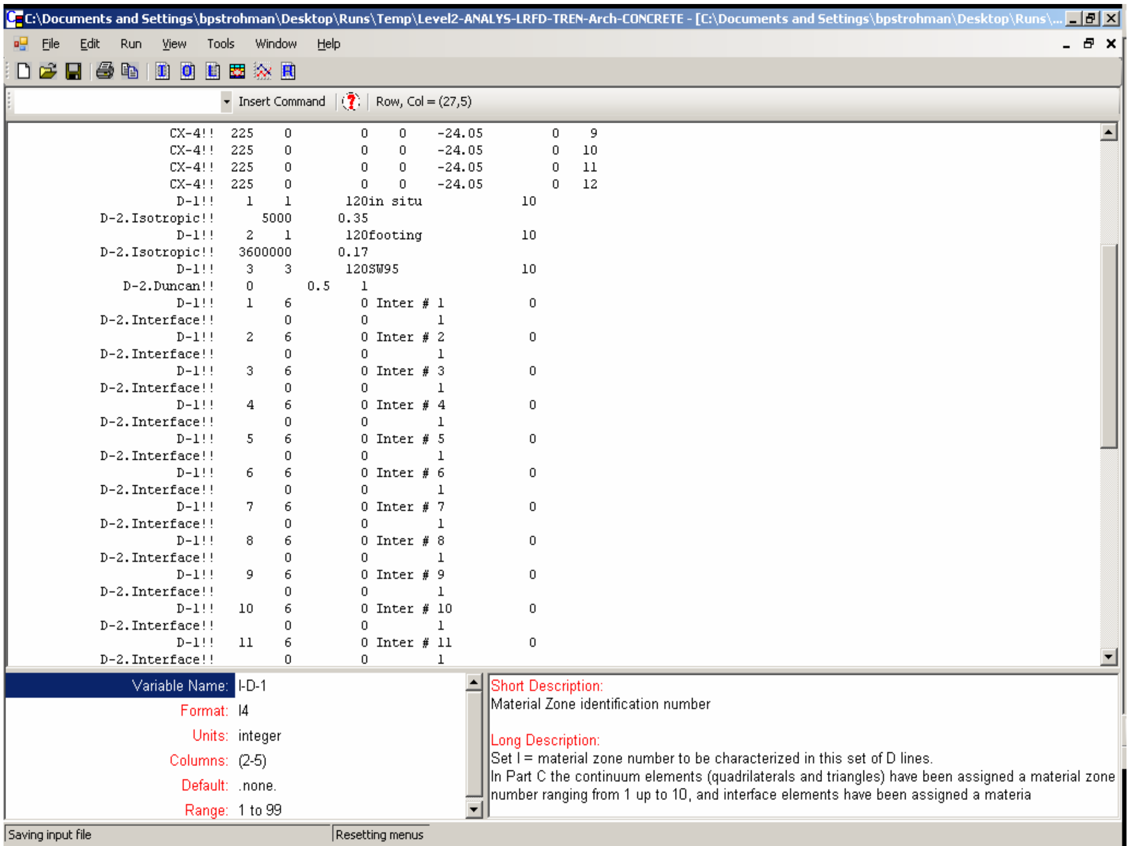


Figure 6-19 – Text Input Screen Before Being Modified

Once in this format, input the interface properties using the shortcut method described in Chapter 5, D-2 of the *User Manual.* In this method, the interface angles are defined at only three points along the radius of the arch: at the crown, at the point where the two radii intersect, and at the last node before the node that defines the footing. These interface angles are computed in the following manner:

Interface angle in the first arch segment, $\theta(i) = 90 - (i-1)*(\Delta /(m-1))$

Interface angle in the second arch segment, $\phi(j) = 90 - \Delta - j*(\delta /n)$

where:

$i = 1, 2, \ldots..m$

$j = 1, 2, \ldots..n$

Δ = angle in degrees in segment 1 = 40°

δ = angle in degrees in segment 2 = 62.87°

m = number of nodes in segment 1 = 13

p = number of nodes in segment 2 = 7

Therefore:

Interface angle at crown, $\theta(1) = 90 - (1\text{-}1)*(40°/(13\text{-}1)) = 90°$

Interface angle at intersection of radii, $\theta(13) = 90 - (13\text{-}1)*(40°/(13\text{-}1)) = 50°$

Interface angle at node number p-1, $\phi(6) = 90 - 40 - j*(62.87/7) = -3.89°$

The modified text input file is shown in Figure 6-20.

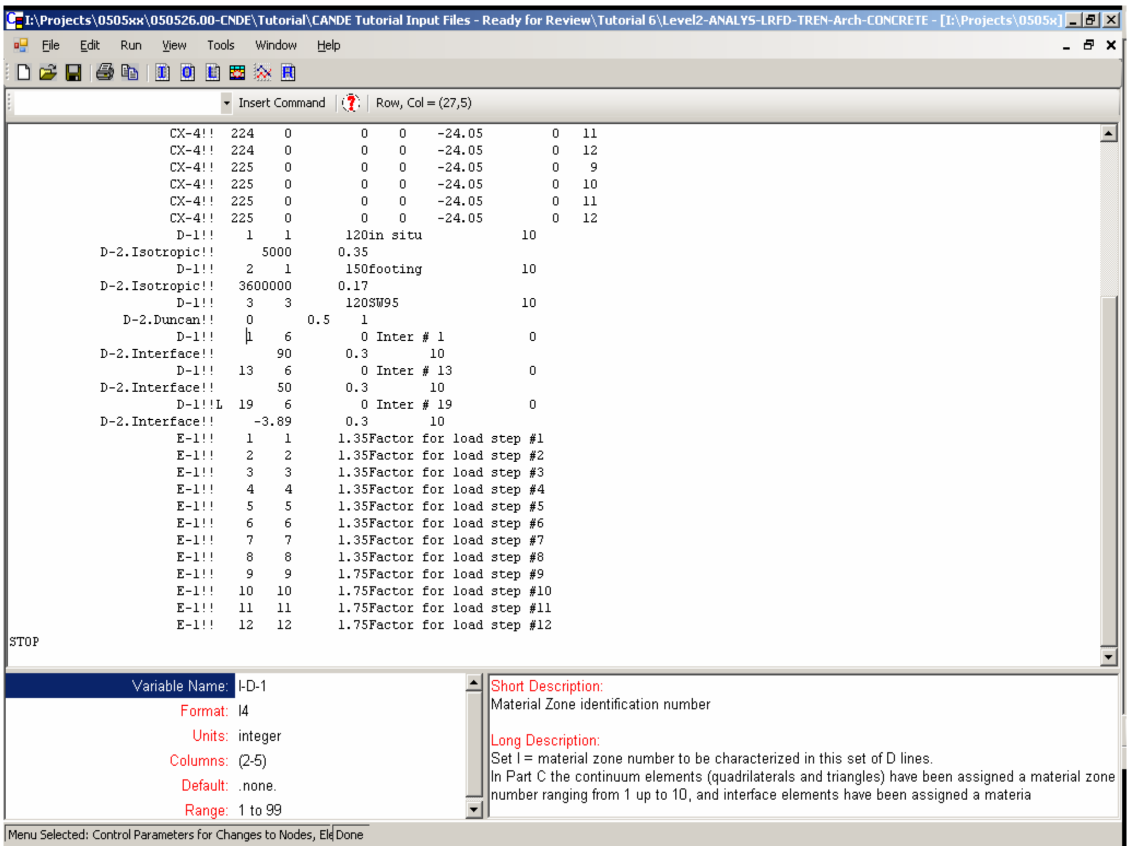


Figure 6-20 – Text Input Screen After Being Modified

After inputting the interface properties in the text input mode, save the file and then reopen the problem as a regular CANDE file. There should now only be three interface material definitions shown on the control panel on the left of the screen as shown in Figure 6-21.

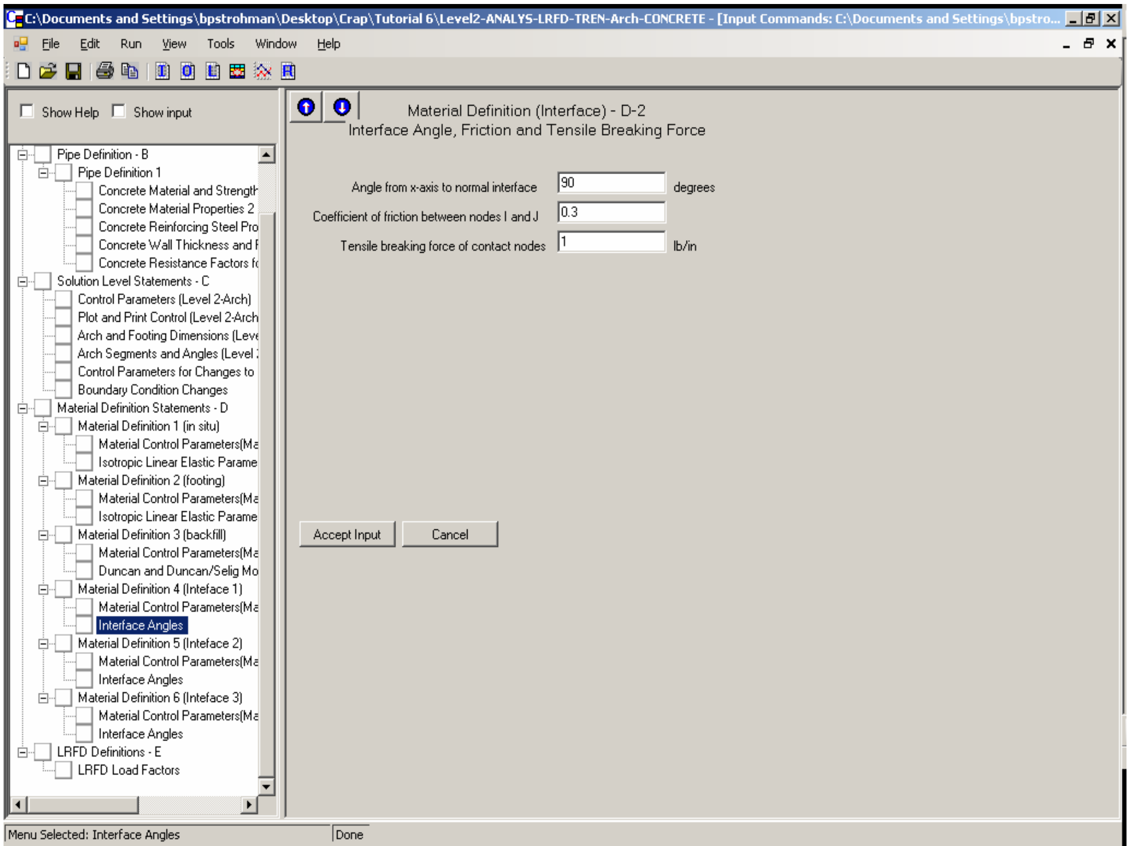


Figure 6-21 – Control Panel After Manually Inputting Interface Properties Using the Shortcut Method (Similar for interface screens 2 and 3)

When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

6.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The "Plotting Parameters" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting parameters menu and click the check box "Show constr. increment numbers." Click "OK," set Load Increment to 8 to show the entire mesh and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 6-22.

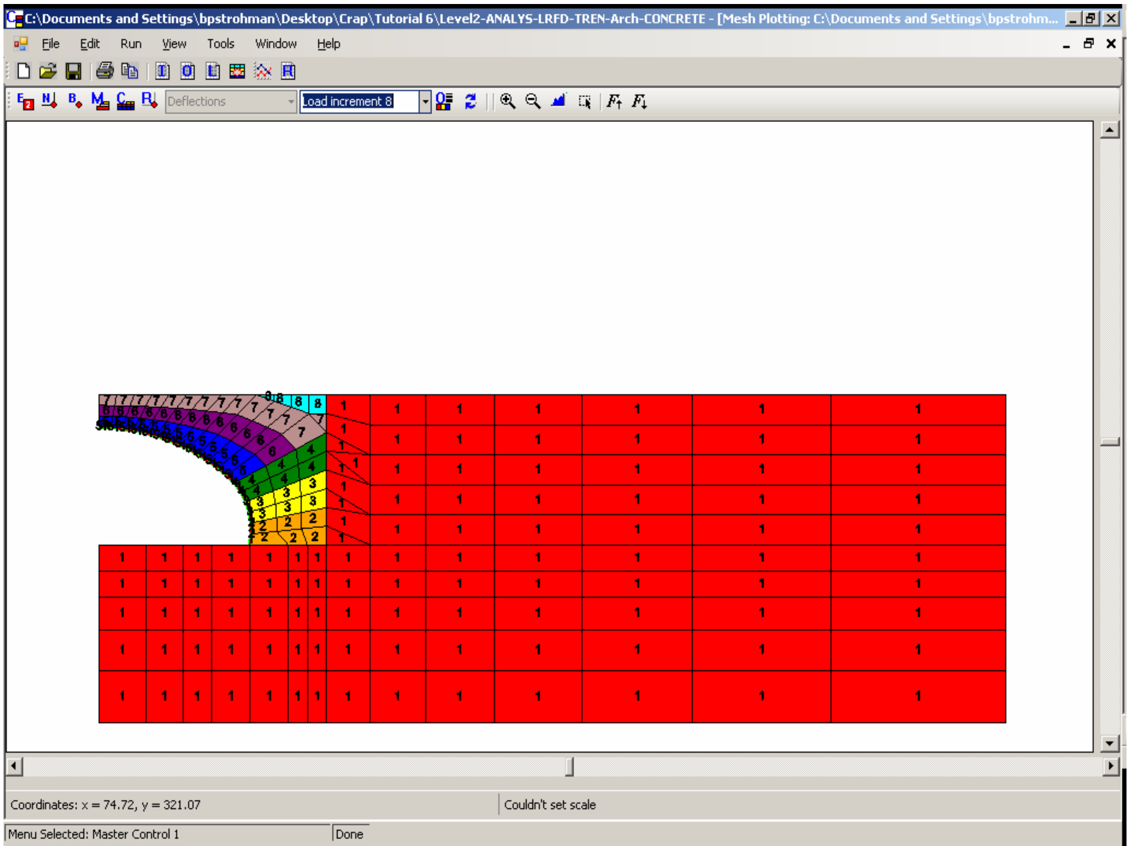


Figure 6-22 – Mesh Plot for Load Steps 1 to 8

To check the boundary conditions, click on the toolbar icon labeled “B” to turn on boundary conditions. Note that the blue dots represent horizontal rollers, the green dots are pins, and the red dots are the applied live load. Set the Load Increment 12 and the mesh plot should look like Figure 6-23.

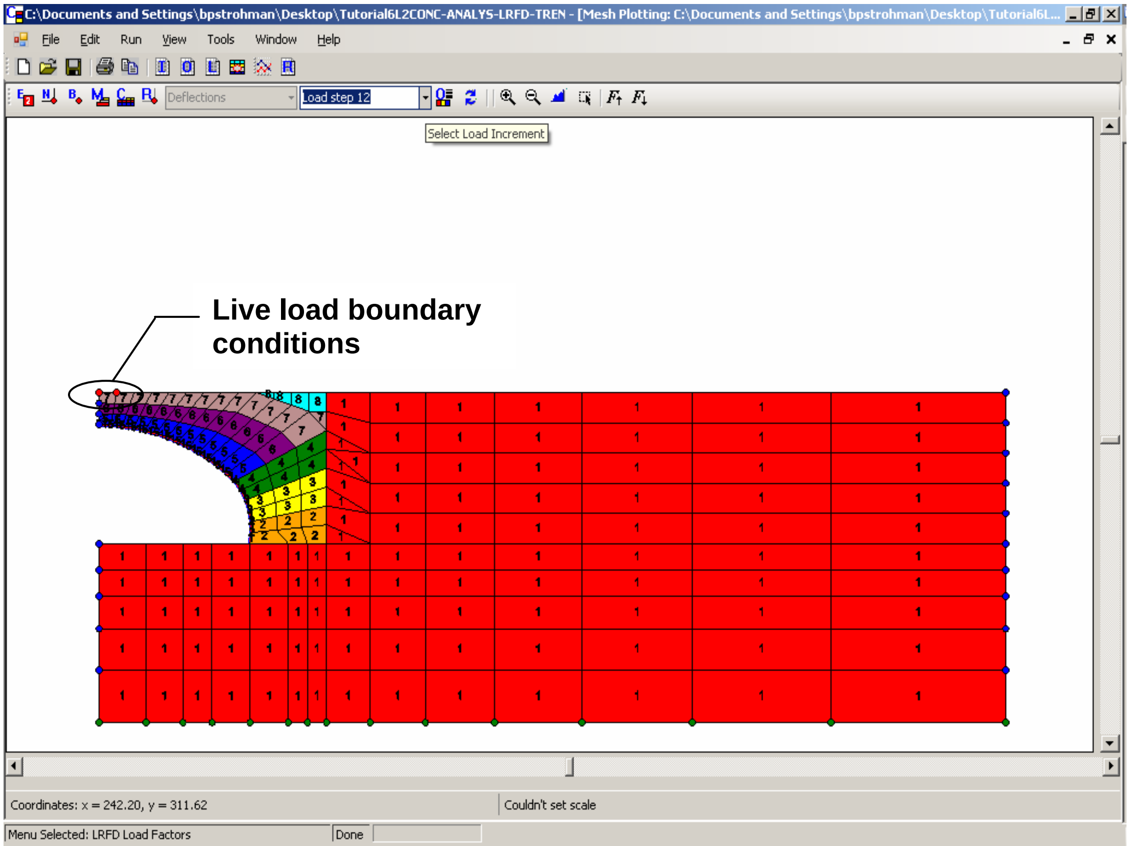


Figure 6-23 – Mesh Plot including Boundary Conditions - Load Step 12

To view vertical strain, open the plotting parameters window and set the deflection magnification factor to 10 (see Figure 6-24).

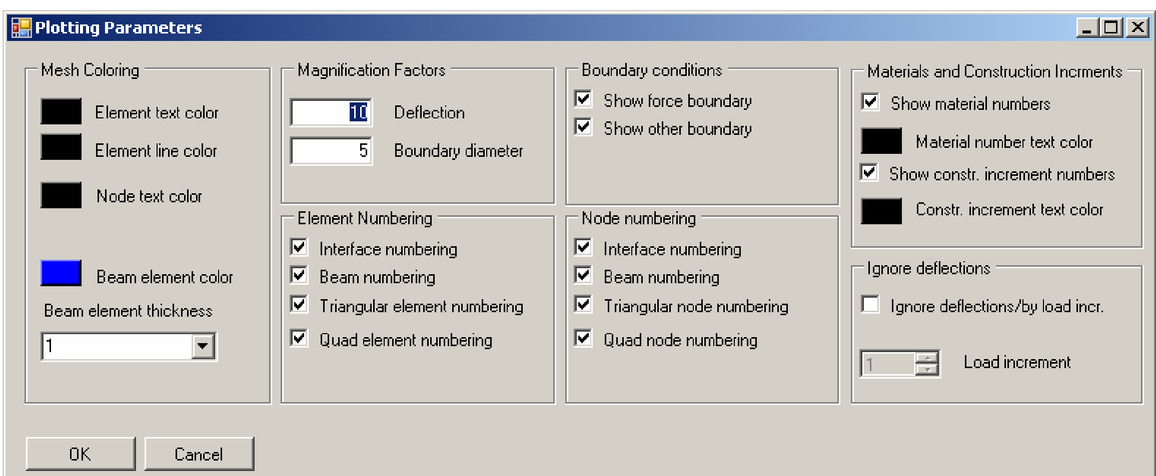


Figure 6-24 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Vertical Strain. Set the increment to 12 and the screen should look like Figure 6-25, which also shows the deflected mesh geometry. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

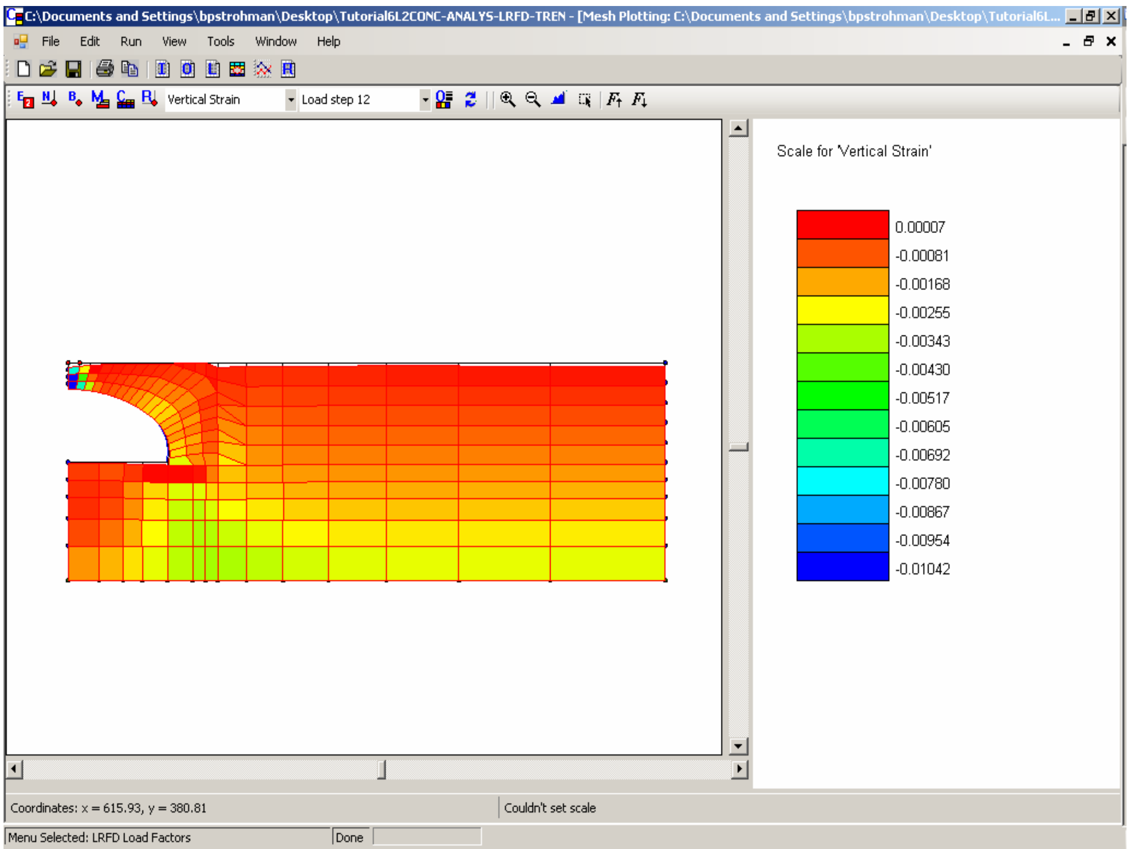


Figure 6-25 – Vertical Strain Load Step 12

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 6-26 compares the bending moments after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 3, 6, and 9, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Bending moment” and the screen should appear as Figure 6-26.

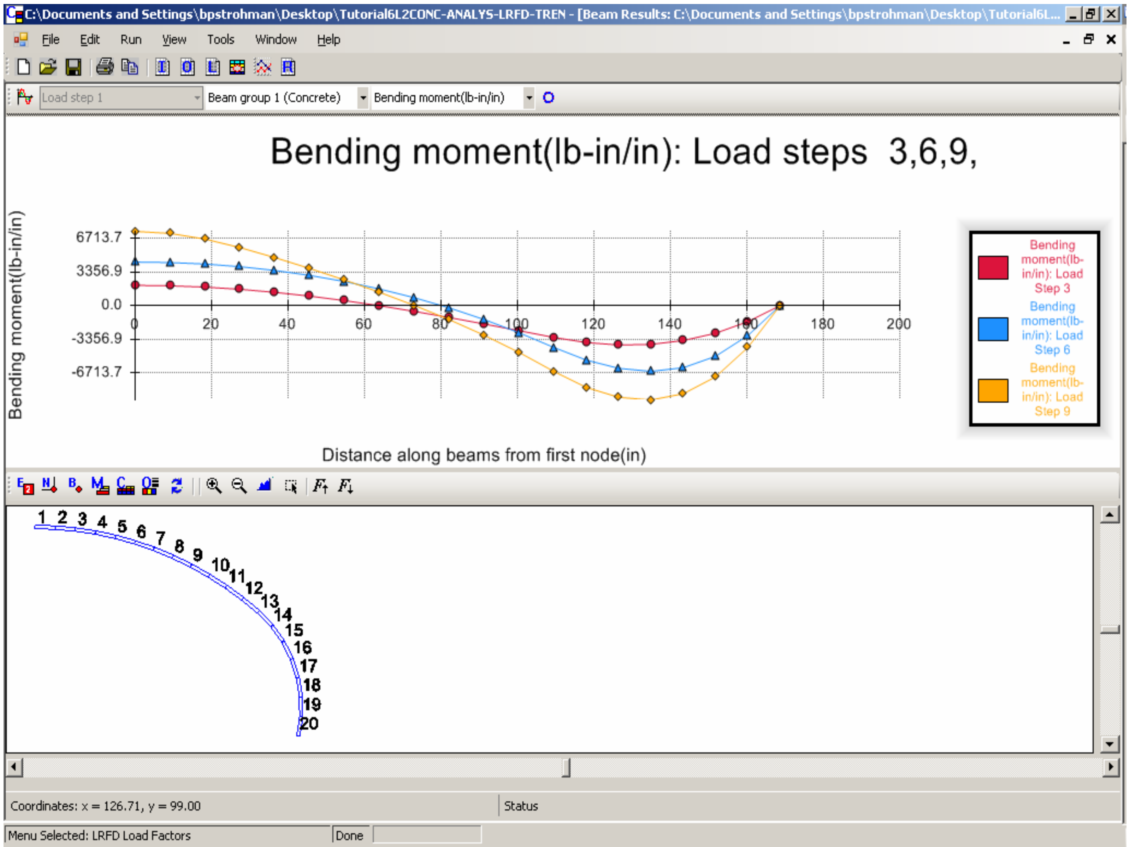


Figure 6-26 – Bending Moments for Load Steps 3, 6, and 9

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 6-27) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 6-27 shows the final assessment printed at the end of the file. Figure 6-28 displays the ultimate beam element forces for the final earth load step, load step 9 (Dead Load Only). Note that information from this figure will be used later in this tutorial in the live load rating calculation.

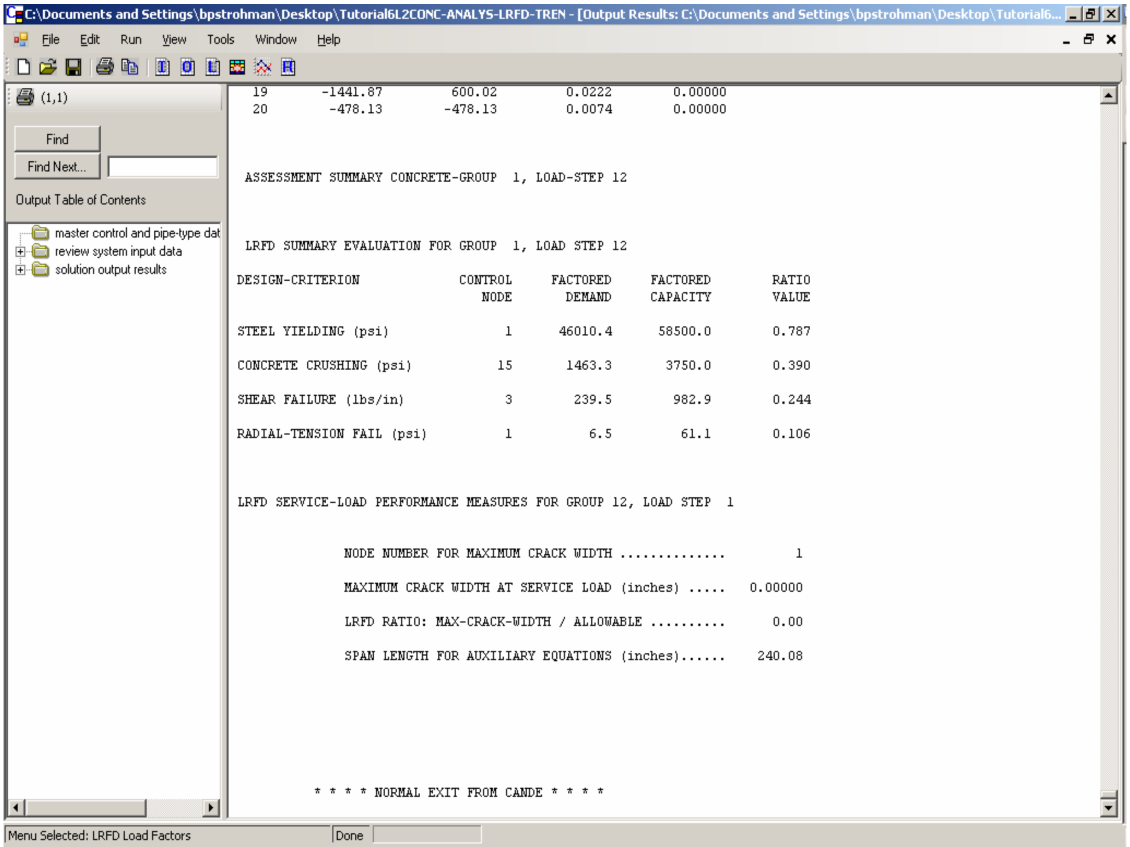
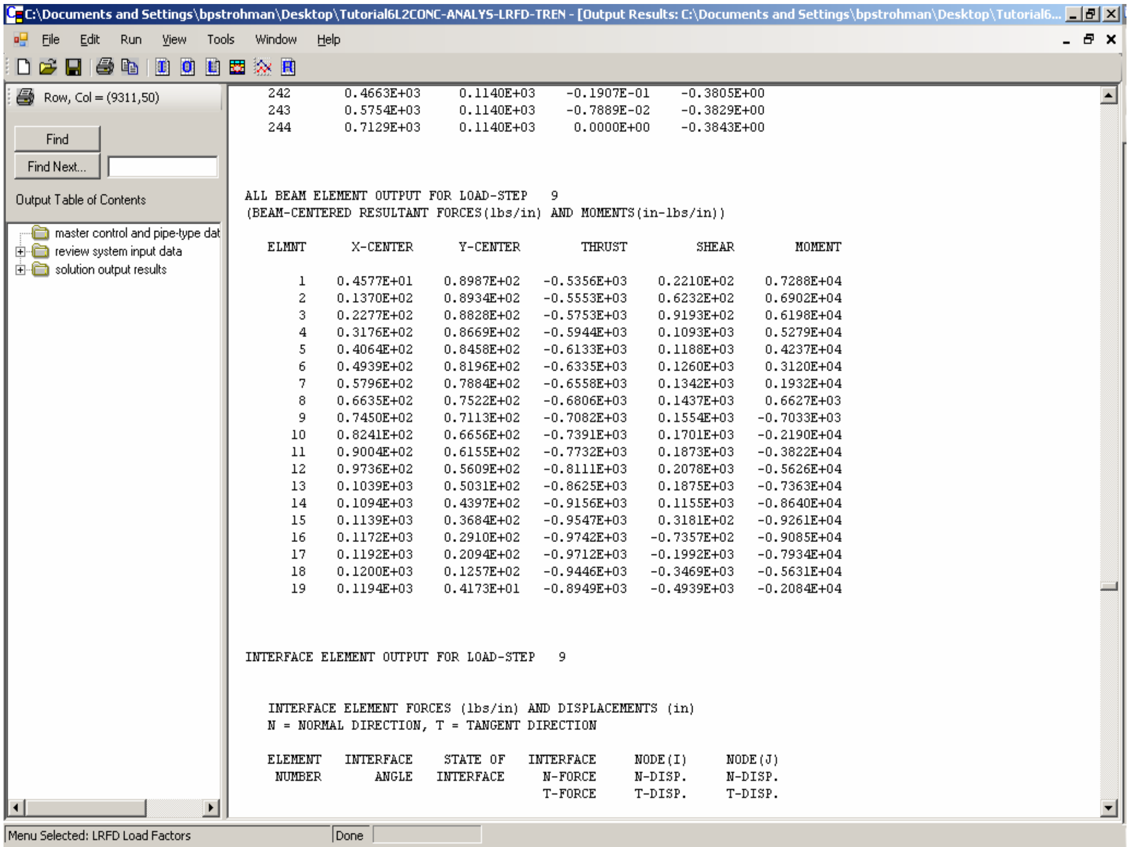


Figure 6-27 – Design Assessment Summary – Load Step 12


C:\Documents and Settings\bpstrohman\Desktop\Tutorial6L2CONC-ANALYS-LRFD-TREN - [Output Results: C:\Documents and Settings\bpstrohman\Desktop\Tutorial6...

File Edit Run View Tools Window Help

Row, Col = (9311,50)

Find

Find Next...

Output Table of Contents

master control and pipe-type dat

review system input data

solution output results

242	0.4663E+03	0.1140E+03	-0.1907E-01	-0.3805E+00
243	0.5754E+03	0.1140E+03	-0.7889E-02	-0.3829E+00
244	0.7129E+03	0.1140E+03	0.0000E+00	-0.3843E+00

ALL BEAM ELEMENT OUTPUT FOR LOAD-STEP 9

(BEAM-CENTERED RESULTANT FORCES(lbs/in) AND MOMENTS(in-lbs/in))

ELMNT	X-CENTER	Y-CENTER	THRUST	SHEAR	MOMENT
1	0.4577E+01	0.8987E+02	-0.5356E+03	0.2210E+02	0.7288E+04
2	0.1370E+02	0.8934E+02	-0.5553E+03	0.6232E+02	0.6902E+04
3	0.2277E+02	0.8828E+02	-0.5753E+03	0.9193E+02	0.6198E+04
4	0.3176E+02	0.8669E+02	-0.5944E+03	0.1093E+03	0.5279E+04
5	0.4064E+02	0.8458E+02	-0.6133E+03	0.1188E+03	0.4237E+04
6	0.4939E+02	0.8196E+02	-0.6335E+03	0.1260E+03	0.3120E+04
7	0.5796E+02	0.7884E+02	-0.6558E+03	0.1342E+03	0.1932E+04
8	0.6635E+02	0.7522E+02	-0.6806E+03	0.1437E+03	0.6627E+03
9	0.7450E+02	0.7113E+02	-0.7082E+03	0.1554E+03	-0.7033E+03
10	0.8241E+02	0.6656E+02	-0.7391E+03	0.1701E+03	-0.2190E+04
11	0.9004E+02	0.6155E+02	-0.7732E+03	0.1873E+03	-0.3822E+04
12	0.9736E+02	0.5609E+02	-0.8111E+03	0.2078E+03	-0.5626E+04
13	0.1039E+03	0.5031E+02	-0.8625E+03	0.1875E+03	-0.7363E+04
14	0.1094E+03	0.4397E+02	-0.9156E+03	0.1155E+03	-0.8640E+04
15	0.1139E+03	0.3684E+02	-0.9547E+03	0.3181E+02	-0.9261E+04
16	0.1172E+03	0.2910E+02	-0.9742E+03	-0.7357E+02	-0.9085E+04
17	0.1192E+03	0.2094E+02	-0.9712E+03	-0.1992E+03	-0.7934E+04
18	0.1200E+03	0.1257E+02	-0.9446E+03	-0.3469E+03	-0.5631E+04
19	0.1194E+03	0.4173E+01	-0.8949E+03	-0.4939E+03	-0.2084E+04

INTERFACE ELEMENT OUTPUT FOR LOAD-STEP 9

INTERFACE ELEMENT FORCES (lbs/in) AND DISPLACEMENTS (in)

N = NORMAL DIRECTION, T = TANGENT DIRECTION

ELEMENT NUMBER	INTERFACE ANGLE	STATE OF INTERFACE	INTERFACE N-FORCE T-FORCE	NODE(I) N-DISP. T-DISP.	NODE(J) N-DISP. T-DISP.

Menu Selected: LRFD Load Factors

Done

Figure 6-28 – Ultimate Beam Element Forces for Load Step 9

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 6-29 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 6-30. The three tabs shown in Figures 6-29 and 6-30 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

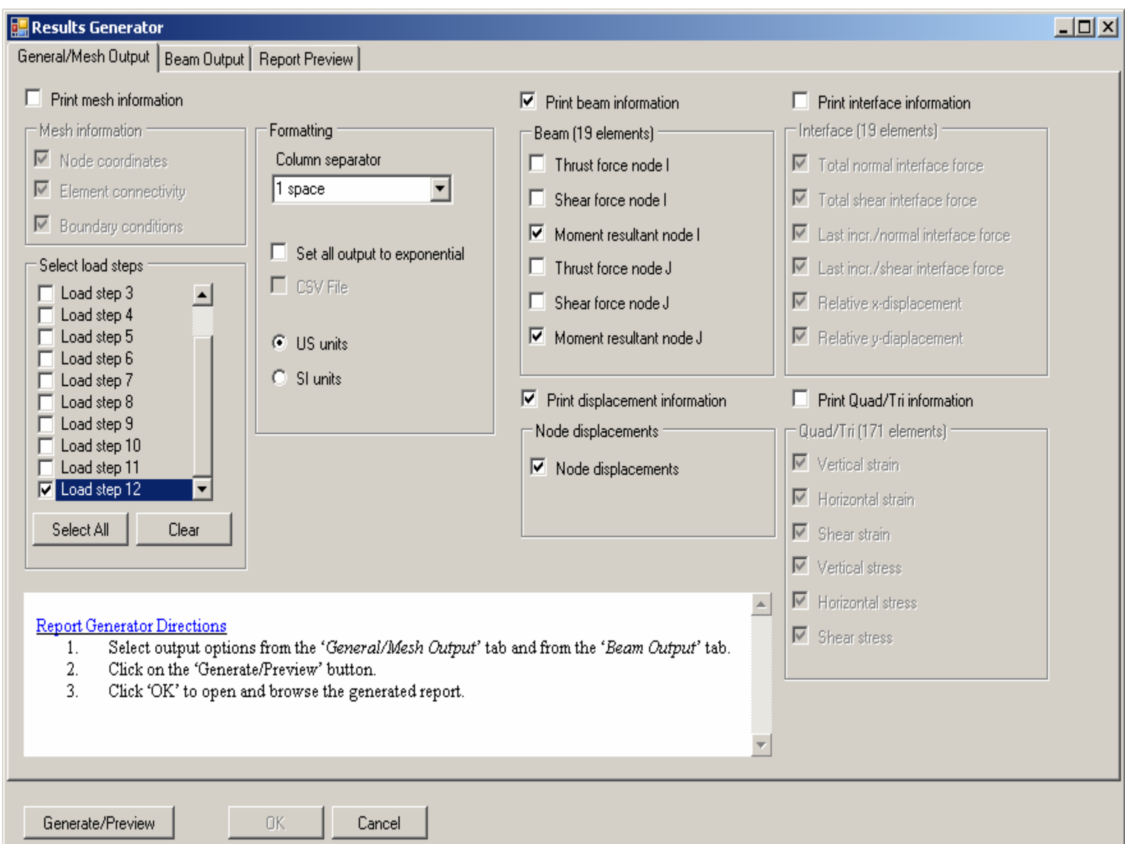


Figure 6-29 – Results Generator Input Screen – Load Step 12 Moments and Deflections

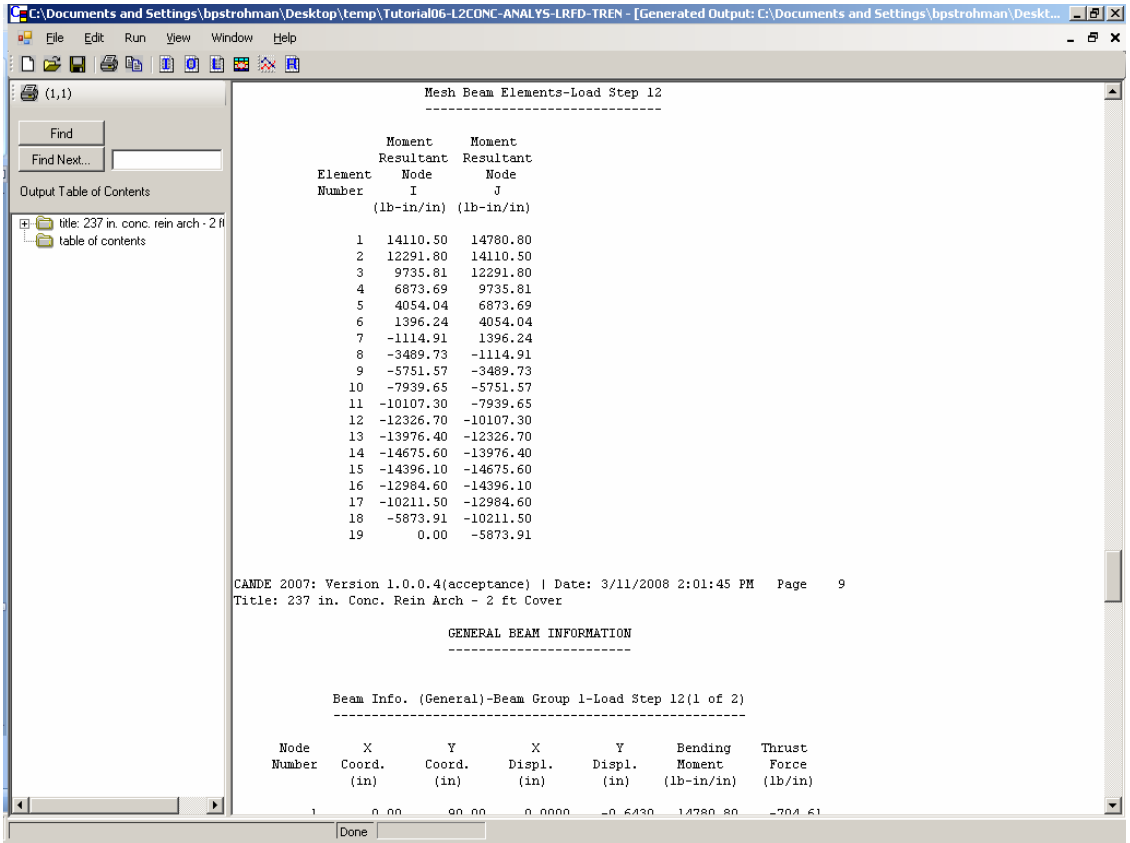


Figure 6-30 – Results Generator – Load Step 12 Moments

Live Load Rating Procedure

After the user has reviewed the results from CANDE, the live load rating can be evaluated using the CANDE output. The live load rating procedure utilizes service loads and therefore, for this problem the user must modify Input Screen E-1 as depicted in Figure 6-31 and rerun the analysis.

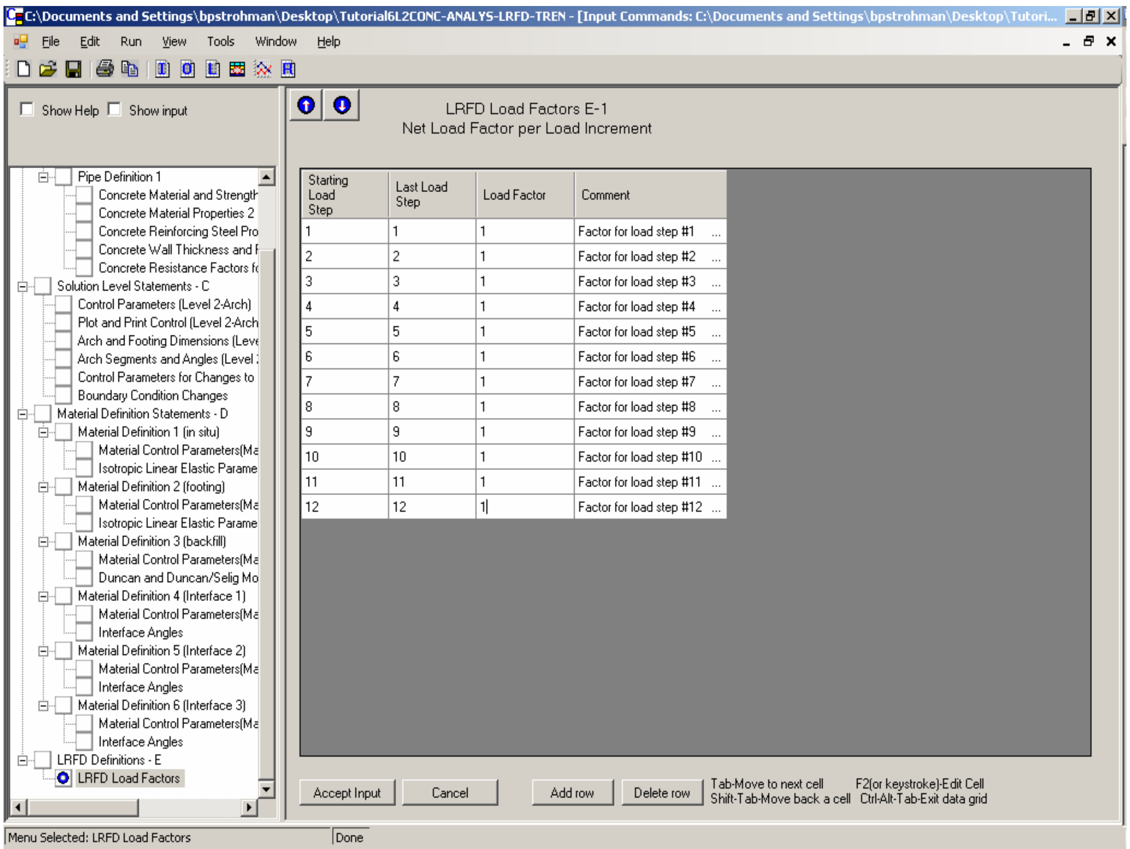


Figure 6-31 – Input Screen E-1, Modified for Live Load Rating Calculation

After modifying input screen E-1, save the file, click "Run" and "CANDE-2007" on the main toolbar to execute the program. This will open the "Running CANDE" window which will allow you to monitor the run as it progresses. When the program is complete, a check box "CANDE Analysis Complete" window will appear. Click "OK" and check if the "Running CANDE" window states "Normal Exit From CANDE." If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the "Close" button in the "Running CANDE" window to return to the user interface.

The live load rating will be determined for the moment and shear forces at the crown of the arch (Beam Element #1). A live load rating calculation requires the service dead load and live load components for each the moment and shear forces.

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file to find the information needed to calculate the live load rating.

The following section describes where in the output file the information needed to calculate the live load rating was extracted (Figure 6-35 depicts the live load rating calculation and details how this extracted data was used). Figure 6-32 displays the service beam element forces for the final earth load step, load step 9 (Dead Load Only), and highlights the values extracted for the live load rating calculation. Figure 6-33 shows the service beam element forces for the final load step, load step 12 (Dead Load + Live Load), and highlights the values extracted for the live load rating calculation. Note Figure 6-28 is shown again below as Figure 6-34 for the purpose of displaying the extracted information for the live load rating calculation. Figure 6-35 shows the calculation of the live rating load rating for the moment and shear forces at the crown of the arch (Beam Element #1).

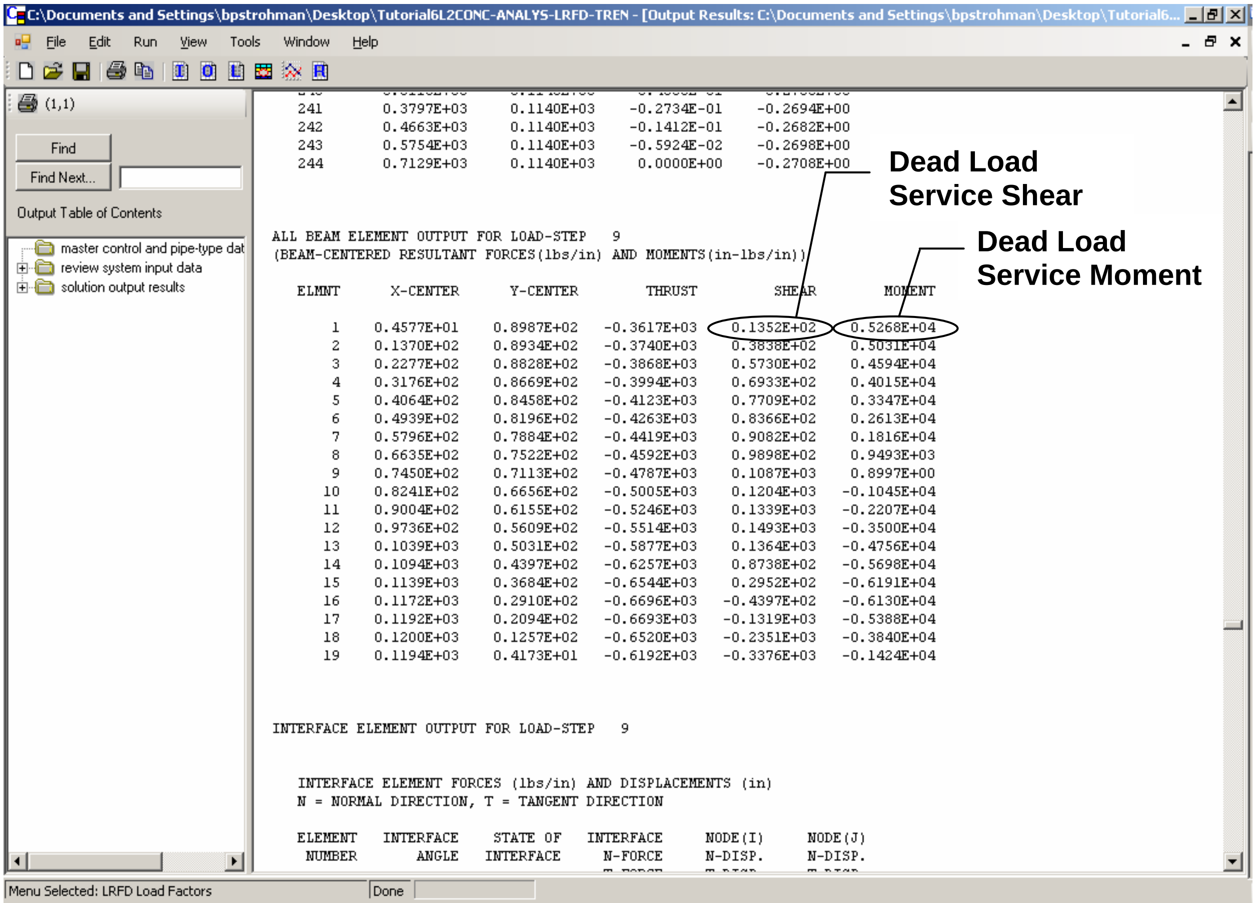


Figure 6-32 – Service Beam Element Forces for Load Step 9, Showing Values Extracted for Live Load Rating Calculation

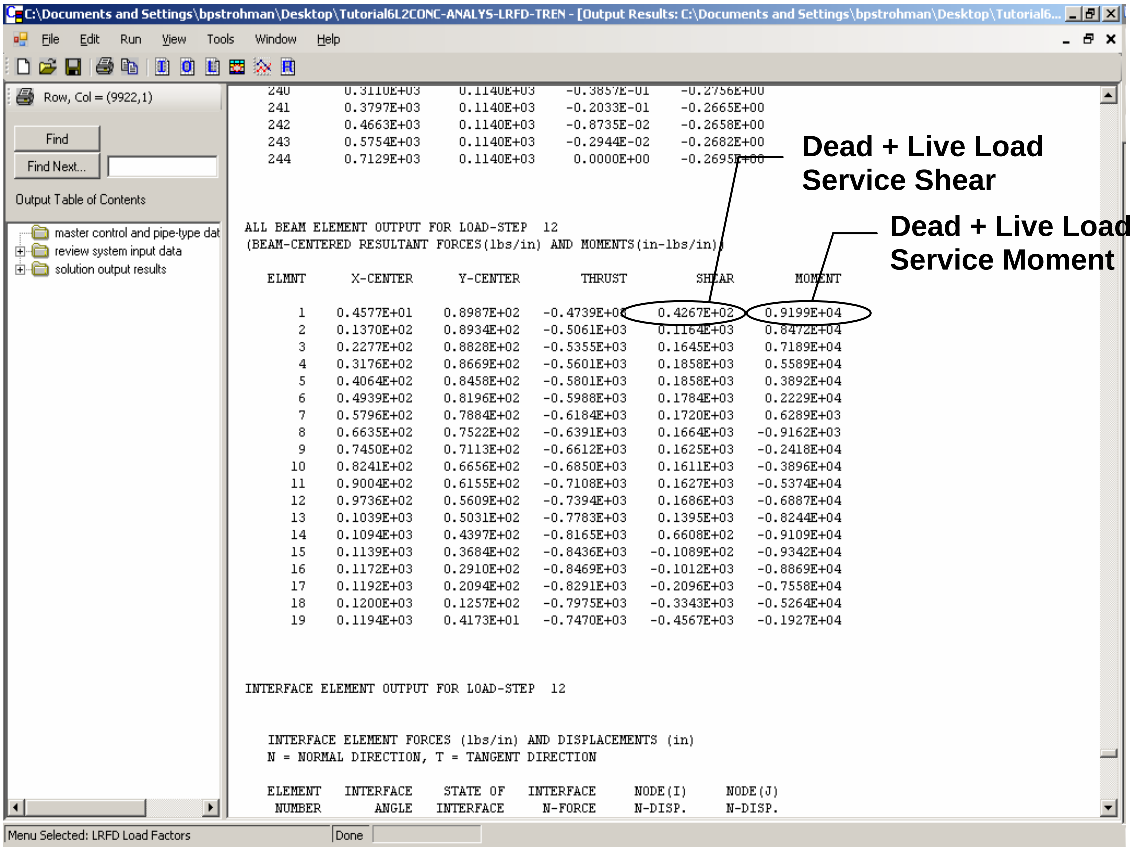
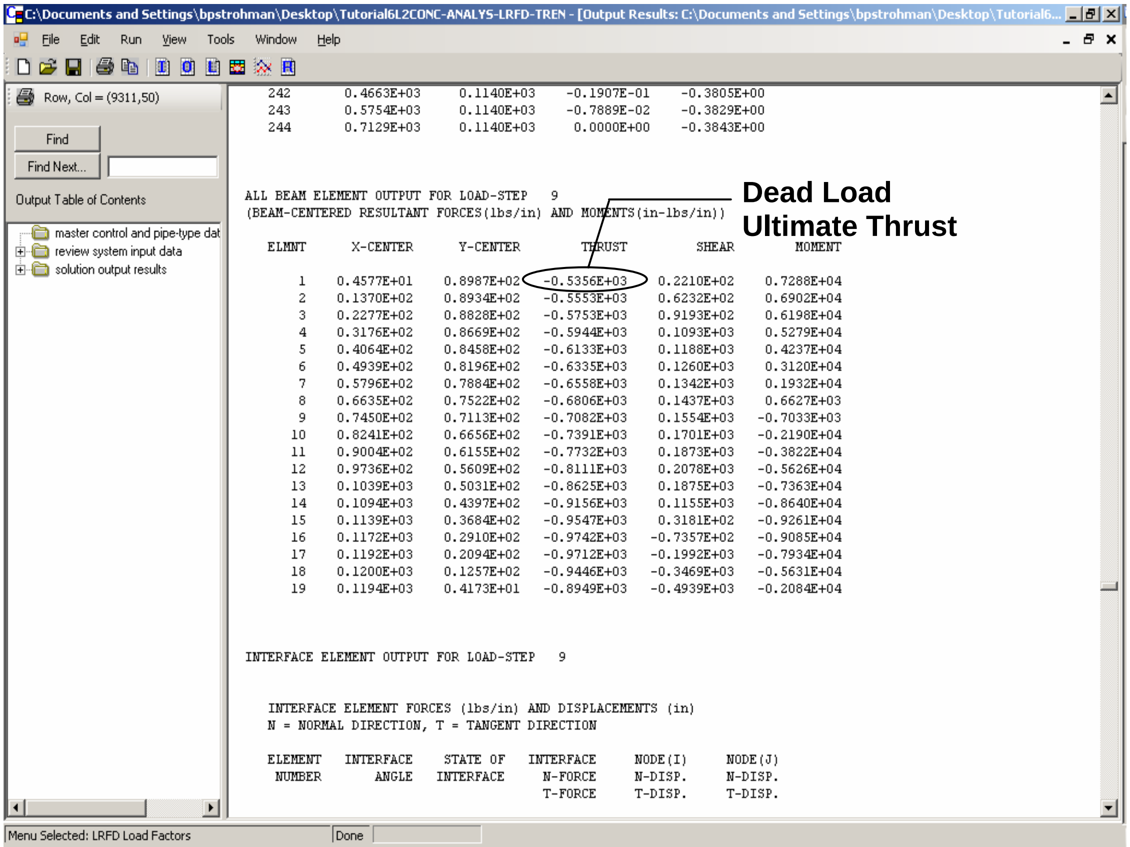


Figure 6-33 – Service Beam Element Forces for Load Step 12, Showing Values Extracted for Live Load Rating Calculation




```
    242      0.4663E+03     0.1140E+03    -0.1907E-01    -0.3805E+00
    243      0.5754E+03     0.1140E+03    -0.7889E-02    -0.3829E+00
    244      0.7129E+03     0.1140E+03     0.0000E+00    -0.3843E+00


 ALL BEAM ELEMENT OUTPUT FOR LOAD-STEP    9
 (BEAM-CENTERED RESULTANT FORCES(lbs/in) AND MOMENTS(in-lbs/in))

    ELMNT      X-CENTER      Y-CENTER       THRUST        SHEAR       MOMENT

        1    0.4577E+01    0.8987E+02   -0.5356E+03   0.2210E+02   0.7288E+04
        2    0.1370E+02    0.8934E+02   -0.5553E+03   0.6232E+02   0.6902E+04
        3    0.2277E+02    0.8828E+02   -0.5753E+03   0.9193E+02   0.6198E+04
        4    0.3176E+02    0.8669E+02   -0.5944E+03   0.1093E+03   0.5279E+04
        5    0.4064E+02    0.8458E+02   -0.6133E+03   0.1188E+03   0.4237E+04
        6    0.4939E+02    0.8196E+02   -0.6335E+03   0.1260E+03   0.3120E+04
        7    0.5796E+02    0.7884E+02   -0.6558E+03   0.1342E+03   0.1932E+04
        8    0.6635E+02    0.7522E+02   -0.6806E+03   0.1437E+03   0.6627E+03
        9    0.7450E+02    0.7113E+02   -0.7082E+03   0.1554E+03  -0.7033E+03
       10    0.8241E+02    0.6656E+02   -0.7391E+03   0.1701E+03  -0.2190E+04
       11    0.9004E+02    0.6155E+02   -0.7732E+03   0.1873E+03  -0.3822E+04
       12    0.9736E+02    0.5609E+02   -0.8111E+03   0.2078E+03  -0.5626E+04
       13    0.1039E+03    0.5031E+02   -0.8625E+03   0.1875E+03  -0.7363E+04
       14    0.1094E+03    0.4397E+02   -0.9156E+03   0.1155E+03  -0.8640E+04
       15    0.1139E+03    0.3684E+02   -0.9547E+03   0.3181E+02  -0.9261E+04
       16    0.1172E+03    0.2910E+02   -0.9742E+03  -0.7357E+02  -0.9085E+04
       17    0.1192E+03    0.2094E+02   -0.9712E+03  -0.1992E+03  -0.7934E+04
       18    0.1200E+03    0.1257E+02   -0.9446E+03  -0.3469E+03  -0.5631E+04
       19    0.1194E+03    0.4173E+01   -0.8949E+03  -0.4939E+03  -0.2084E+04



 INTERFACE ELEMENT OUTPUT FOR LOAD-STEP    9


    INTERFACE ELEMENT FORCES (lbs/in) AND DISPLACEMENTS (in)
    N = NORMAL DIRECTION, T = TANGENT DIRECTION

    ELEMENT   INTERFACE    STATE OF   INTERFACE     NODE(I)     NODE(J)
     NUMBER       ANGLE   INTERFACE     N-FORCE     N-DISP.     N-DISP.
                                        T-FORCE     T-DISP.     T-DISP.
```

Figure 6-34 – Ultimate Beam Element Forces for Load Step 9 (same as Figure 6-28, Except Showing Values Extracted for Live Load Rating Calculation)

Load Rating Calculation

LOAD RATING - AASHTO MANUAL FOR CONDITION EVALUATION OF BRIDGES

Properties

The following template shows an example load rating calculation for the reinforced concrete arch in Tutorial Problem 6. The Loads used for the load rating are the AASHTO HS-20: Single Unit Truck.

Note: All references apply to AASHTO 2000 Manual for Condition Evaluation of Bridges Second Edition unless otherwise noted.

NOTES

1. Input values shown in grey.
2. := indicates definition of a variable.
3. = prints a previously defined variable.

Width of section being designed (not wall thickness)	$b \equiv 12 \cdot in \cdot ft^{-1}$	
Design compressive strength of concrete	$f_{cp} \equiv 5 \cdot ksi$	
Yield strength of steel reinforcement	$f_y := 65 \cdot ksi$	
Load factor for dead loads	$A_1 := 1.3$	Section 6.5.3
Load Factor for live load inventory loads	$A_{2\_inv} := 2.17$	Section 6.5.3
Load factor for live load operating loads	$A_{2\_oper} := 1.3$	Section 6.5.3
Load factor for live load legal loads at operating level:	$A_{2\_Legal\_oper} := 1.3$	Section 6.5.3
Resistance factor for flexure	$\phi := 1.00$	AASHTO 17th ED. Section 16.4.6.3
Resistance factor for shear	$\phi_v := 0.90$	
Arch crown radius	$r_t := 157 \cdot in$	

Design Data at Example Section

Moment and shear load rating factors are determined at the arch crown (beam element #1) for these example calculations.

Dead Load Service Moment	$M_{DC} := 5268 \frac{lbf \cdot in}{in}$	(From Figure 6-32, Beam Element #1)
Dead Load and Live Load Service Moment	$M_{DC\_LL} := 9199 \frac{lbf \cdot in}{in}$	(From Figure 6-33, Beam Element #1)
HS-20 Live Load Service Moment	$M_{LL\_HS20} := M_{DC\_LL} - M_{DC}$	
	$M_{LL\_HS20} = 3931 \frac{lbf \cdot in}{in}$	
Dead Load Service Shear	$V_{DC} := 13.52 \frac{lbf}{in}$	(From Figure 6-32, Beam Element #1)
Dead Load and Live Load Shear	$V_{DC\_LL} := 42.67 \frac{lbf}{in}$	(From Figure 6-33, Beam Element #1)
HS-20 Live Load Service Shear	$V_{LL\_HS20} := V_{DC\_LL} - V_{DC}$	
	$V_{LL\_HS20} = 29.15 \frac{lbf}{in}$	

Figure 6-35 – Live Load Rating Calculation

Ultimate Dead Load Thrust $N_{u1} := 535.6 \frac{lbf}{in}$ (For Moment and Shear Calculations, From Figure 6-35, Beam Element #1)

Inside Steel Area at Arch Crown, A1 (For Moment Calculations) $A_{sf\_A1} := 0.022 \frac{in^2}{in}$ (All Reinforcing Data From Model Inputs)

Depth of Inside Steel From Outside Face of Arch $d := 14.7in$

Element Thickness at Arch Crown $h := 16in$

Outside Steel Area at Arch Crown, A3 (For Shear Calculations) $A_3 := 0.022 \frac{in^2}{in}$

Depth of Outside Steel From Inside Face of Arch $d_2 := 14.7in$

Compute Nominal Moment Resistance: Strength Limit State I (AASHTO 17th ED 16.4.6.6)

$$\beta_1 := if\left[f_{cp} < 4 \cdot ksi, 0.85, if\left[f_{cp} > 8ksi, 0.65, 0.85 - 0.05 \cdot \left(\frac{f_{cp}}{ksi} - 4\right)\right]\right]$$

$$g_1 := f_{cp} \cdot b \cdot \beta_1 \qquad g_1 = 4\,ksi$$

Nominal Moment Resistance: A1 steel section

$$M_u := \frac{1}{2} \cdot \frac{g_1 \cdot N_{u1} \cdot h - A_{sf\_A1}^{\ 2} \cdot f_y^{\ 2} + 2 \cdot A_{sf\_A1} \cdot f_y \cdot \phi \cdot g_1 \cdot d - 2 \cdot A_{sf\_A1} \cdot f_y \cdot N_{u1} - N_{u1}^{\ 2}}{g_1} \qquad M_u = 297.874 \frac{kip \cdot in}{ft}$$

$$M_n := \frac{M_u}{\phi} \qquad M_n = 297.9 \frac{kip \cdot in}{ft}$$

Design Load Rating

Design Inventory Rating Factor

$$RF_{design\_inv} := \frac{\phi \cdot M_n - A_1 \cdot M_{DC}}{A_{2\_inv} \cdot M_{LL\_HS20}} \qquad RF_{design\_inv} = 2.11$$

Design Operating Rating Factor

$$RF_{design\_oper} := \frac{\phi \cdot M_n - A_1 \cdot M_{DC}}{A_{2\_oper} \cdot M_{LL\_HS20}} \qquad RF_{design\_oper} = 3.52$$

Compute Nominal Shear Resistance: Strength Limit State I (AASHTO 12.10.4.2.5)

Reinforcement ratio at A3 Steel $\rho_{prov} := \frac{A_3}{b \cdot d_2}$ $\rho_{prov} = 0.001$

Factored Moment at A3 Steel $M_{u1} := 1371 \frac{in \cdot kip}{ft}$

Factored Shear at A3 Steel $V_u := 15.57 \frac{kip}{ft}$

Figure 6-35 – Live Load Rating Calculation (continued)

Moment for $^M/_{Vd}$ ratio $M_{nu} := \max\left[\left[M_{u1} - N_{u1}\cdot\left(\frac{4\cdot h - d_2}{8}\right)\right], \left(0\cdot\frac{in\cdot kip}{ft}\right)\right]$ $M_{nu} = 1\times 10^3\,\frac{in\cdot kip}{ft}$

$^M/_{Vd}$ ratio $MVD := \min\left(\frac{M_{nu}}{V_u\cdot d_2}, 3\right)$

Shear capacity at A3 Steel

$$V_{u\_2} := \min\left[\phi_v\cdot b\cdot d_2\cdot\sqrt{f_{cp}\cdot psi}\cdot\left(1.1 + 63\cdot\rho_{prov}\right)\cdot\frac{\left(0.8 + \frac{1.6\cdot in}{d_2}\right)\cdot\left[1 + \frac{N_{u1} \div \left(kip\cdot ft^{-1}\right)}{24\cdot h \div in}\right]}{1 + \frac{d_2}{2\cdot r_t + 0.5\cdot h}}\cdot\frac{4}{MVD + 1}, 4.5\cdot\sqrt{f_{cp}\cdot psi}\cdot d_2\right]$$

Nominal Shear Resistance $V_n := \frac{V_{u\_2}}{\phi_v}$ $V_n = 13.2\frac{kip}{ft}$

Design Load Rating

Design Inventory Rating Factor

$RF_{Vdesign\_inv} := \frac{\phi_v\cdot V_n - A_1\cdot V_{DC}}{A_{2\_inv}\cdot V_{LL\_HS20}}$ $RF_{Vdesign\_inv} = 15.3$

Design Operating Rating Factor

$RF_{Vdesign\_oper} := \frac{\phi_v\cdot V_n - A_1\cdot V_{DC}}{A_{2\_oper}\cdot V_{LL\_HS20}}$ $RF_{Vdesign\_oper} = 25.6$

Load Information

Design Load

Weight of HS-20 Design Truck $W_{HS20} := 36ton$

Example Calculation of the Load Rating Summary

Note that this is not the load rating of the structure. This is only an example calculation based on the the calculations shown for sections at the crown and the haunch.

Load Rating Summary

$Inventory_{HS} := W_{HS20}\cdot\min\left(RF_{design\_inv}, RF_{Vdesign\_inv}\right)$ $Inventory_{HS} = 75.9ton$

$Operating_{HS} := W_{HS20}\cdot\min\left(RF_{design\_oper}, RF_{Vdesign\_oper}\right)$ $Operating_{HS} = 126.6ton$

Figure 6-35 – Live Load Rating Calculation (continued)

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 7

May 2008

This page intentionally left blank

7. CANDE TEST PROBLEM 7

7.1 Problem Definition

Analyze a 120 in. x 84 in. reinforced concrete box culvert with standard ASTM steel placement with 2 ft of fill over the top of the culvert using LRFD analysis. The problem is shown schematically in Figure 7-1. The analysis will be with Level 2, using an automated finite element box mesh for an embankment installation.

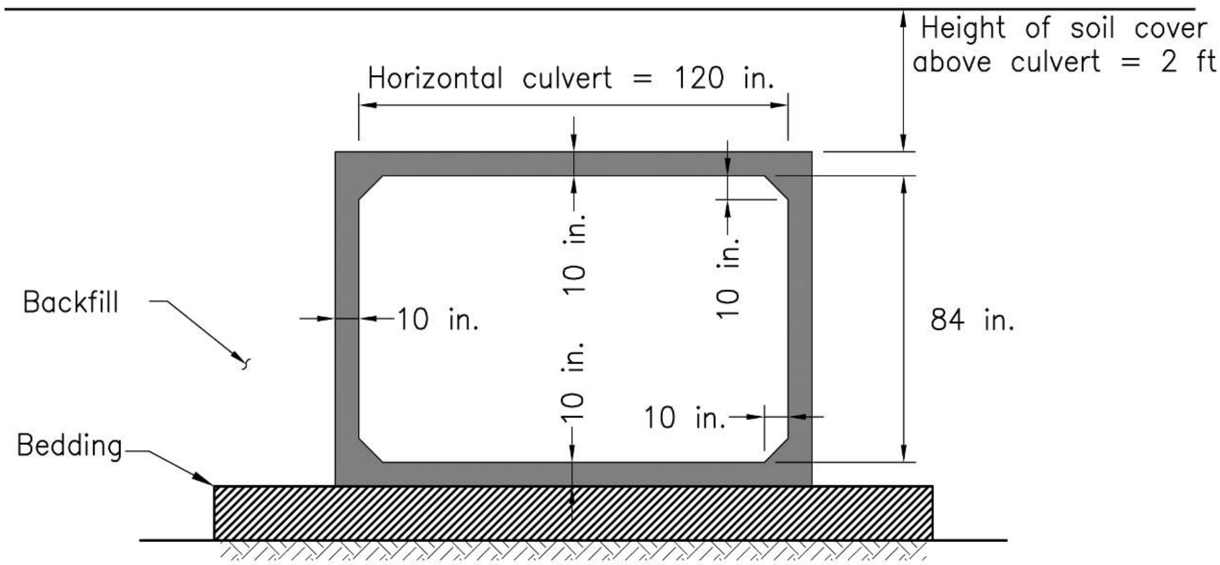


Figure 7-1 Details of Problem 7

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - LRFD

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Box mesh

Soil mesh pattern - Embankment

Pipe material type - Concrete

Reinforcement shape - Boxes

Soil parameters - Canned Duncan/Selig soil models except as noted:

In situ soil - Canned Isotropic-Linear Elastic, Young's modulus = 2,500 psi, Poisson's ratio = 0.25

Bedding soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Backfill soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils

Compressive strength of concrete (f'c) - 5,000 lb/in.^{2}

Shear strength equation - Boxes/3-sided < 2 ft fill (AASHTO 5.13.3.6)

Concrete strain at tension rupture - 0.0 in./in.

Note: Assuming zero tensile strength for the concrete assures the design will be for a cracked section.

Compressive strain at end of elastic range - 0.001 in./in.

Compressive strain at the initial strength limit - 0.002 in./in.

Unit weight of concrete for body weight - 150 lb/ft^3

Crack width model - Gergely-Lutz

Analysis Mode - Small deformation

Yield stress of reinforcing steel - 65,000 lb/in.^{2}

Inner surface spacing between rows of rebar - 2 in.

Outer surface spacing between rows of rebar - 2 in.

Number of inner cage layers of reinforcement - 1

Number of outer cage layers of reinforcement - 1

Type of reinforcement - Deformed wire

Nonlinear material behavior - Option 3 plus steel yielding behavior

Nominal concrete wall thickness - 10 in.

Top / side / bottom slab concrete thicknesses - 10 in.

Vertical / horizontal haunch dimensions - 10 in.

Steel area for outer sidewalls and connecting slabs - 0.0208 in.^{2}/in.

Steel area for inner wall of top slab - 0.0458 in.^{2}/in.

Steel area for inner wall of bottom slab - 0.0242 in.^{2}/in.

Steel area for inner wall of side slabs - 0.02 in.^{2}/in.

Ratio of AS1 length to one-half the span length - 1.0

Uniform concrete cover thickness to all steel centers - 1.5 in.

Number of construction steps - See *User Manual*, Chapter 5, C-2 – The default mesh provides 4 construction steps to the top of the box culvert. For the 2 foot height of soil cover above the box, use 2 additional construction steps, making a total of 6.

One half of horizontal span - Horizontal distance from the vertical centerline of the box culvert to the vertical centerline of the wall = 120 in. / 2 + 10 in. / 2 = 65 in.

One half of vertical rise - Vertical distance from the horizontal centerline of the box culvert to the horizontal centerline of the wall = 84 in. / 2 + 10 in. / 2 = 47 in.

Height of soil cover above box - 2 ft

Density of soil above truncated mesh - 120 lb/ft^3

Bedding depth - 12 in.

LRFD load factor - 1.35

Load modifier - 1.05 (non-redundant for earth load)

7.2 Creating the CANDE Input Document

Figures 7-2 through 7-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking "Next" on Screen 3, CANDE will display a screen indicating that an input path is initiated - click "Finish." The user is prompted for a filename and directory.

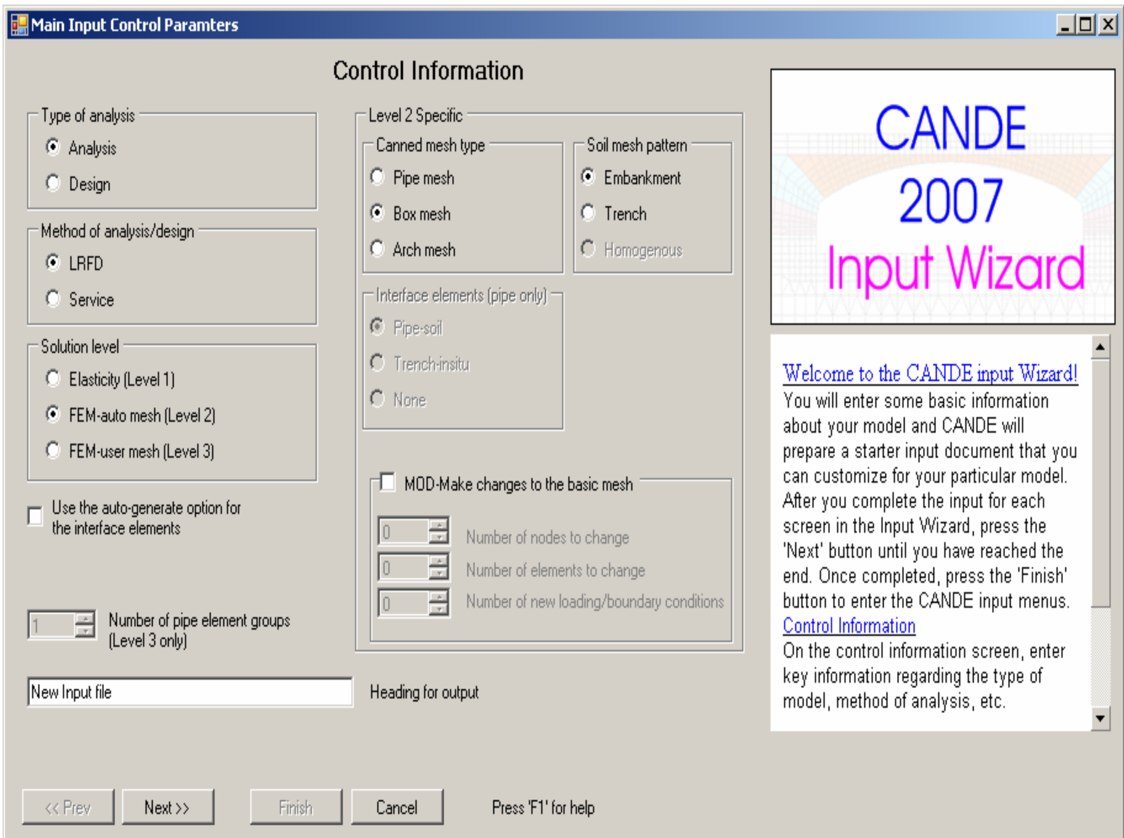


Figure 7-2 – Input Wizard, Screen 1

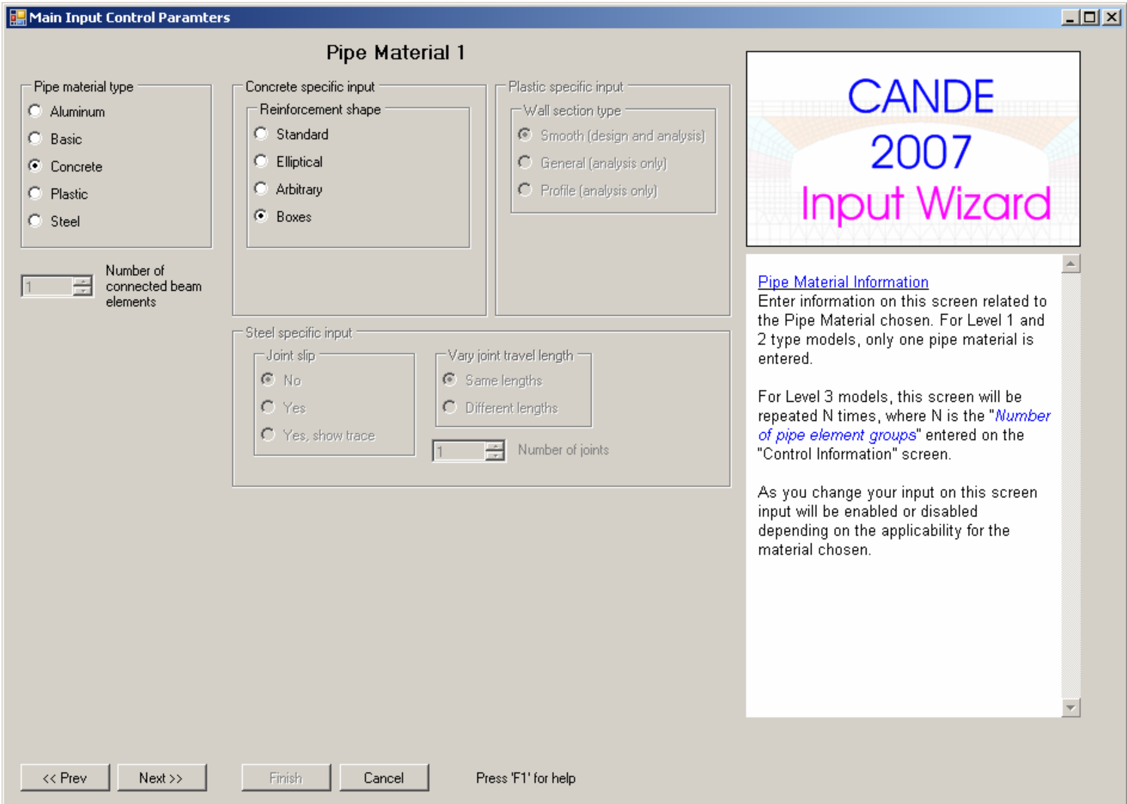


Figure 7-3 – Input Wizard, Screen 2

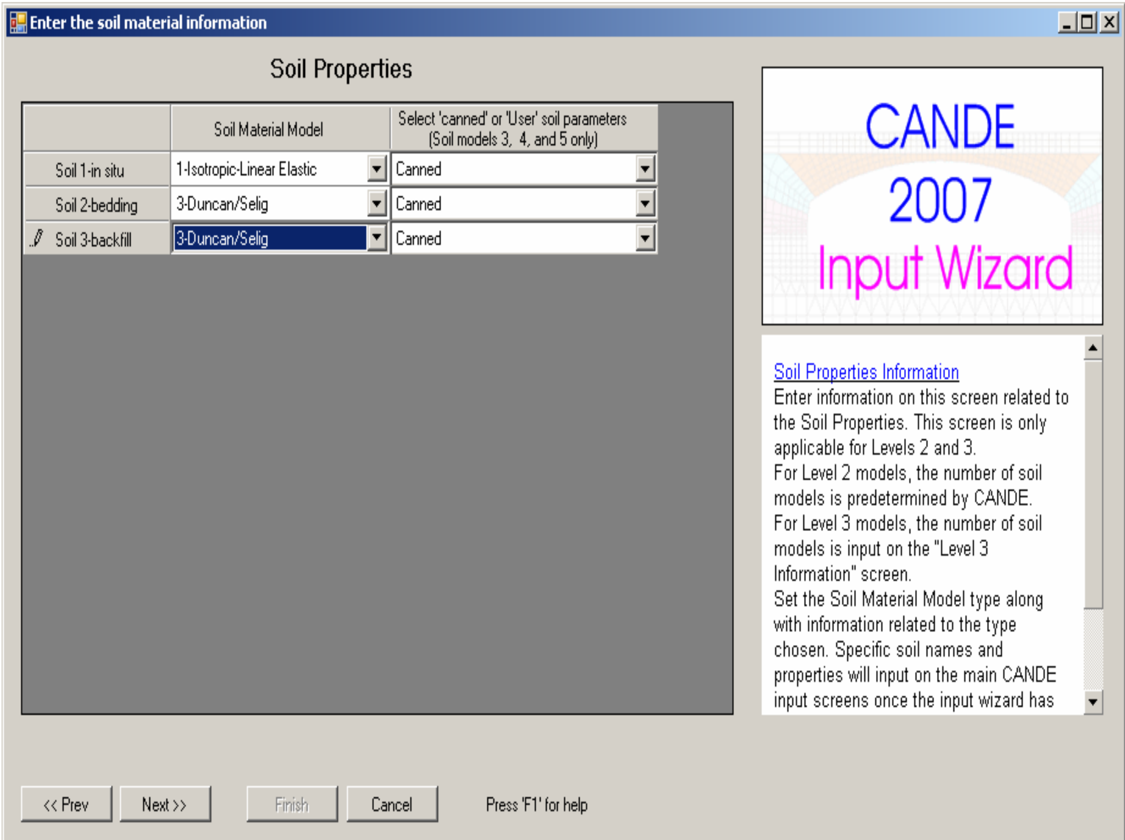


Figure 7-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 7-5. Enter an appropriate heading for output and click “Accept Input.”

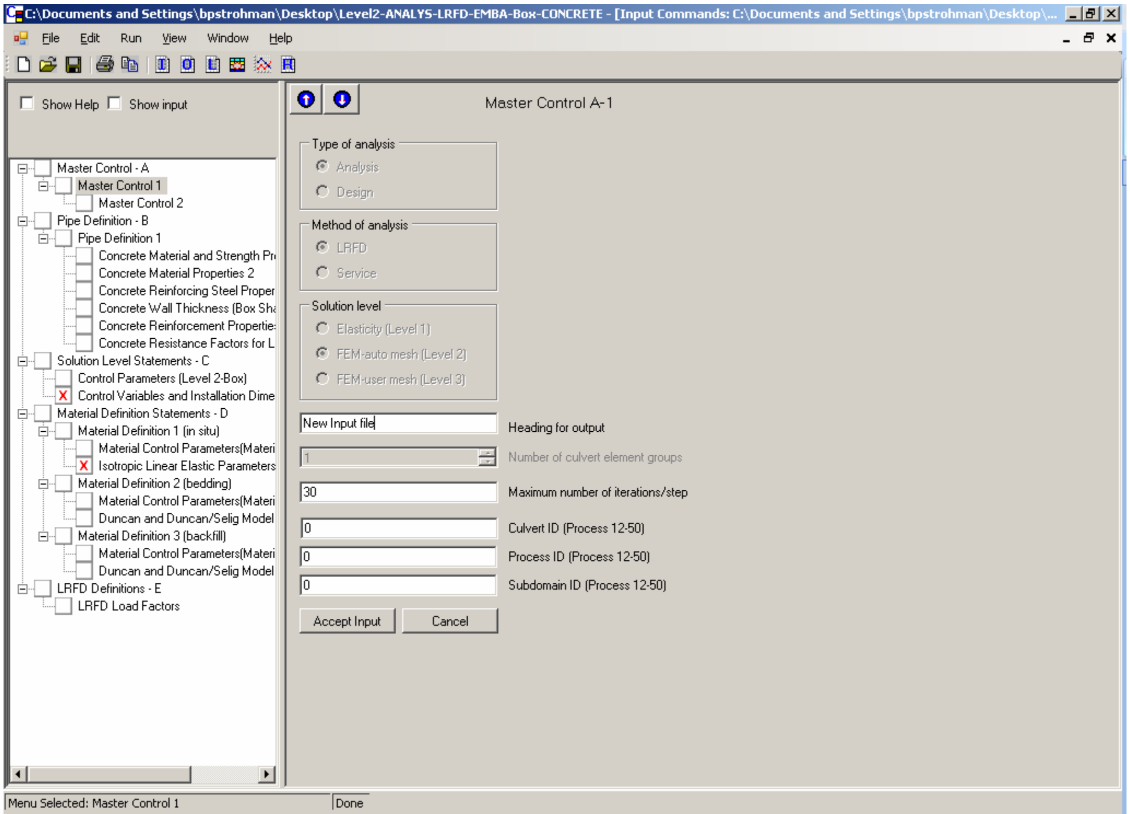


Figure 7-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 7-5 shows “X” for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 7-6 through 7-14 show the completed input for the screens requiring data for the tutorial, except that only one material definition screen is shown. Input the remaining definition screens. As input is modified, a ‘blue circle’ icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click “Accept Input” and the blue circle in the control panel will disappear indicating all fields have data.

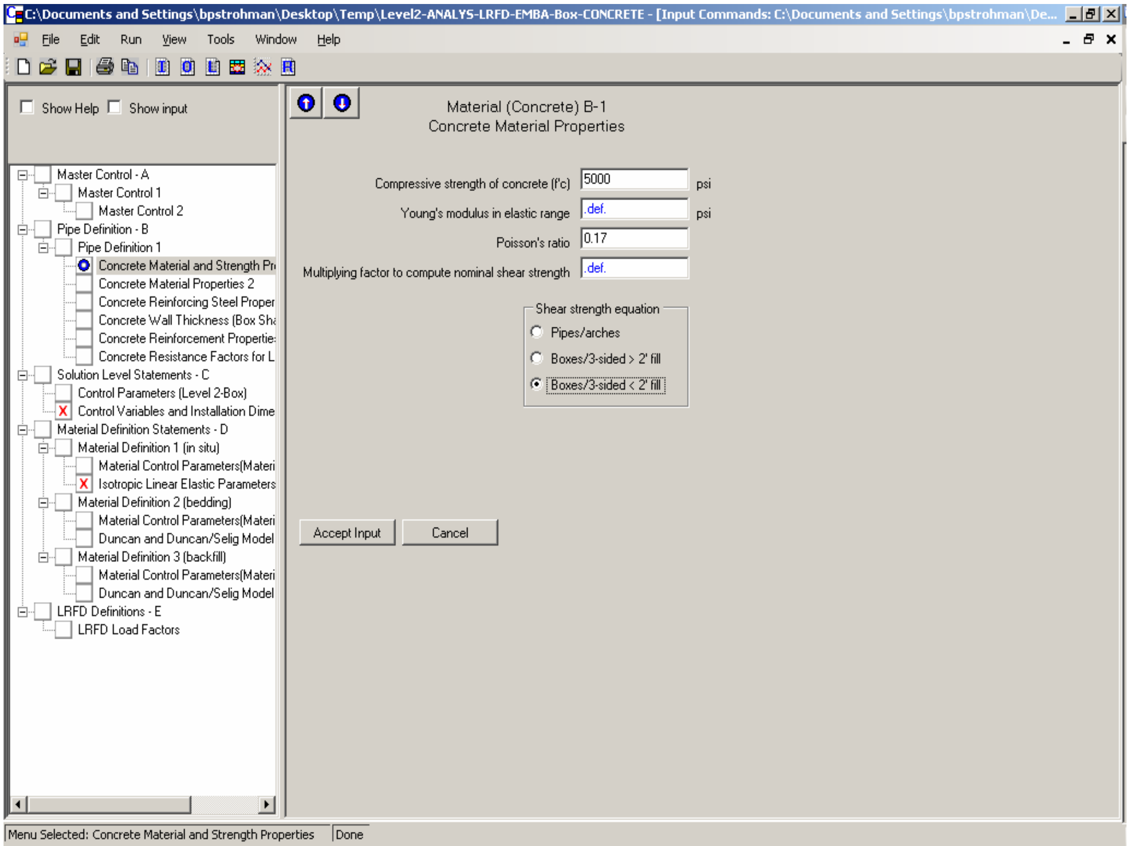


Figure 7-6 – Input Screen B-1

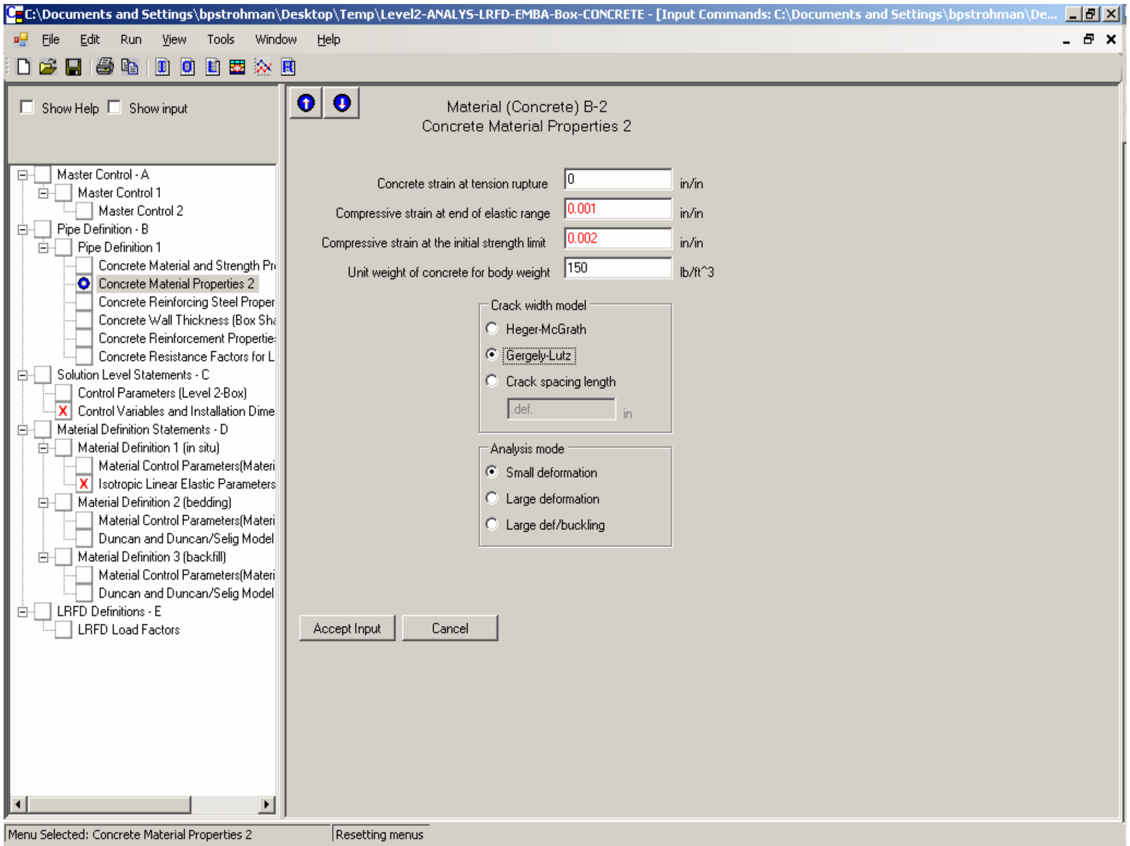


Figure 7-7 – Input Screen B-2

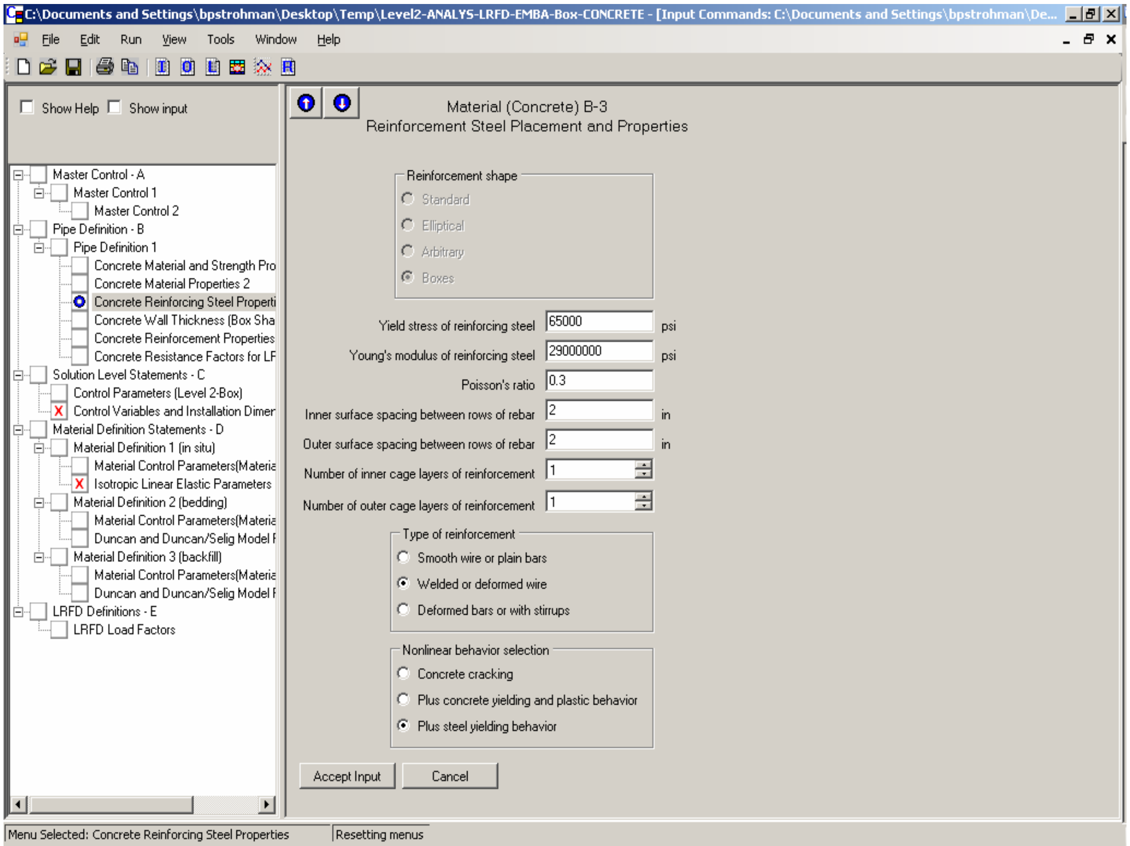


Figure 7-8 – Input Screen B-3

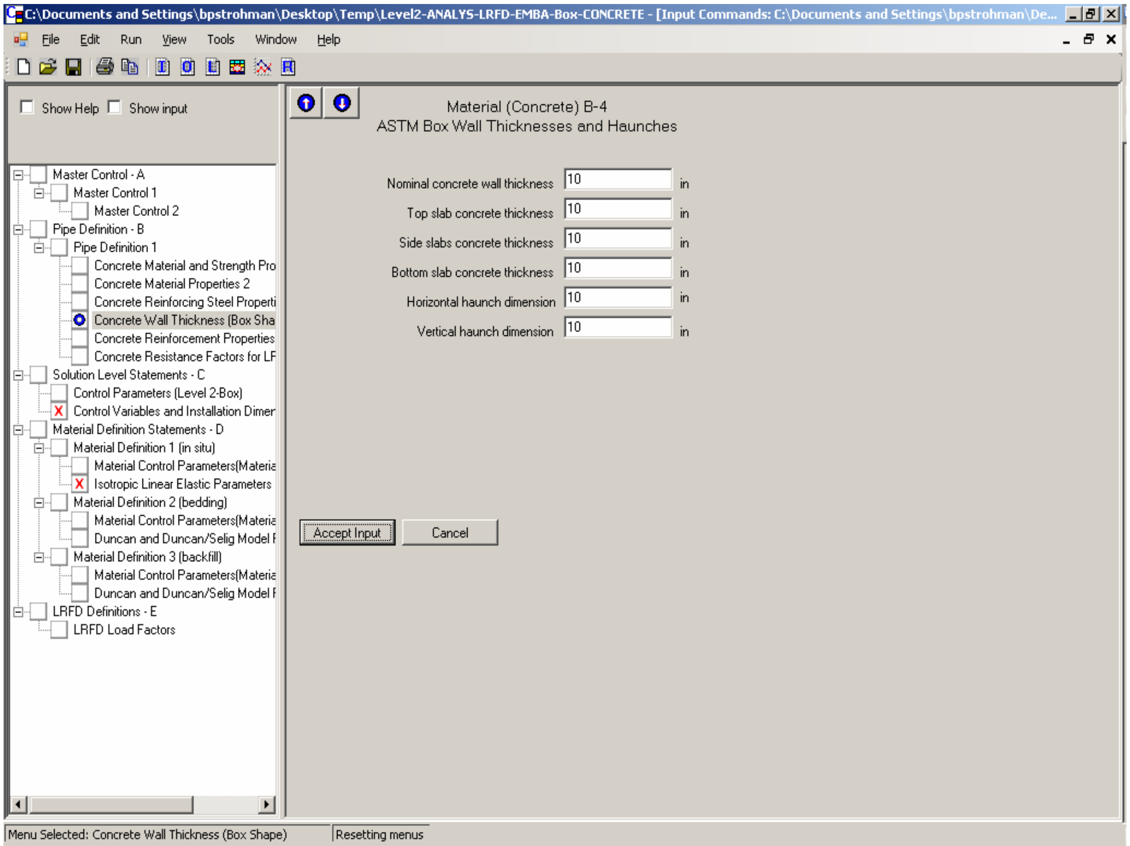


Figure 7-9 – Input Screen B-4

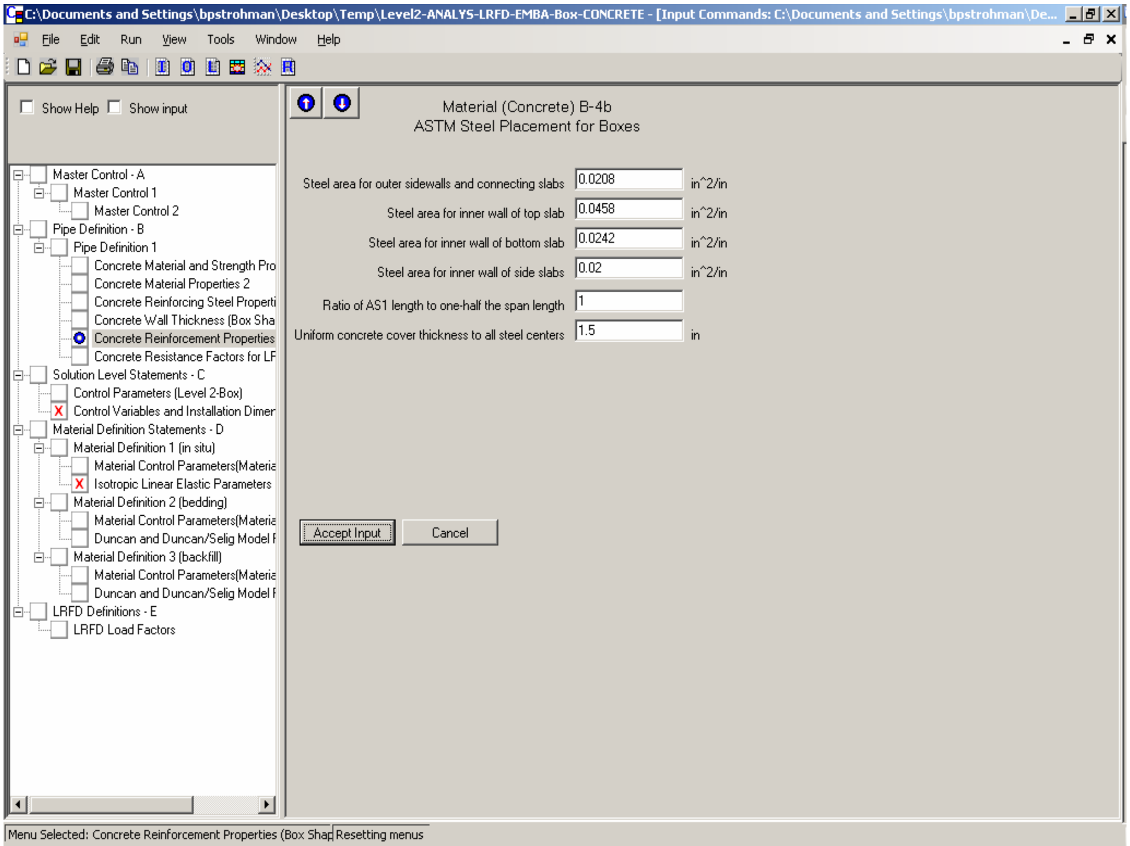


Figure 7-10 – Input Screen B-4b

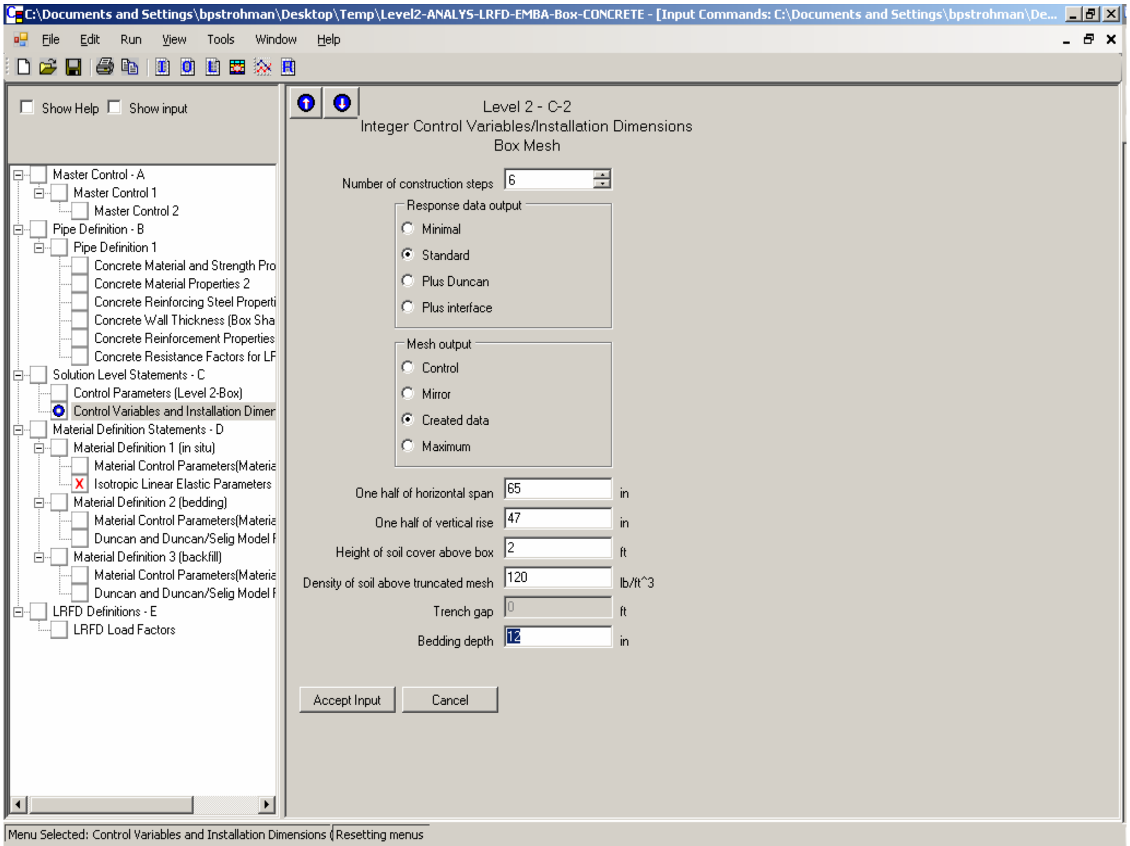


Figure 7-11 – Input Screen C-2

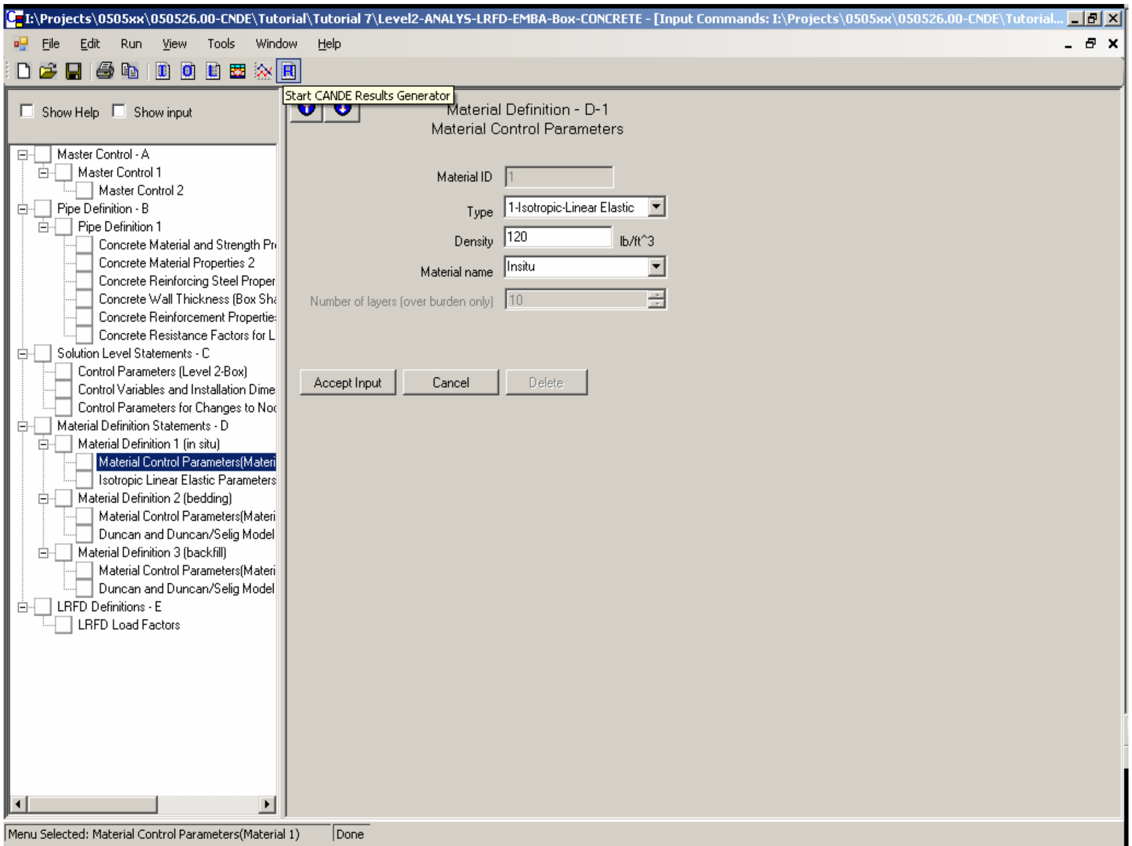


Figure 7-12 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

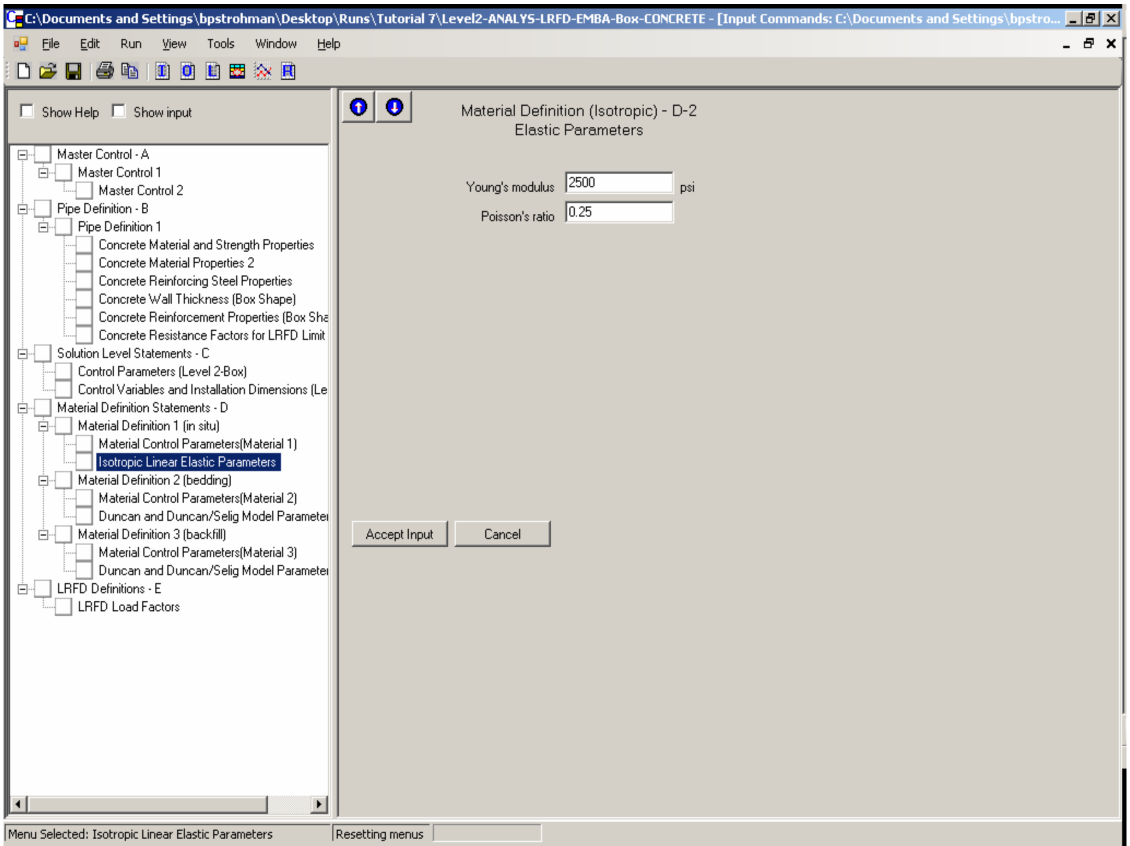


Figure 7-13 – Input Screen D-2 for Material 1 – In Situ Soil

(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

Under LRFD load factor in Figure 7-14 enter the combined value of the load factor and load modifier, i.e. – 1.35 * 1.05 = 1.42

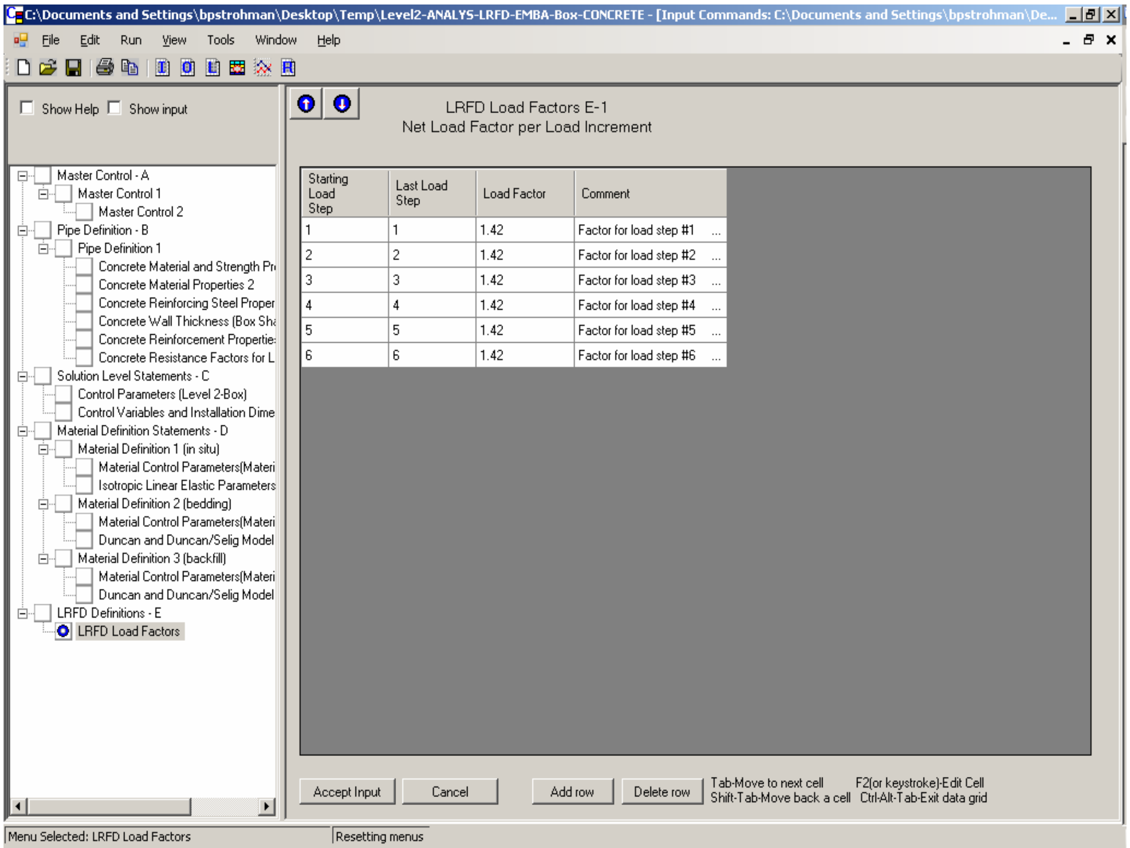


Figure 7-14 – Input Screen E-1

When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

7.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The "Plotting Parameters" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting parameters menu and click the check box "Show constr. increment numbers." Click "OK," set Load Increment to 6 to show the entire mesh and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 7-15.

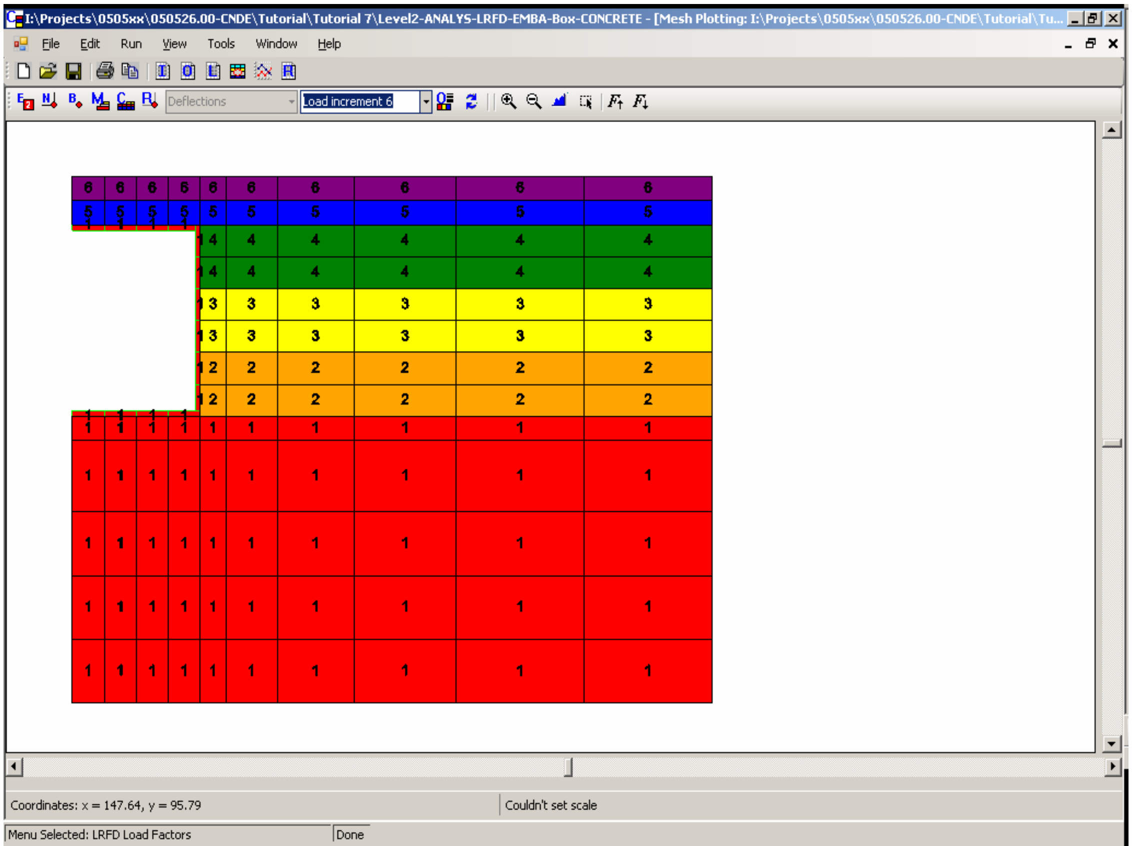


Figure 7-15 – Mesh Plot for Load Steps 1 to 6

To view deflections, open the plotting parameters window and set the deflection magnification factor to 10, click the "Ignore deflections/by load incr." check box and set the Load Increment to 1 (see Figure 7-16).

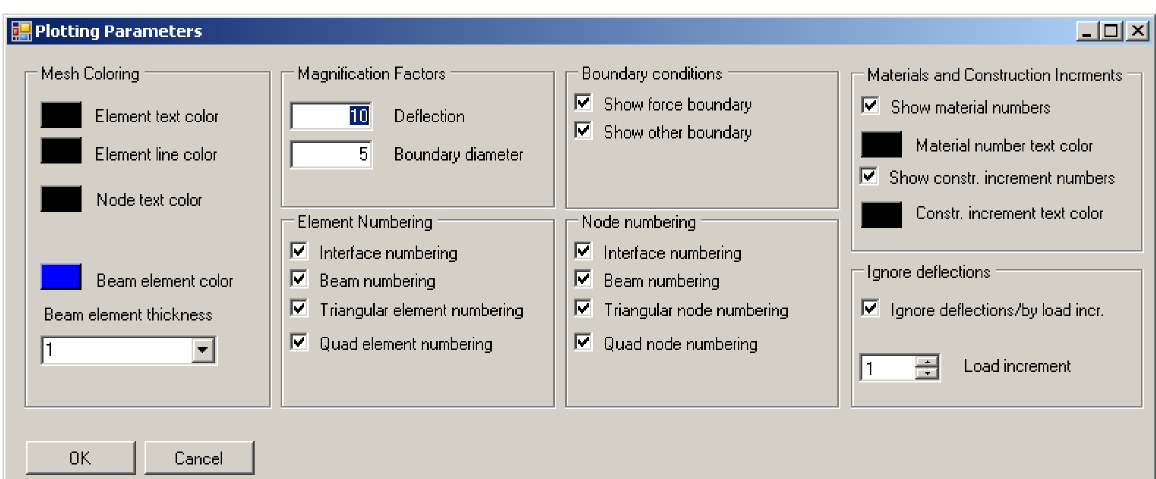


Figure 7-16 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Deflections. Note that if the Load Increment is set to 1, the deflections due to Load Increment 1 are shown, but when the Load Increment is set to 2, the Increment 1 deflections are ignored. Set the increment to 6 and the screen should look like Figure 7-17. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

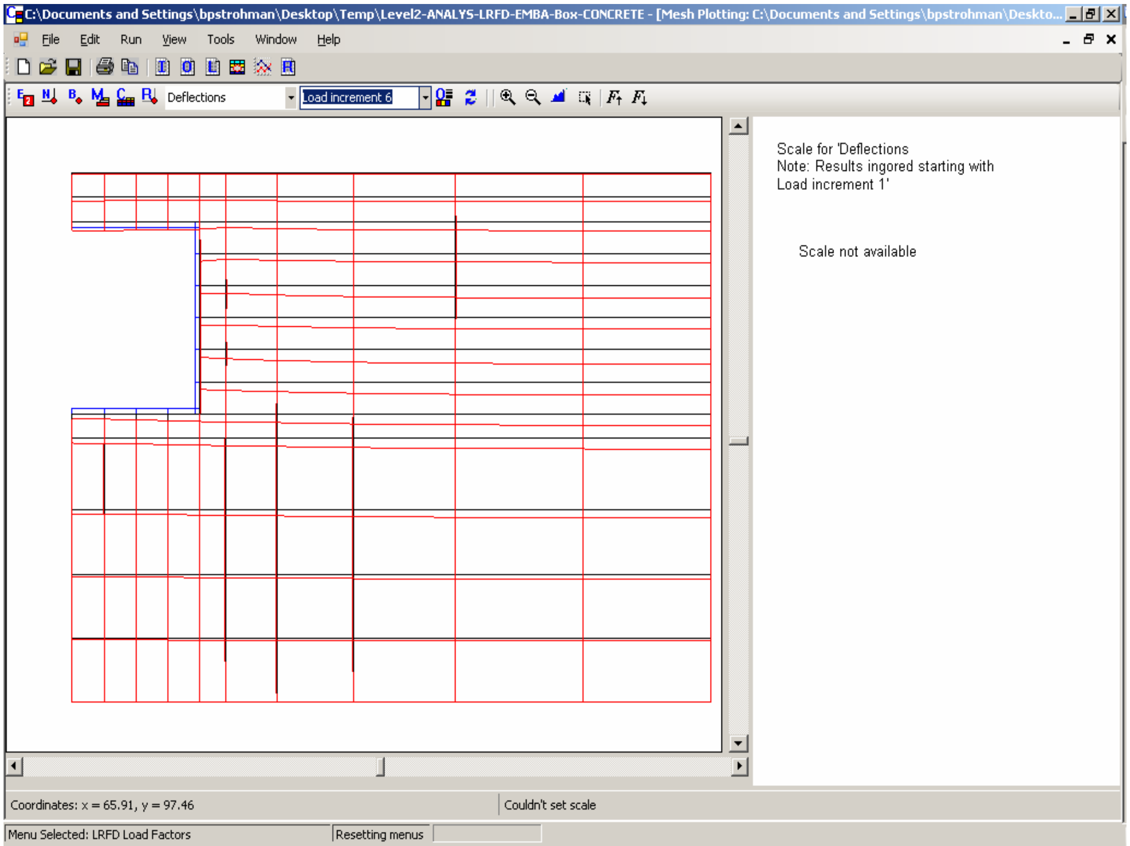


Figure 7-17 – Deflections, Excluding Those Due to Self Weight of In Situ Soil

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 7-18 compares the bending moments after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 3, 4, and 6, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Bending moment” and the screen should appear as Figure 7-18.

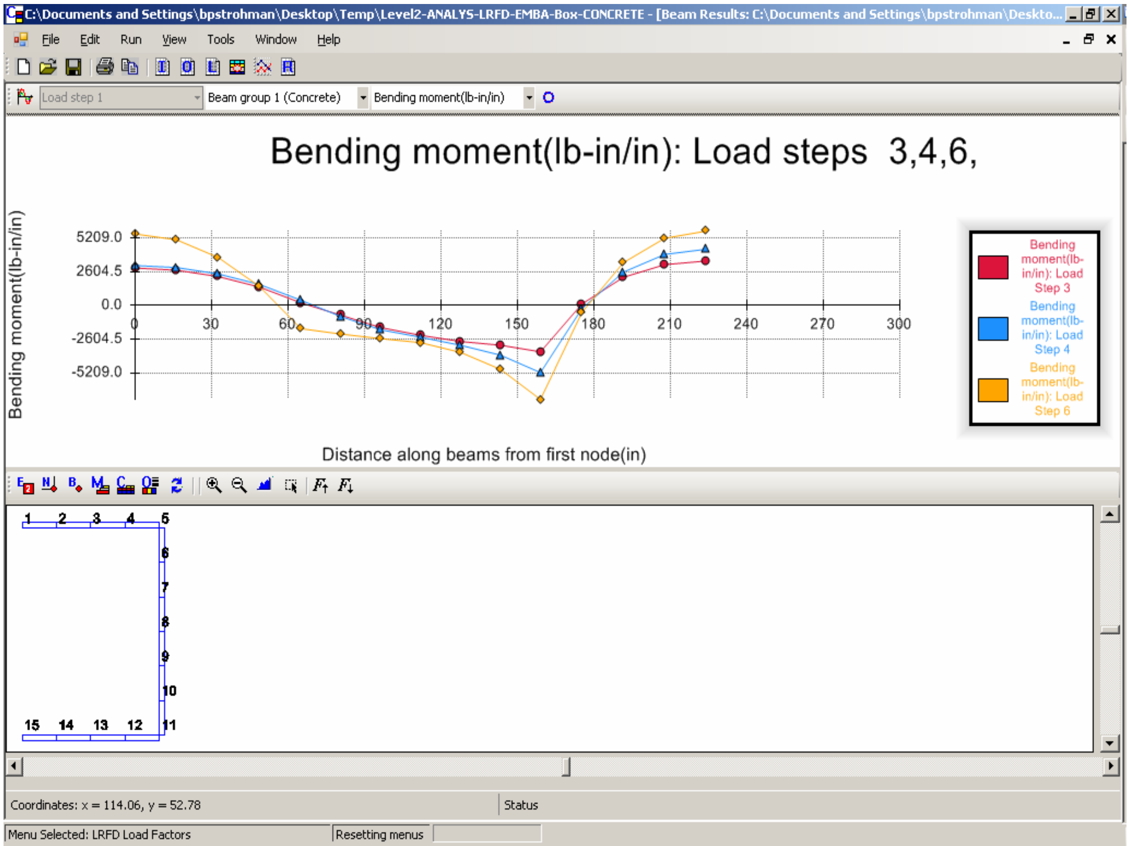


Figure 7-18 – Bending Moment for Load Steps 3, 4, and 6

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 7-19) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 7-19 shows the final assessment printed at the end of the file.

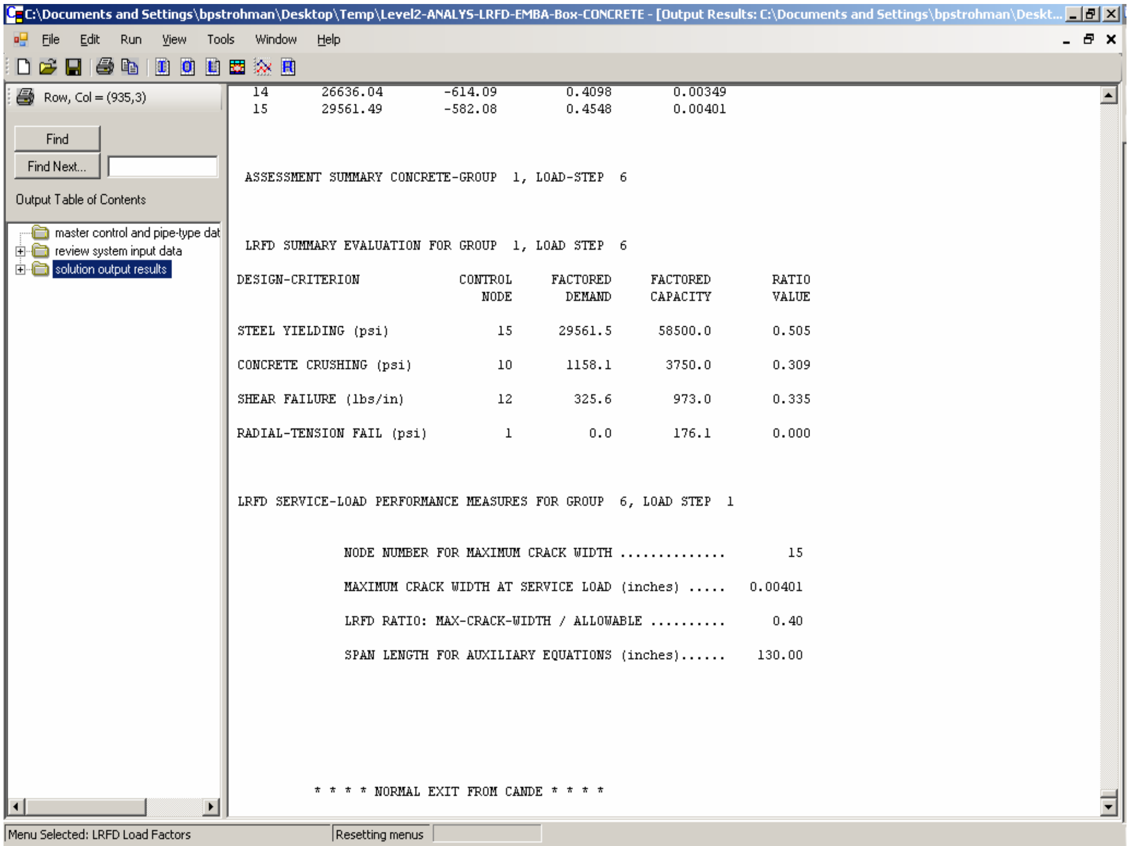


Figure 7-19 – Design Assessment Summary – Load Step 6

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 7-20 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 7-21. The three tabs shown in Figures 7-20 and 7-21 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

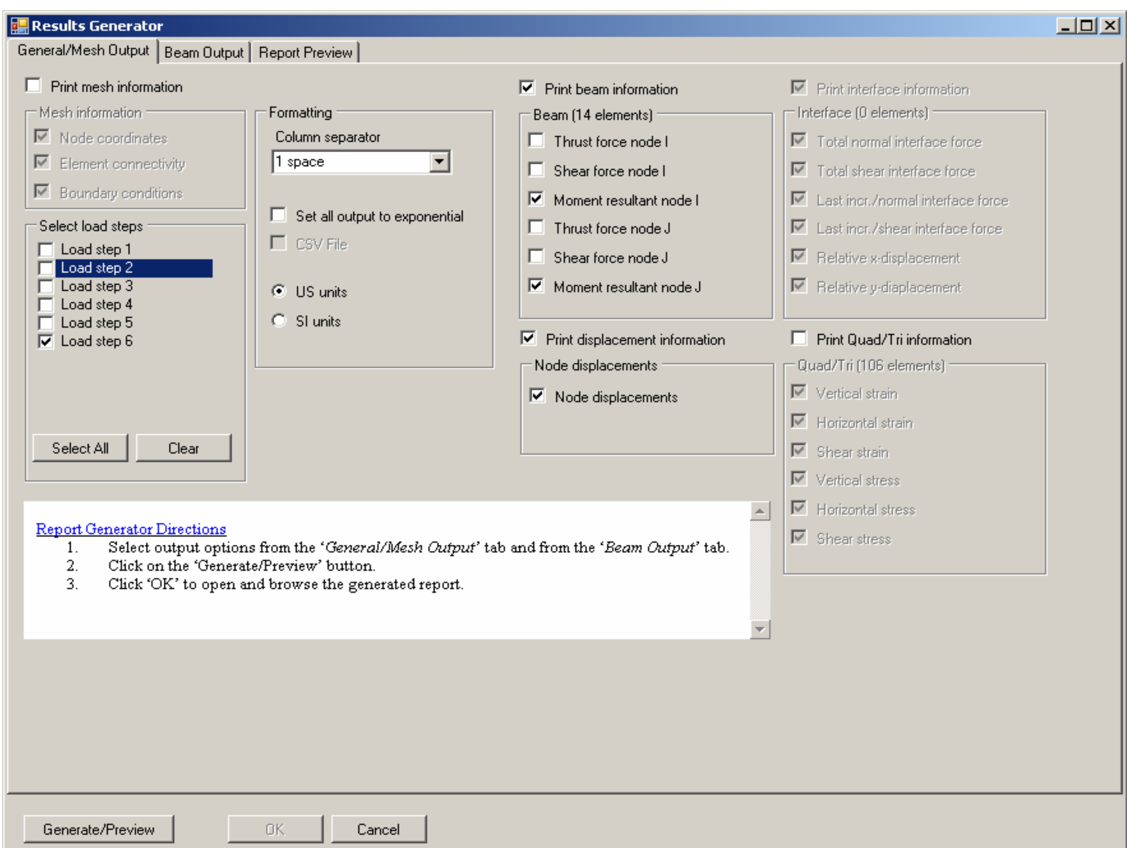
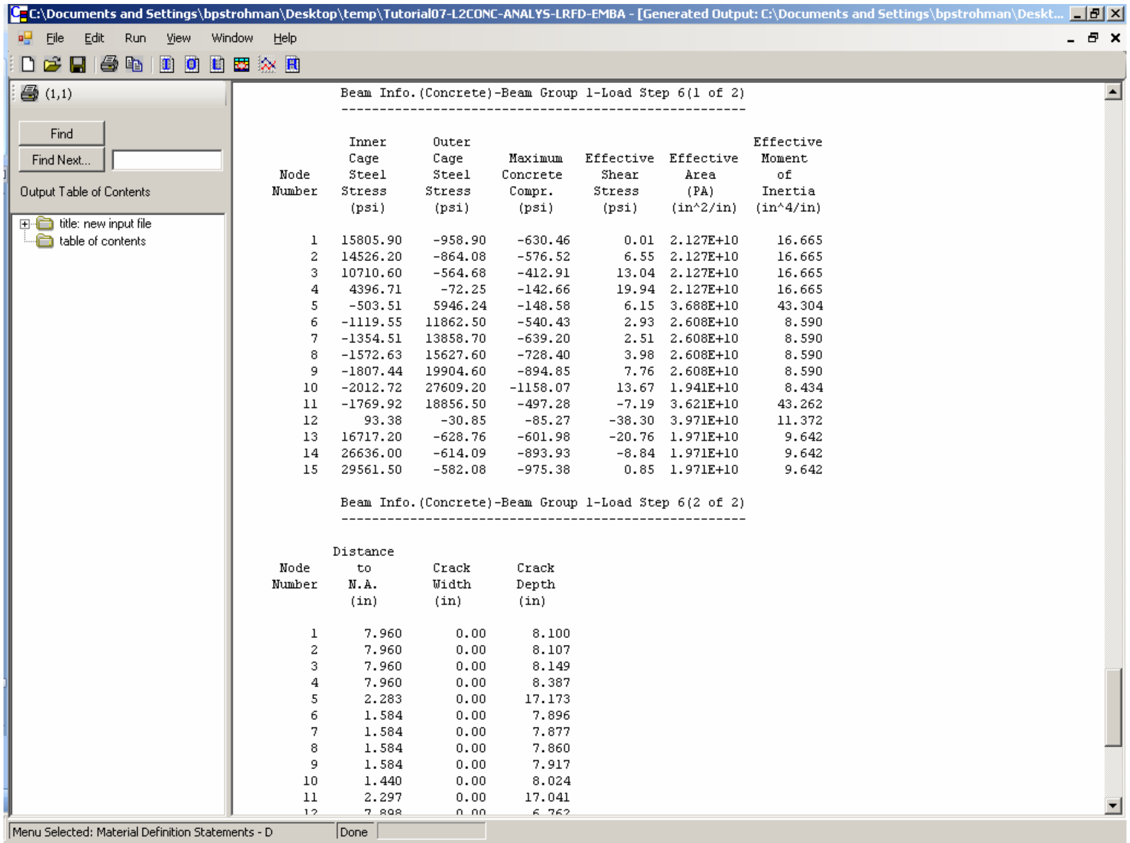


Figure 7-20 – Results Generator Input Screen – Load Step 6 Moments and Deflections



Beam Info.(Concrete)-Beam Group 1-Load Step 6(1 of 2)

Node Number	Inner Cage Steel Stress (psi)	Outer Cage Steel Stress (psi)	Maximum Concrete Compr. (psi)	Effective Shear Stress (psi)	Effective Area (PA) (in^2/in)	Effective Moment of Inertia (in^4/in)
1	15805.90	-958.90	-630.46	0.01	2.127E+10	16.665
2	14526.20	-864.08	-576.52	6.55	2.127E+10	16.665
3	10710.60	-564.68	-412.91	13.04	2.127E+10	16.665
4	4396.71	-72.25	-142.66	19.94	2.127E+10	16.665
5	-503.51	5946.24	-148.58	6.15	3.688E+10	43.304
6	-1119.55	11862.50	-540.43	2.93	2.608E+10	8.590
7	-1354.51	13858.70	-639.20	2.51	2.608E+10	8.590
8	-1572.63	15627.60	-728.40	3.98	2.608E+10	8.590
9	-1807.44	19904.60	-894.85	7.76	2.608E+10	8.590
10	-2012.72	27609.20	-1158.07	13.67	1.941E+10	8.434
11	-1769.92	18856.50	-497.28	-7.19	3.621E+10	43.262
12	93.38	-30.85	-85.27	-38.30	3.971E+10	11.372
13	16717.20	-628.76	-601.98	-20.76	1.971E+10	9.642
14	26636.00	-614.09	-893.93	-8.84	1.971E+10	9.642
15	29561.50	-582.08	-975.38	0.85	1.971E+10	9.642

Beam Info.(Concrete)-Beam Group 1-Load Step 6(2 of 2)

Node Number	Distance to N.A. (in)	Crack Width (in)	Crack Depth (in)
1	7.960	0.00	8.100
2	7.960	0.00	8.107
3	7.960	0.00	8.149
4	7.960	0.00	8.387
5	2.283	0.00	17.173
6	1.584	0.00	7.896
7	1.584	0.00	7.877
8	1.584	0.00	7.860
9	1.584	0.00	7.917
10	1.440	0.00	8.024
11	2.297	0.00	17.041
12	7.898	0.00	6.762

Figure 7-21 – Results Generator – Load Step 6 Beam Stresses

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 8

May 2008

This page intentionally left blank

8. CANDE TEST PROBLEM 8

8.1 Problem Definition

Analyze a 144 in. corrugated steel pipe with 8 slotted joints and 60 ft of fill over the top of the pipe using LRFD analysis. The problem is shown schematically in Figure 8-1. The analysis will be with Level 2, using an automated finite element pipe mesh for an embankment installation having no interface elements. The automated finite element mesh will be modified using Level 2-extended to reduce the thickness of the construction steps above the crown of the pipe.

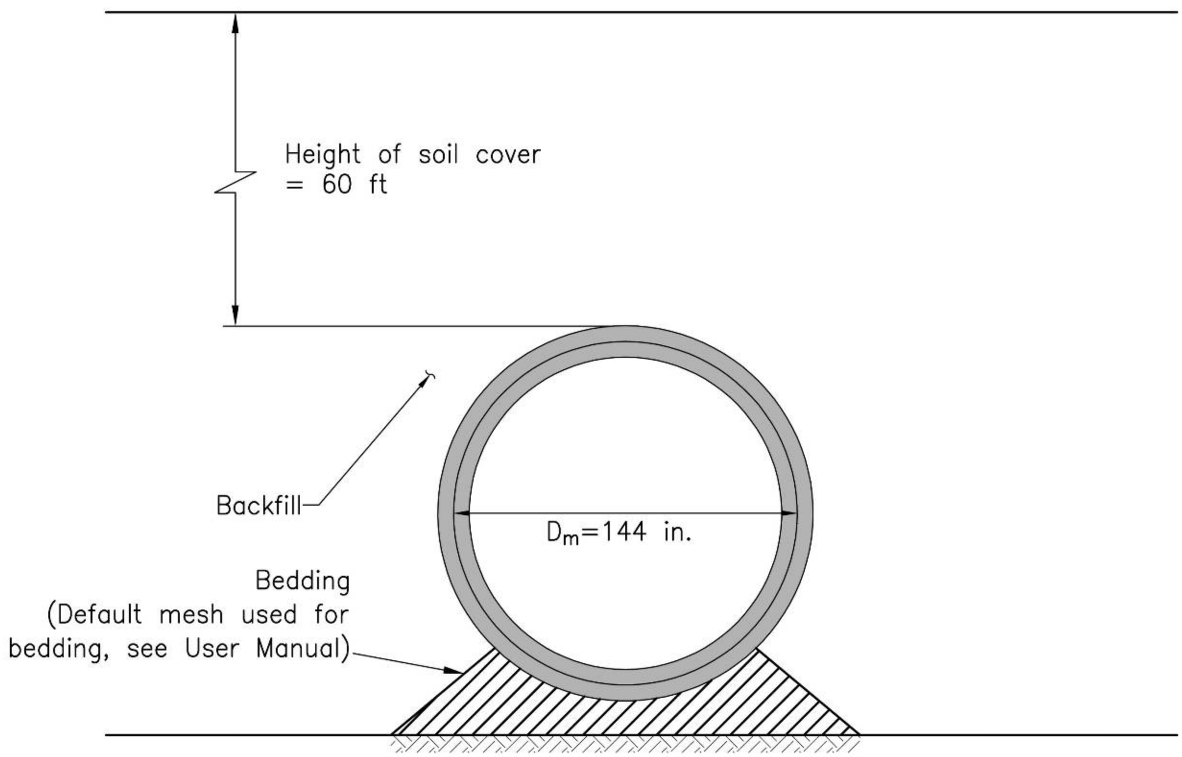


Figure 8-1 Details of Problem 8

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design. All parameters are listed here, where necessary, explanation of how parameters were selected is presented later.

Type of analysis - Analysis

Method of analysis/design - LRFD

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Pipe mesh

Soil mesh pattern - Embankment

Interface elements (pipe only) - None

MOD-Make changes to the basic mesh - check on (problem requires using Level 2-extended to reduce the thickness of the construction steps above the crown of the pipe).

Number of nodes to be changed with new coordinates - 8

Number of elements to be changed with new properties - 14

Pipe material type - Steel

Joint slip - Yes joint slip

Vary joint travel length - Different lengths

Number of slip joints - 8 (see default mesh in *User Manual*, Chapter 5, B-2b).

Soil parameters - Canned overburden-dependent soil models for each of the following materials:

In situ soil - Granular soil good

Bedding soil - Granular soil fair

Backfill soil - Granular soil good

Backpack soil - Granular soil good

Soil density - 120 lb/ft^3 for all soils

Density of steel - 0.0 lb/in.^{3}

Material behavior - Bi-linear stress/strain

Analysis mode - Small deformation

Area of pipe wall unit length - 0.228 in.^{2}/in. (Assumes 0.188 in. thick with 6 in. x 2 in. corrugations)

Moment of inertia of pipe wall / unit length - 0.108 in.^{4}/in. (Assumes 0.188 in. thick with 6 in. x 2 in. corrugations)

Section modulus of pipe wall / unit length - 0.099 in.^{3}/in. (Assumes 0.188 in. thick with 6 in. x 2 in. corrugations)

Ratio of slipping modulus to elastic steel modulus - 0.0003

Joint locations and properties - The following summarizes joint locations and properties:

	Pipe element sequence number	**Ratio of joint/standard length ***
1	1	0.5
2	2	1
3	3	0.5
4	4	0.5
5	5	0.5
6	6	1
7	7	0.5
8	8	1

* see Chapter 5, B-2c,d in the *User Manual*.

Note: The ratios of joint/standard length used in this problem were selected to demonstrate the use of slip joints with different lengths. When solving joint slip problems in practice, values may differ based on the project geometry.

Average diameter of pipe - 144 inch diameter

Height of soil cover - For embankment installations CANDE calculates height of soil cover from the springline of the pipe (see *User Manual*, Chapter 5, C-2). To achieve a total height of fill of 60 ft over the top of the pipe, set the depth to 60 ft + mean pipe radius = 66 ft (See Figure 8-1).

Density of soil above truncated mesh - 120 lb/ft^3

Number of construction steps - See *User Manual*, Chapter 5, C-3 – The default mesh provides 5 construction steps to the top of the mesh, which provides approximately 1.5 pipe diameters of fill over the pipe. For the remaining depth of fill, use 10 steps, making a total of 15 construction steps.

LRFD load factor - 1.95

Load modifier – 1.05 (non-redundant for earth load)

8.2 Creating the CANDE Input Document

Figures 8-2 through 8-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking "Next" on Screen 3, CANDE will display a screen indicating that an input path is initiated - click "Finish." The user is prompted for a filename and directory.

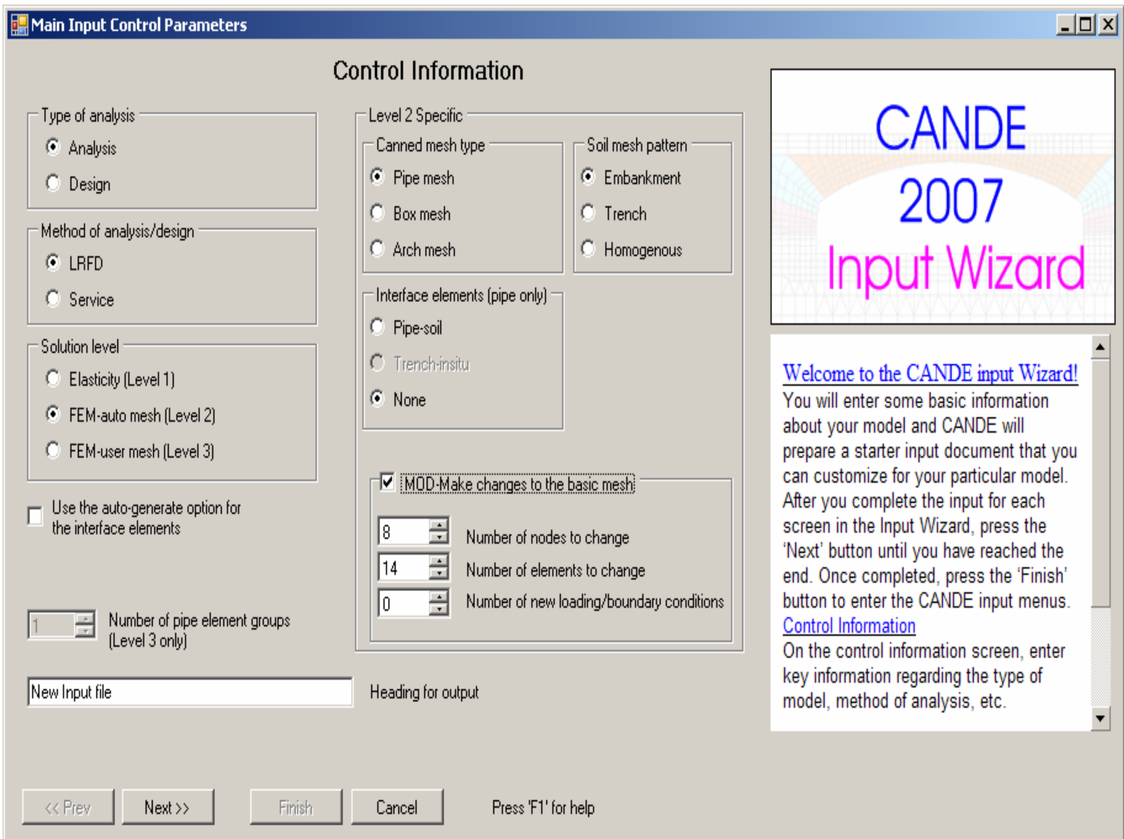


Figure 8-2 – Input Wizard, Screen 1

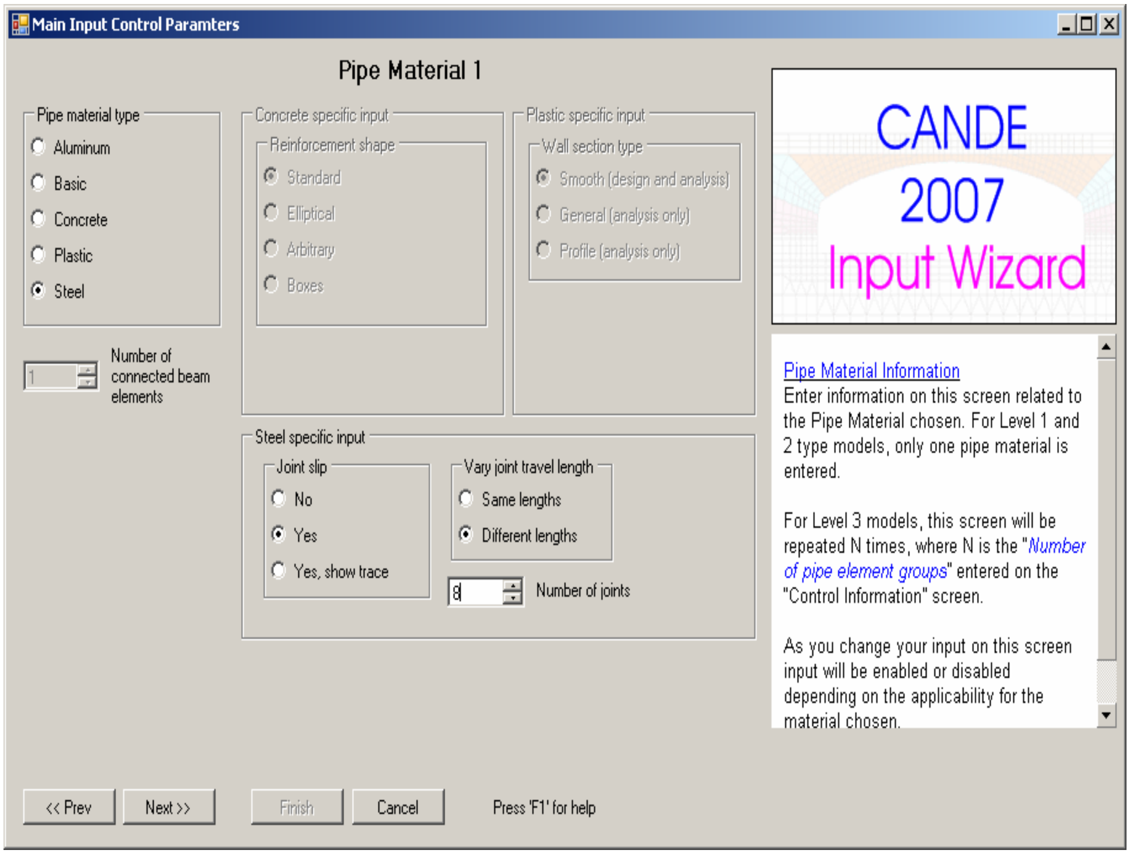


Figure 8-3 – Input Wizard, Screen 2

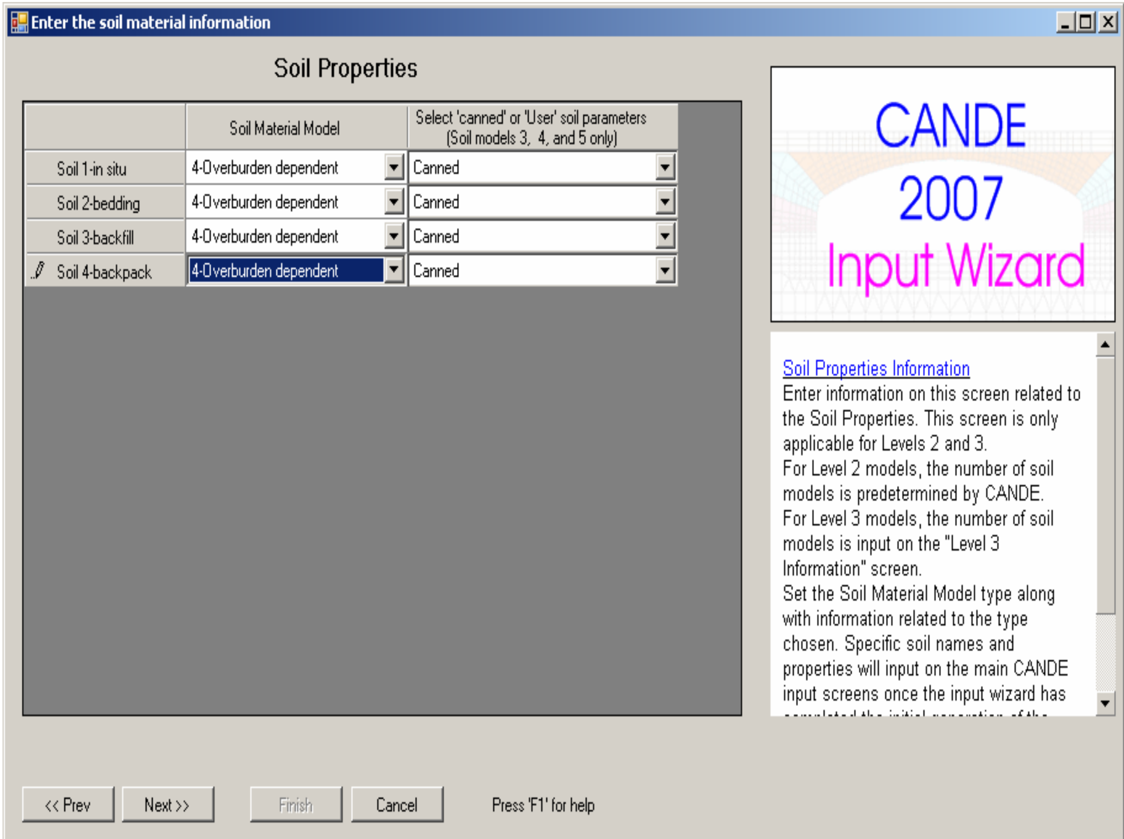


Figure 8-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 8-5. Enter an appropriate heading for output and click "Accept Input."

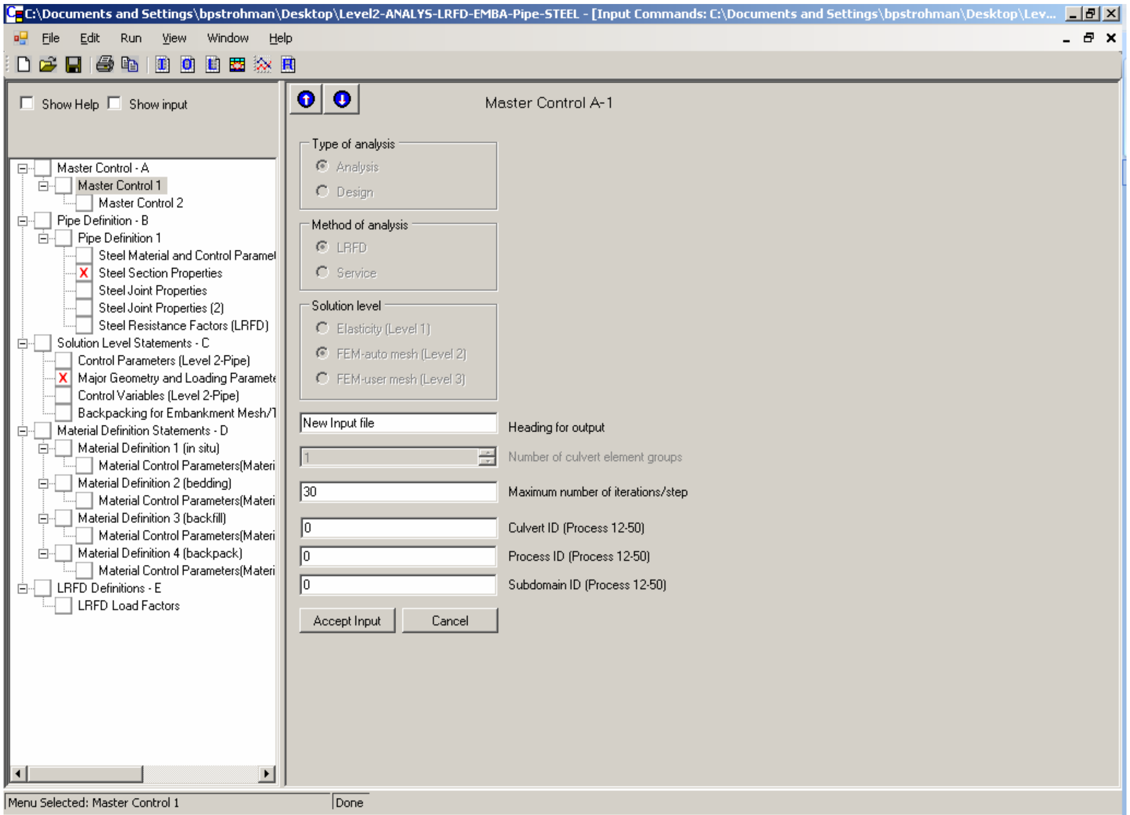


Figure 8-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 8-5 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 8-6 through 8-11 and Figures 8-15 and 8-16 show the completed input for the standard screens requiring data for the tutorial, except that only one material definition screen is shown. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data. Figures 8-12 through 8-14 show the use of Level 2-extended to modify the thickness of the construction steps above the crown of the pipe.

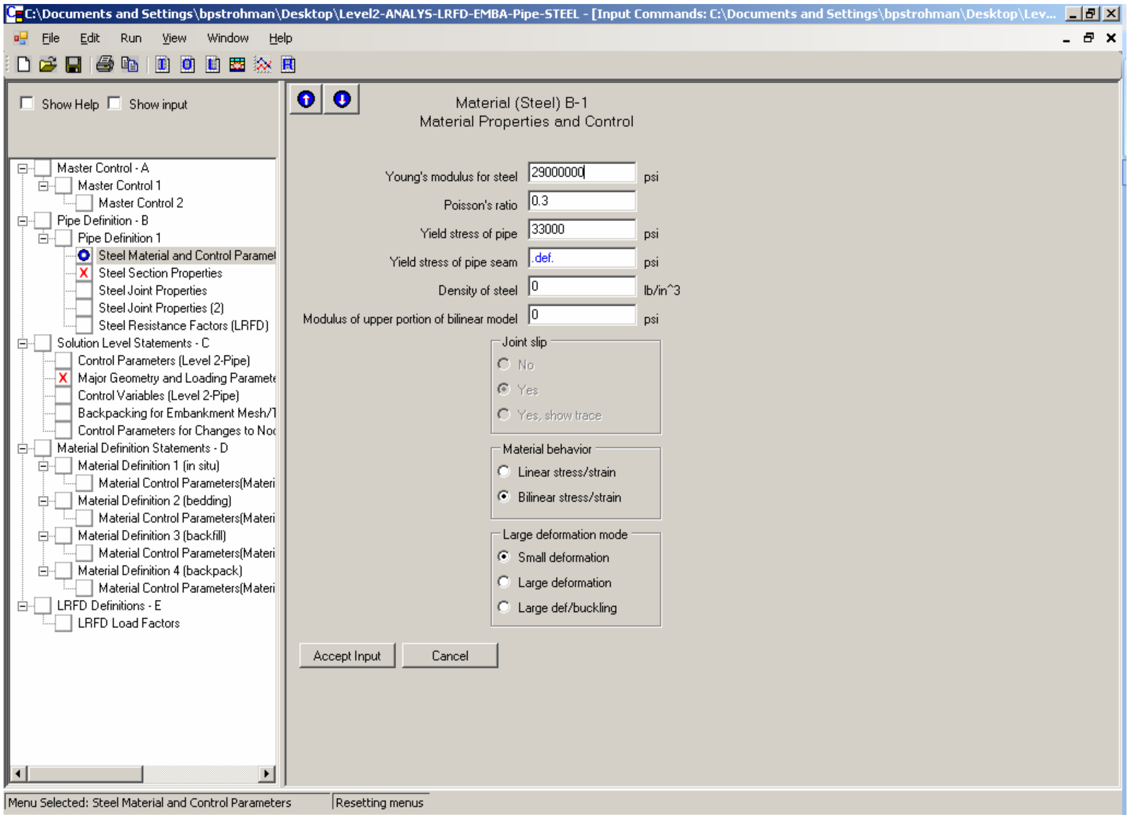


Figure 8-6 – Input Screen B-1

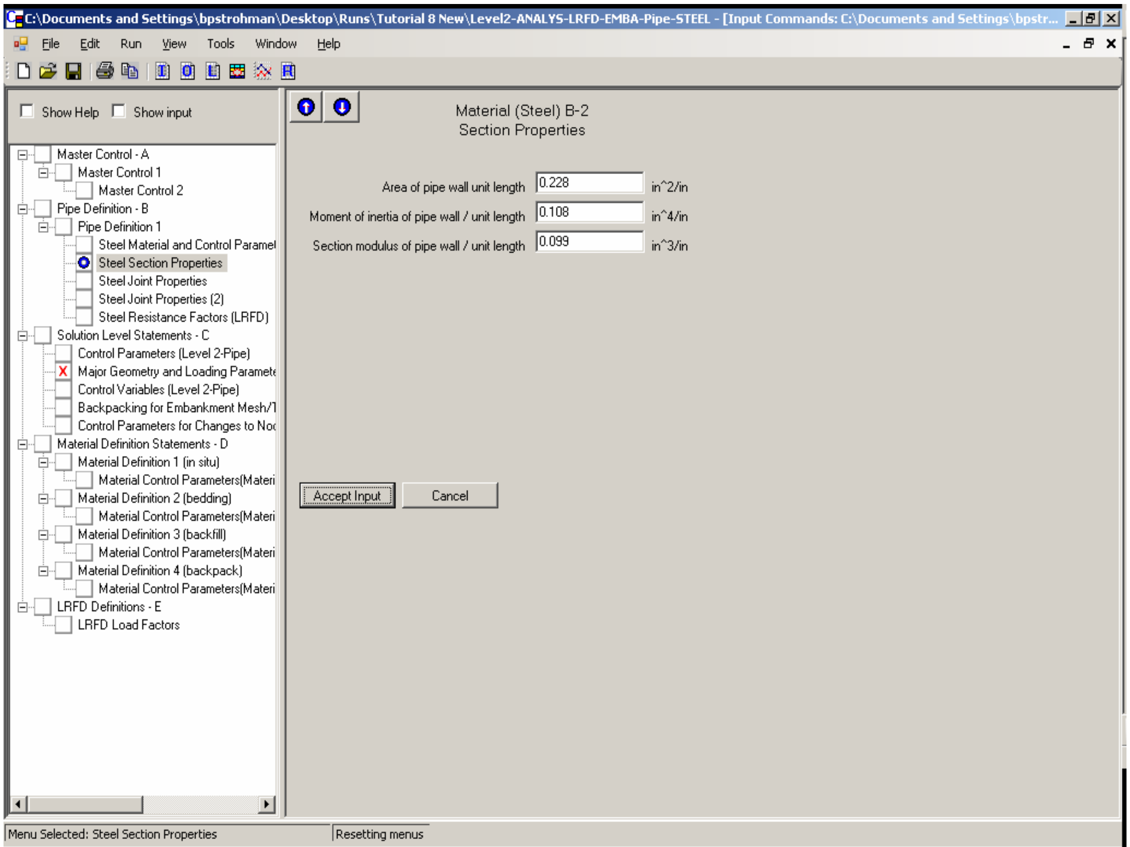


Figure 8-7 – Input Screen B-2

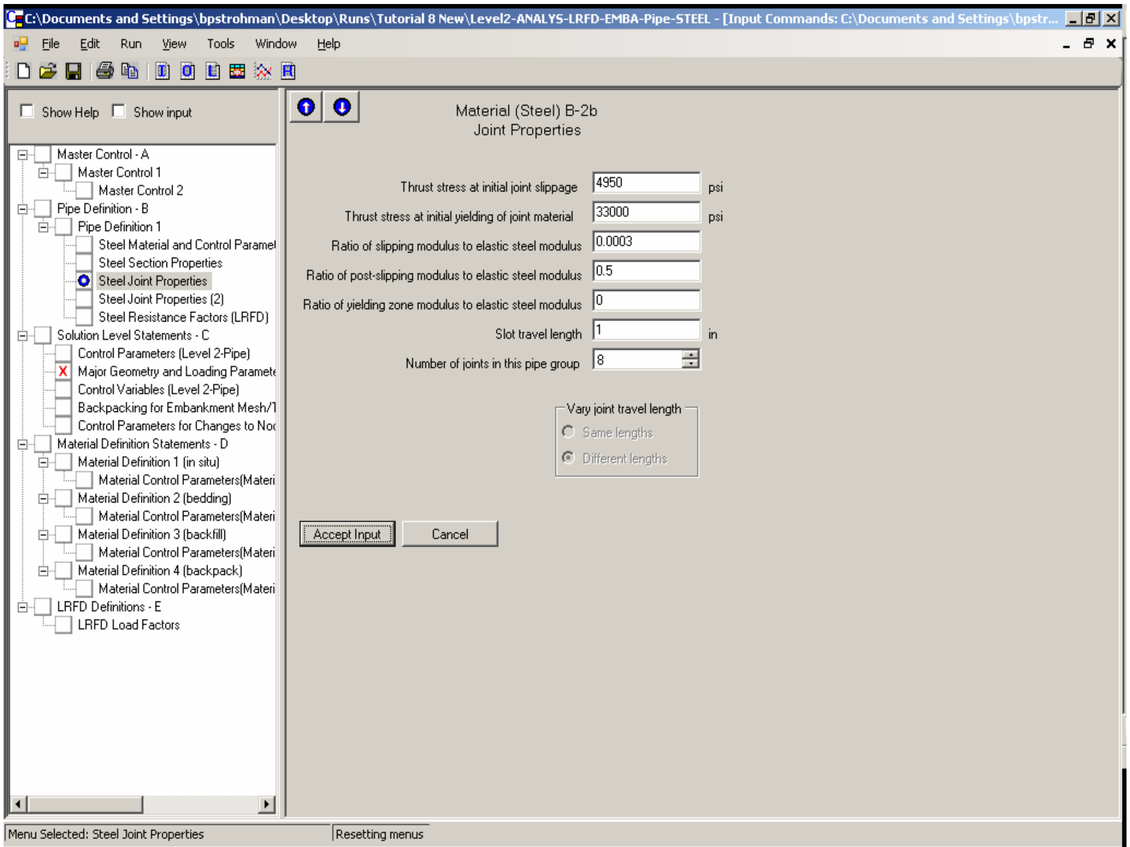


Figure 8-8 – Input Screen B-2b

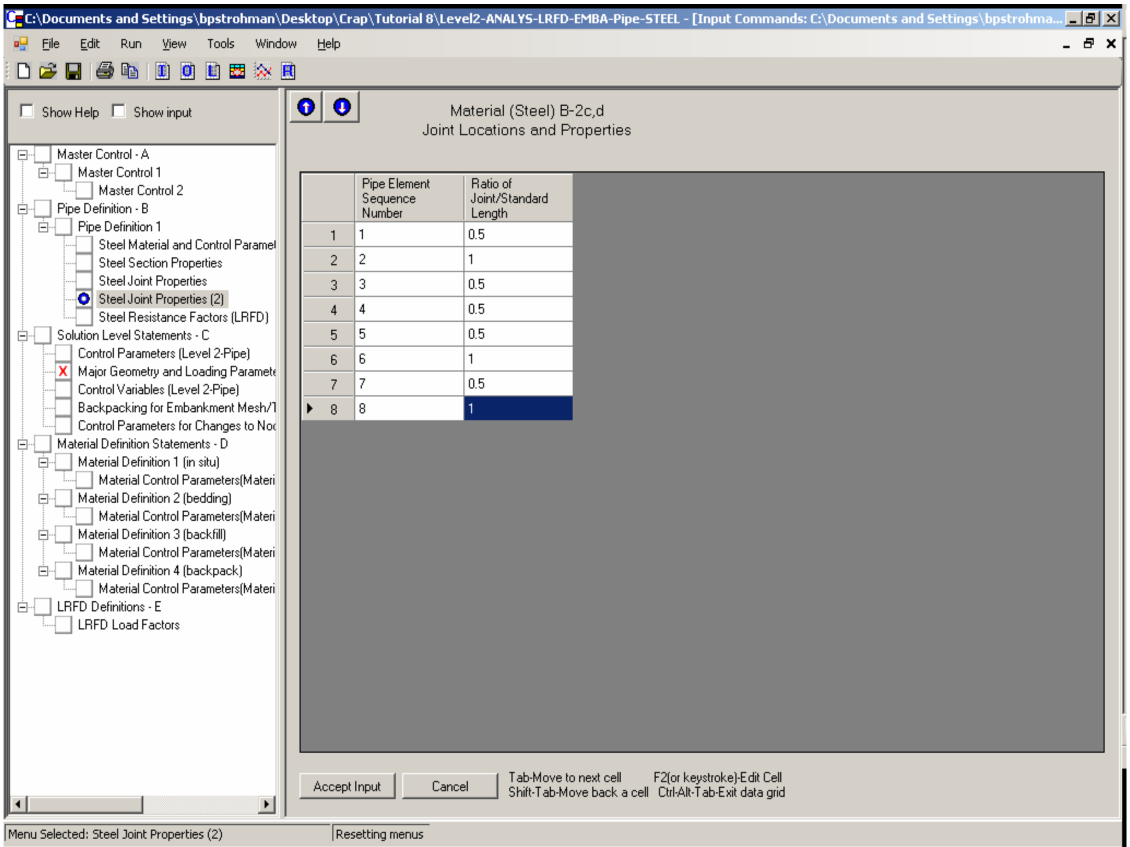


Figure 8-9 – Input Screen B-2c,d

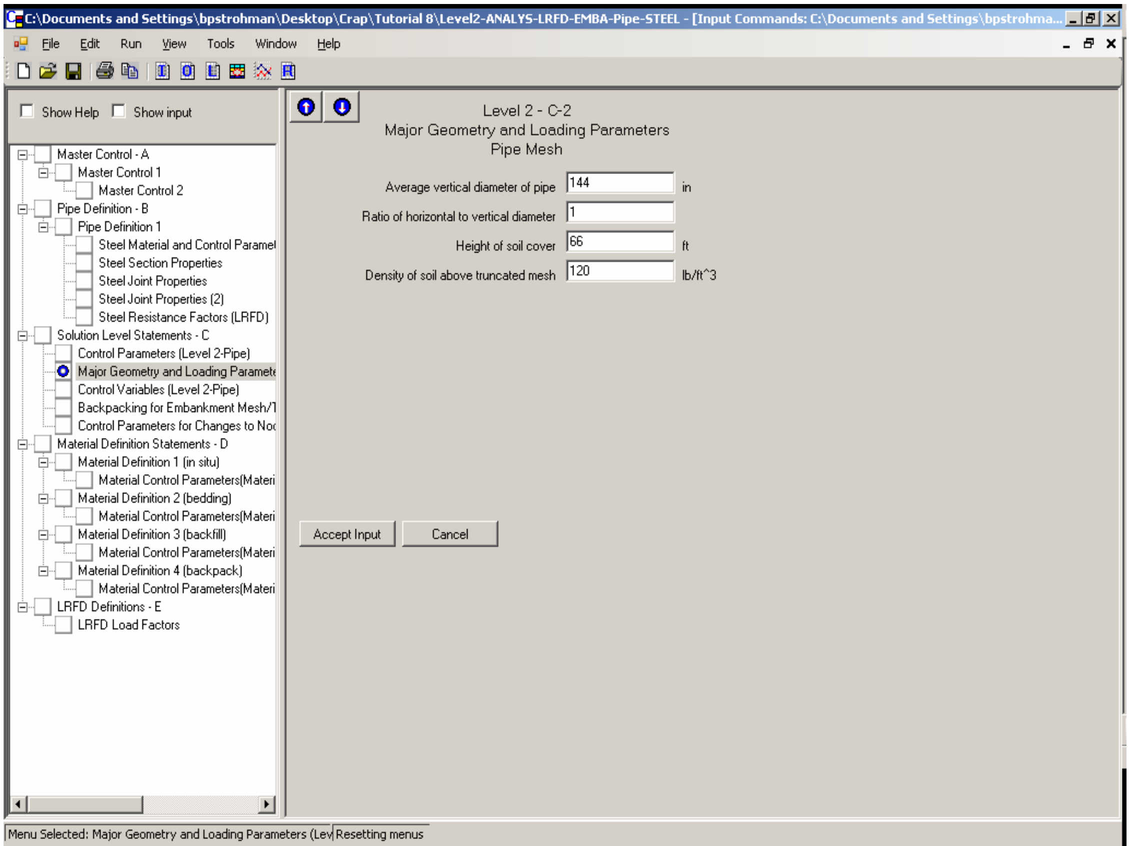


Figure 8-10 – Input Screen C-2

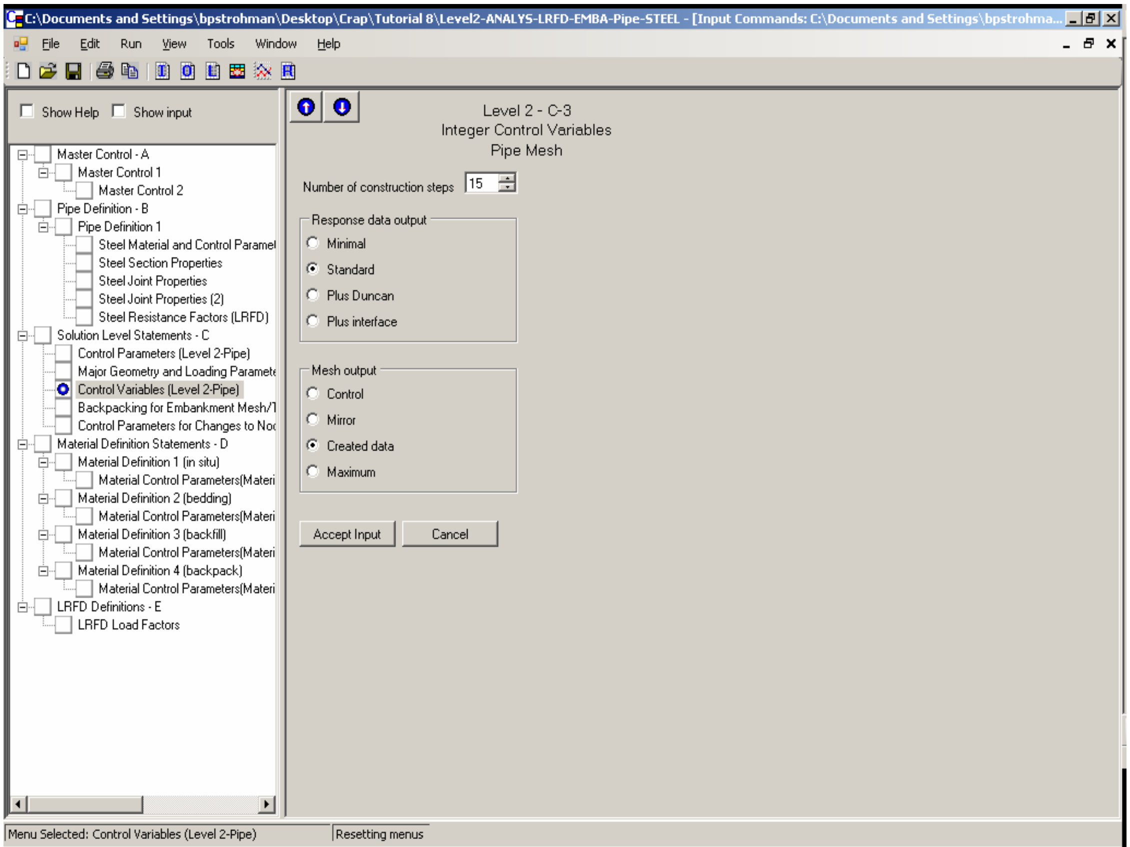


Figure 8-11 – Input Screen C-3

The automated finite element mesh will be modified using Level 2-extended to reduce the thickness of the construction steps above the crown of the pipe. Figures 8-11 through 8-13 show the completed input screens for the Level 2-extended, which require changing the y-coordinate of 8 nodes and the construction step number for 14 elements. The element, node, and construction step numbers are identified in Chapter 5, C-4 of the *User Manual* or can be obtained by making a trial run without the modified mesh and then using the mesh plot.

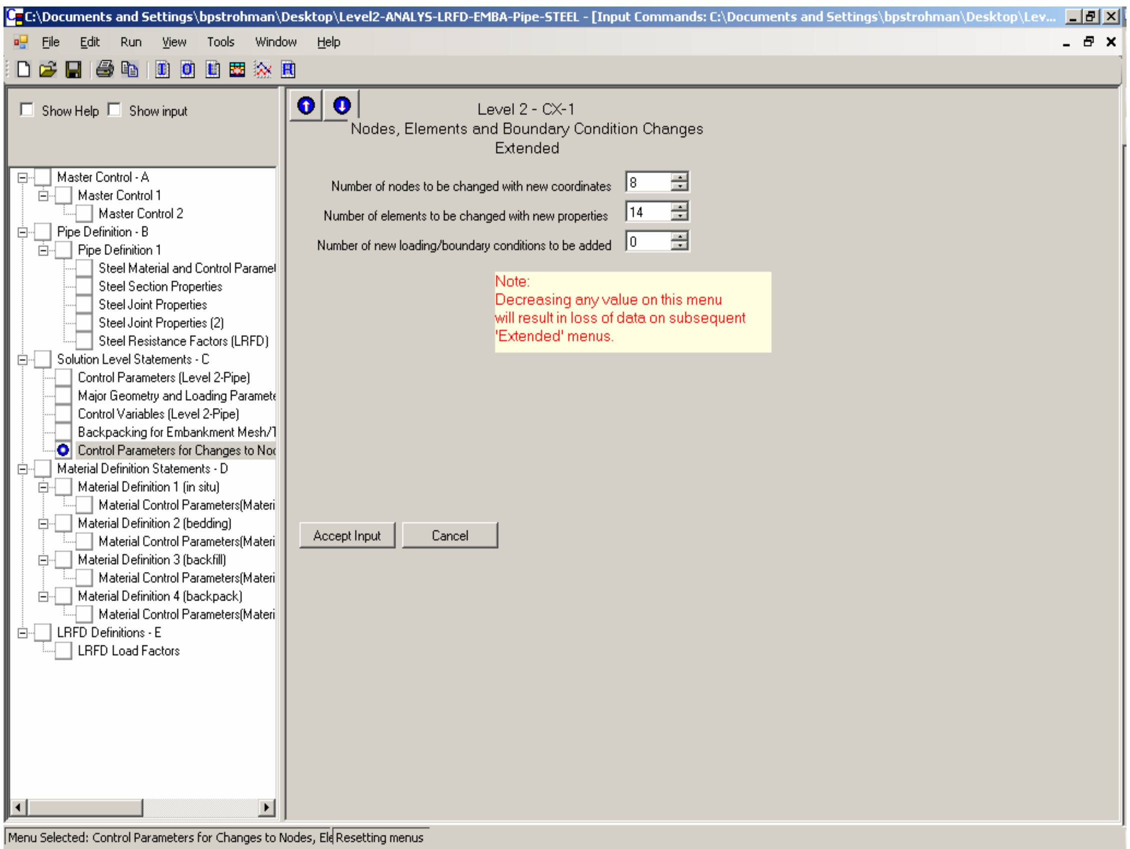


Figure 8-12 – Input Screen CX-1

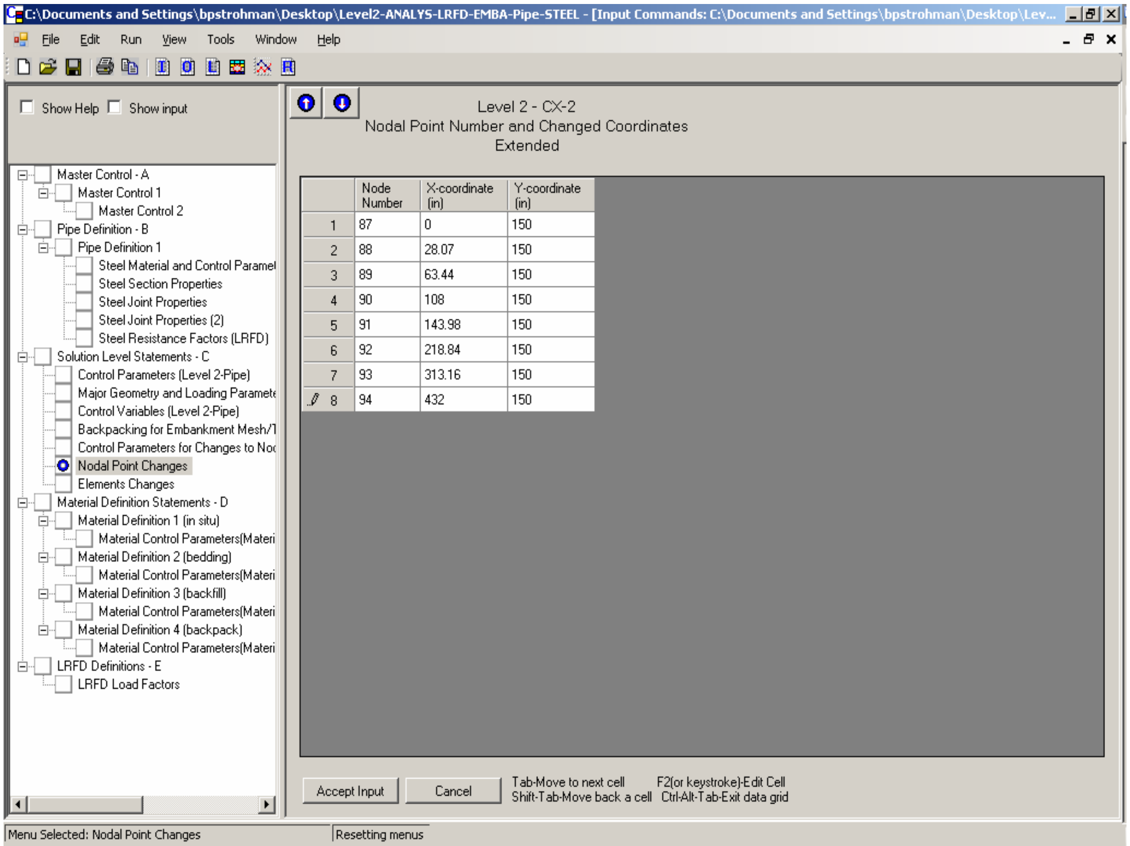


Figure 8-13 – Input Screen CX-2

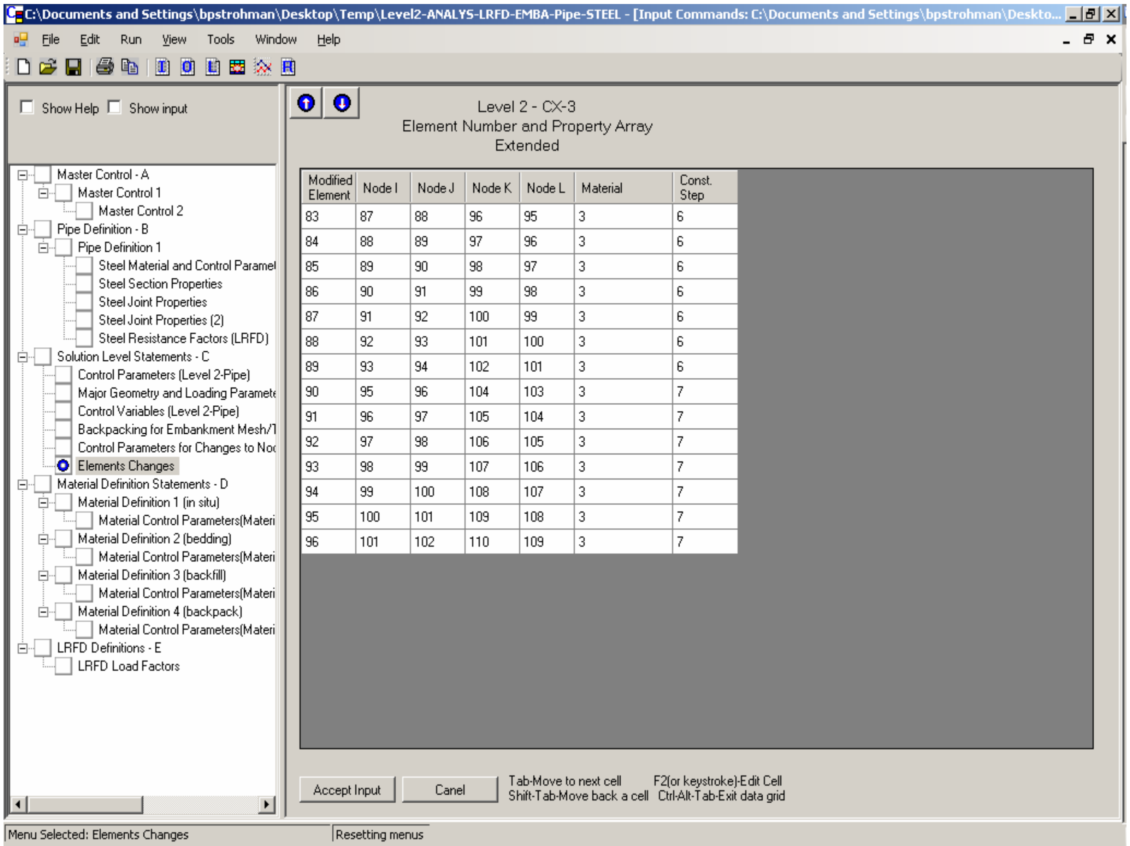


Figure 8-14 – Input Screen CX-3

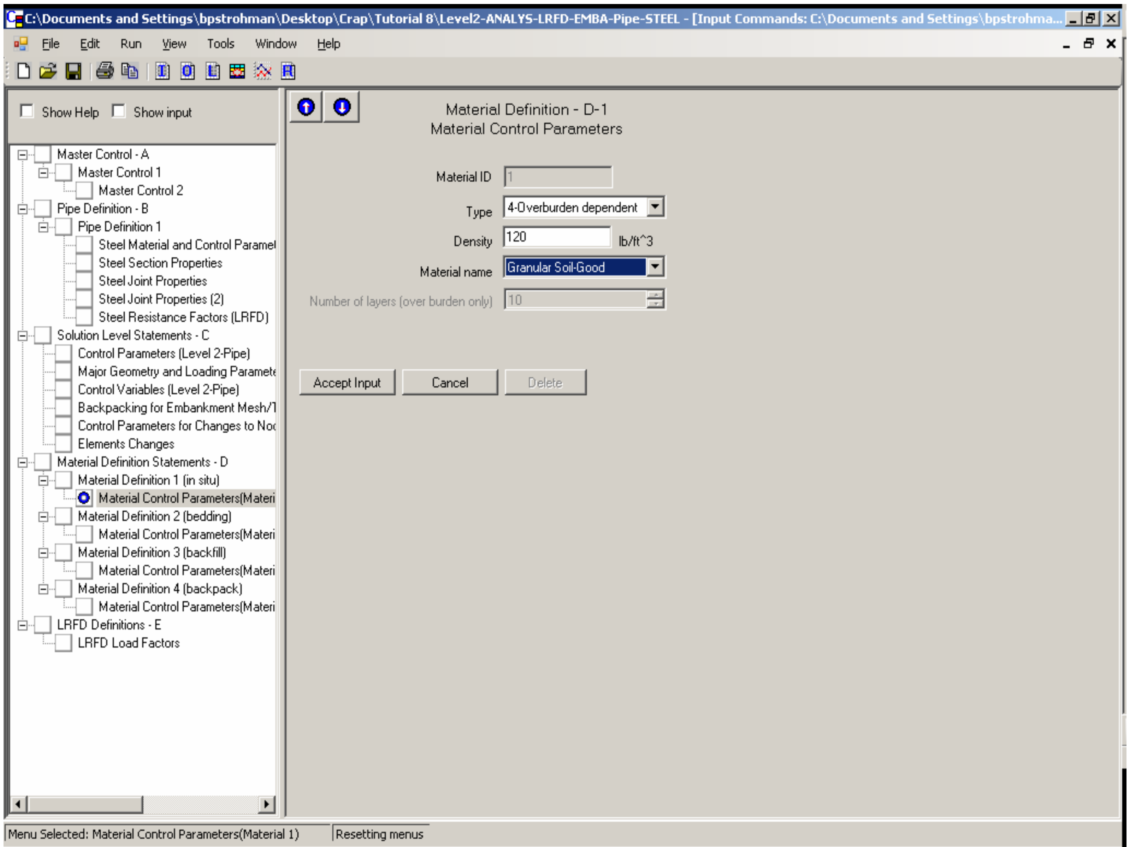
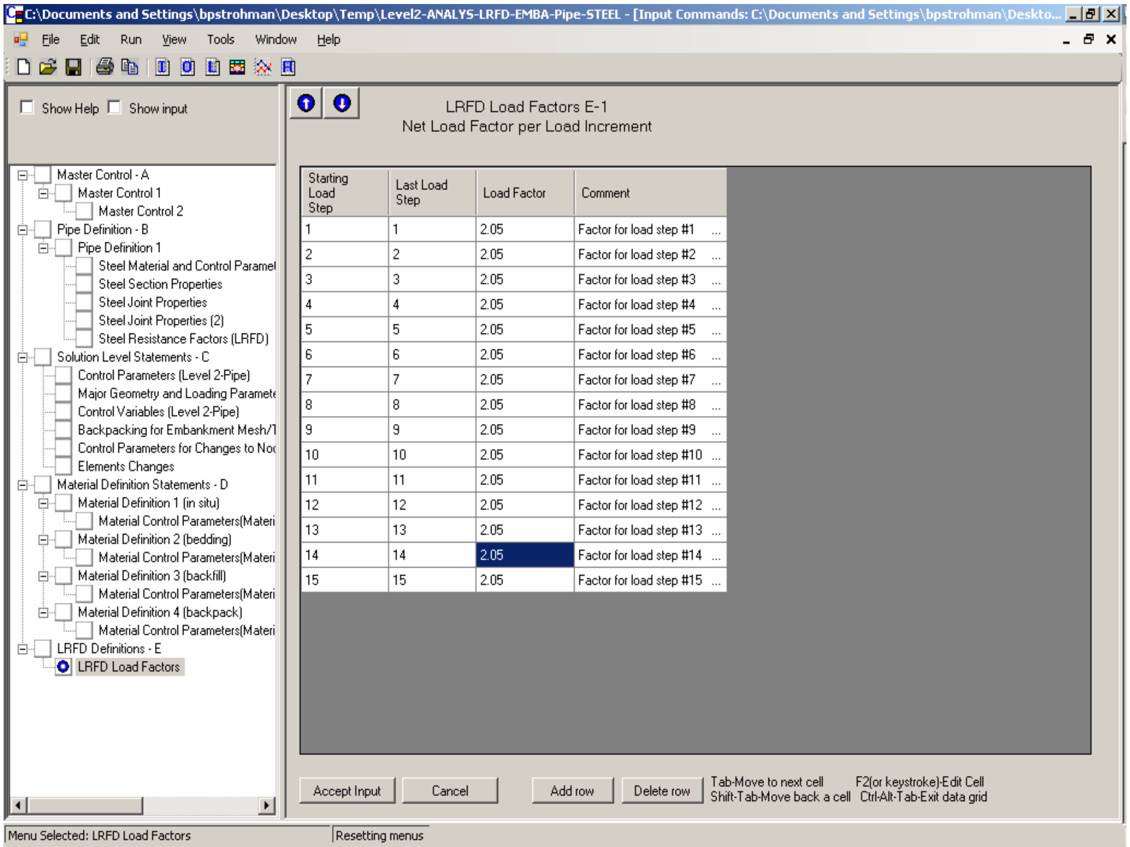


Figure 8-15 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

Under LRFD load factor in Figure 8-15 enter the combined value of the load factor and load modifier, i.e. – 1.95 * 1.05 = 2.05



Starting Load Step	Last Load Step	Load Factor	Comment
1	1	2.05	Factor for load step #1
2	2	2.05	Factor for load step #2
3	3	2.05	Factor for load step #3
4	4	2.05	Factor for load step #4
5	5	2.05	Factor for load step #5
6	6	2.05	Factor for load step #6
7	7	2.05	Factor for load step #7
8	8	2.05	Factor for load step #8
9	9	2.05	Factor for load step #9
10	10	2.05	Factor for load step #10
11	11	2.05	Factor for load step #11
12	12	2.05	Factor for load step #12
13	13	2.05	Factor for load step #13
14	14	2.05	Factor for load step #14
15	15	2.05	Factor for load step #15

Figure 8-16 – Input Screen E-1

When all input is complete, click “Run” and “CANDE-2007” on the main tool bar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “Analysis Complete” window will appear. Click okay and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the text input option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

8.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material zones, boundary conditions, and construction increments may all be added or removed from the plot. The "plotting options" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting options menu and click the check box "Show constr. increment number." Click "ok," set Load Increment to 7 to show the entire mesh (the remaining load is placed above the mesh – see *User Manual*) and click the toolbar icon to turn on the load increments. The mesh plot should look like Figure 8-16.

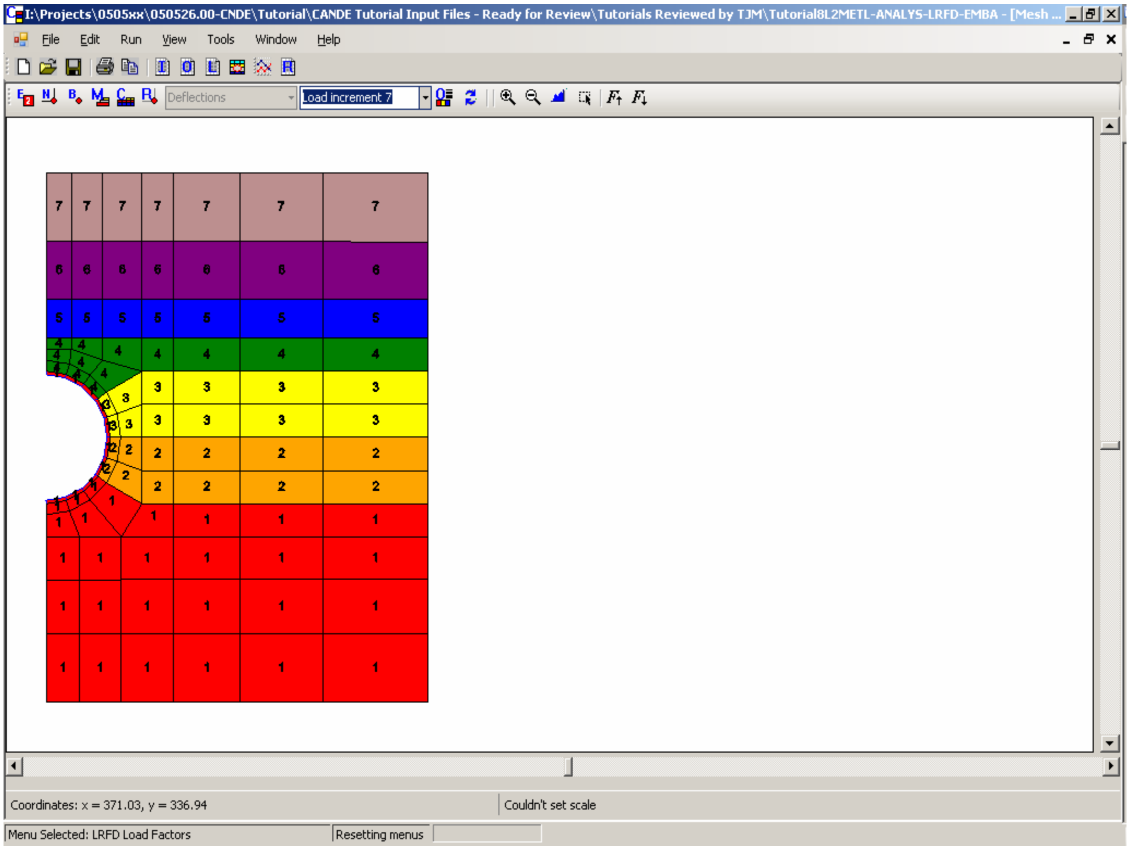


Figure 8-17 – Mesh Plot for Load Steps 1 to 7

To view deflections, open the plotting parameters window and set the deflection magnification factor to 10, click the “Ignore deflections/by load incr.” check box and set the Load Increment to 1 (see Figure 8-17).

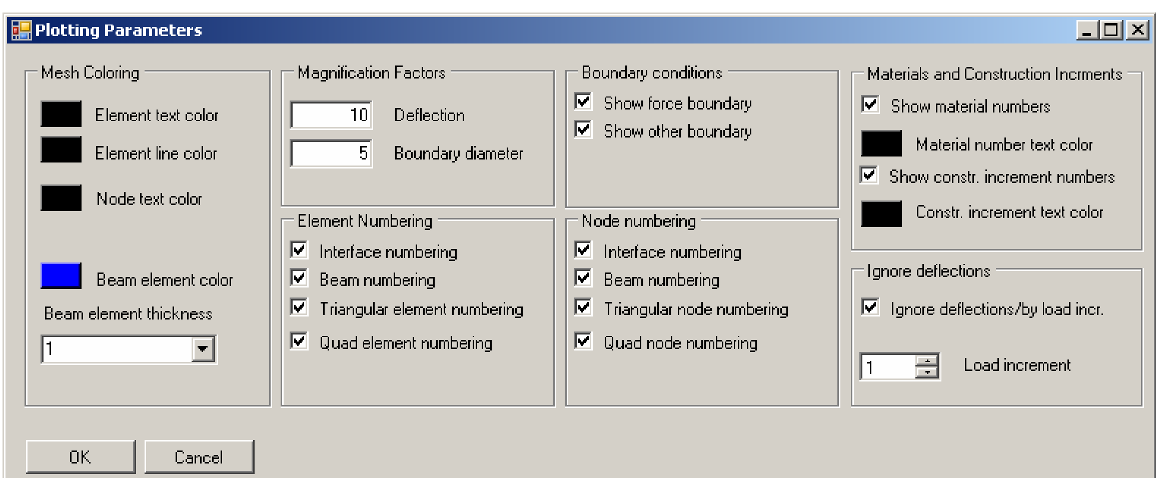


Figure 8-18 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Deflections. Note that if the Load Increment is set to 1, the deflections due to Load Increment 1 are shown, but when the Load Increment is set to 2, the Increment 1 deflections are ignored. Set the increment to 15 and the screen should look like Figure 8-18. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

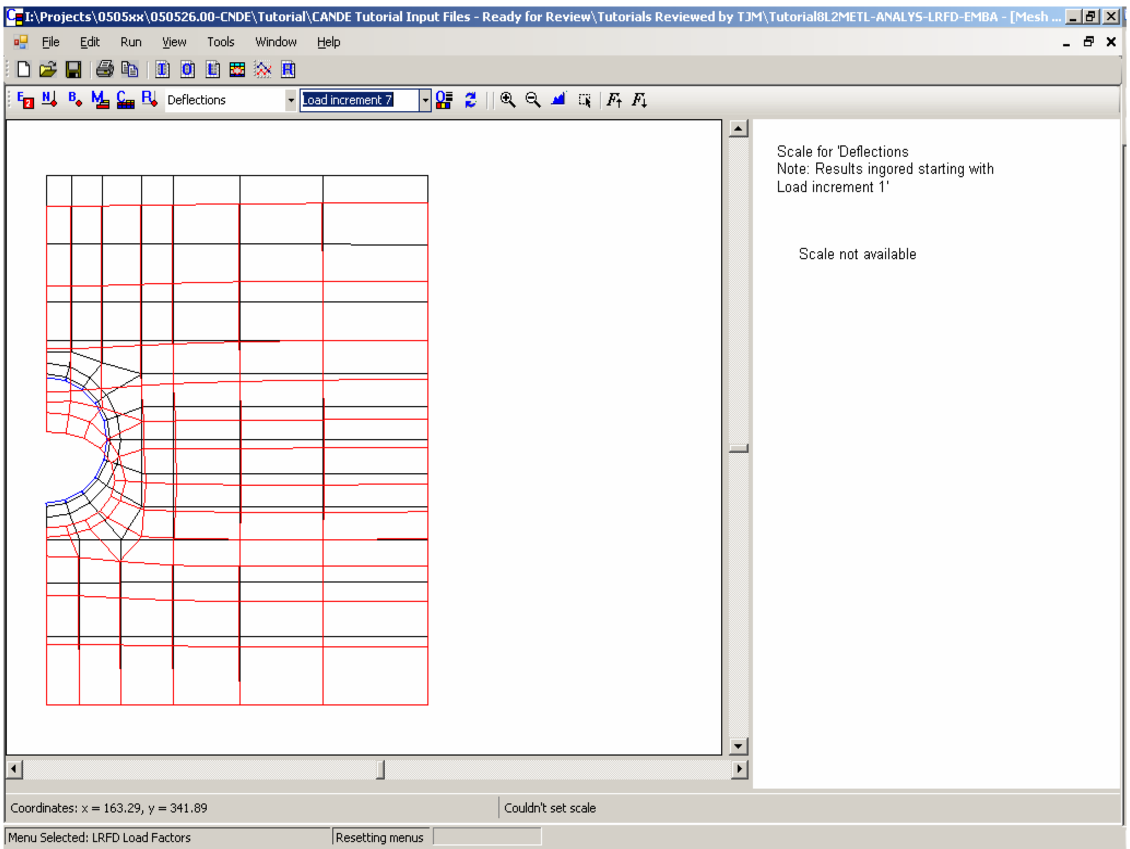


Figure 8-19 – Deflections, Excluding Those Due to Self Weight of In Situ Soil

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 8-19 compares the bending moments after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 8, 12, and 15, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Bending moment” and the screen should appear as Figure 8-19.

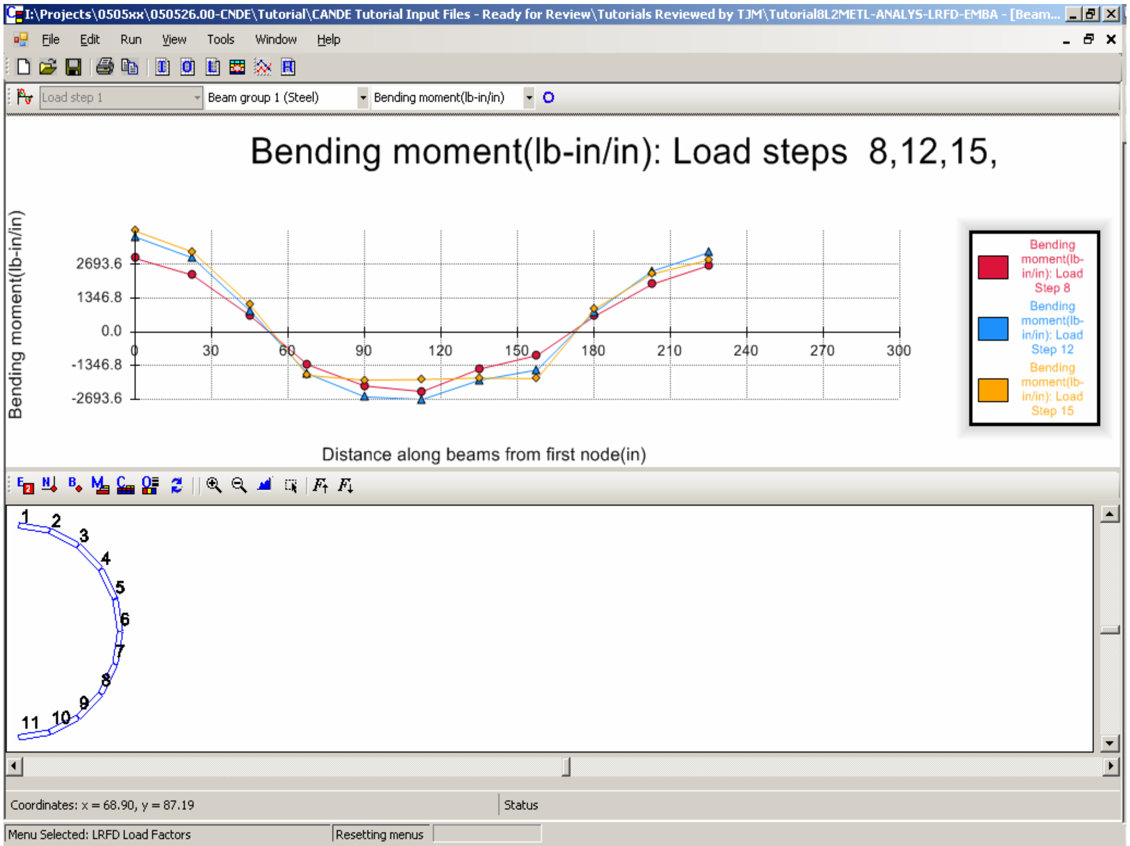


Figure 8-20 – Bending Moments for Load Steps 8, 12, and 15

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 8-20) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 8-20 shows the final assessment printed at the end of the file.

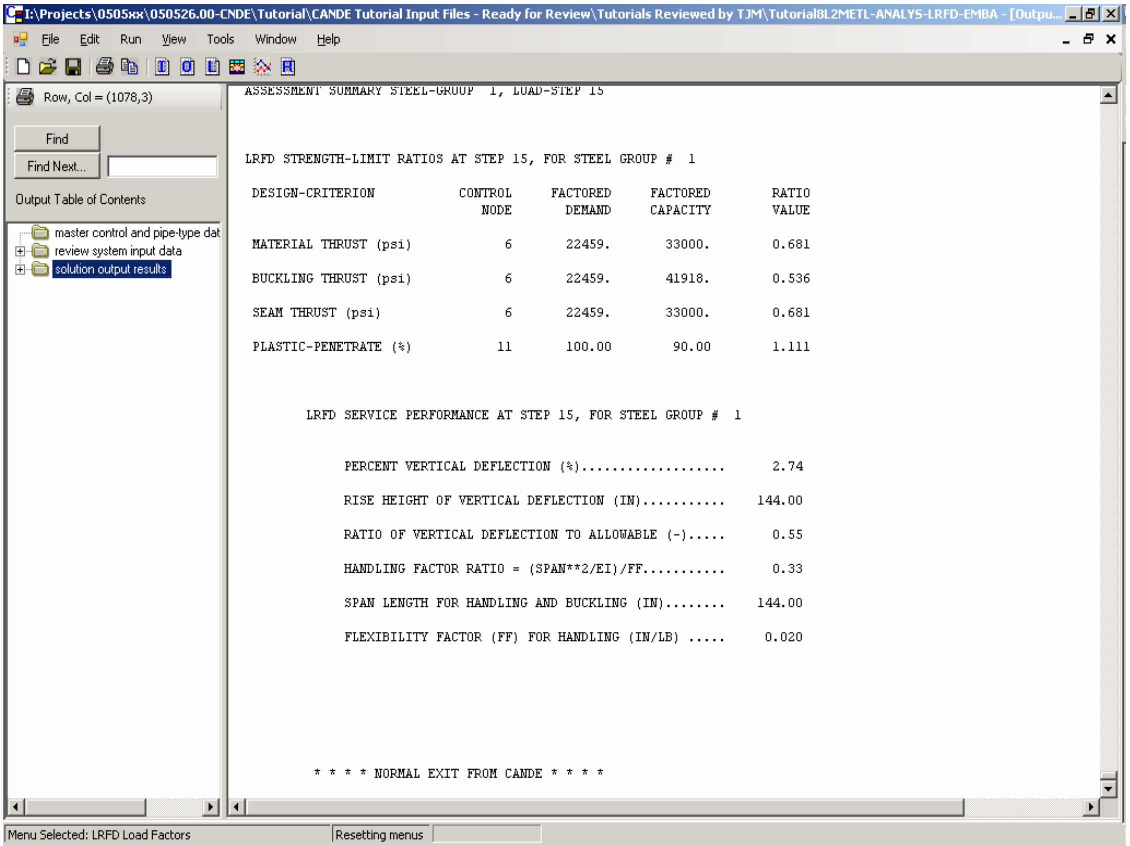


Figure 8-21 – Design Assessment Summary – Load Step 15

One additional tool is the Results Generator which can be used by selecting “View/Results Generator” or the “Start CANDE Results Generator” button. Figure 8-21 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the “Generate/Preview” button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 8-22. The three tabs shown in Figures 8-21 and 8-22 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.
- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.
- Report Preview – displays the desired report.

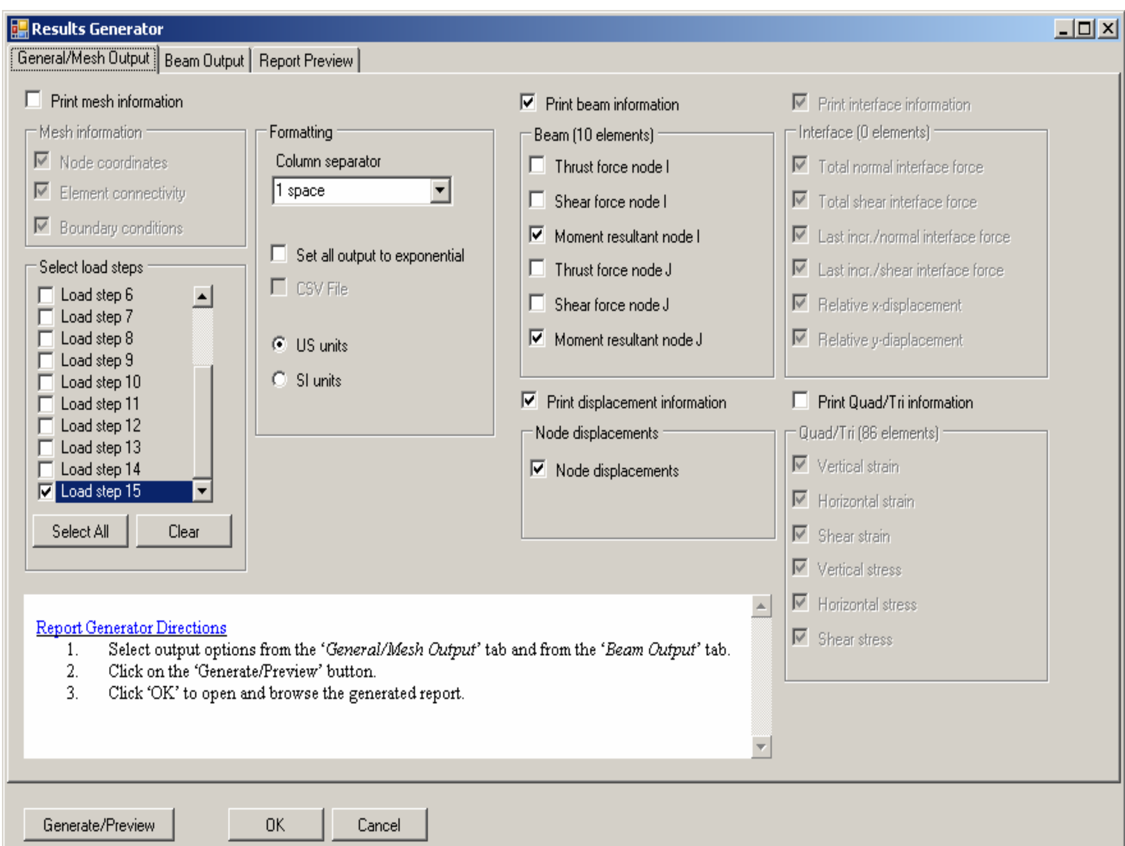
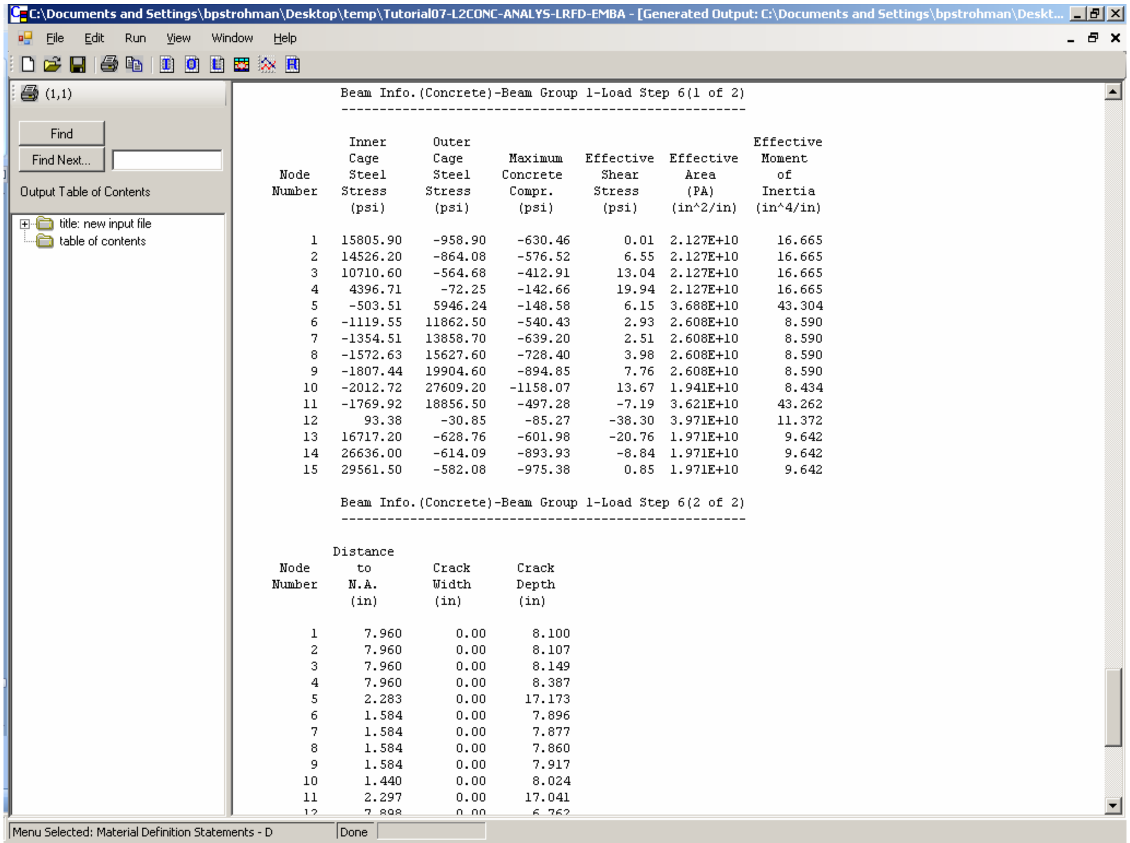


Figure 8-22 – Results Generator Input Screen – Load Step 15 Moments and Deflections



Beam Info.(Concrete)-Beam Group 1-Load Step 6(1 of 2)

Node Number	Inner Cage Steel Stress (psi)	Outer Cage Steel Stress (psi)	Maximum Concrete Compr. (psi)	Effective Shear Stress (psi)	Effective Area (PA) (in^2/in)	Effective Moment of Inertia (in^4/in)
1	15805.90	-958.90	-630.46	0.01	2.127E+10	16.665
2	14526.20	-864.08	-576.52	6.55	2.127E+10	16.665
3	10710.60	-564.68	-412.91	13.04	2.127E+10	16.665
4	4396.71	-72.25	-142.66	19.94	2.127E+10	16.665
5	-503.51	5946.24	-148.58	6.15	3.688E+10	43.304
6	-1119.55	11862.50	-540.43	2.93	2.608E+10	8.590
7	-1354.51	13858.70	-639.20	2.51	2.608E+10	8.590
8	-1572.63	15627.60	-728.40	3.98	2.608E+10	8.590
9	-1807.44	19904.60	-894.85	7.76	2.608E+10	8.590
10	-2012.72	27609.20	-1158.07	13.67	1.941E+10	8.434
11	-1769.92	18856.50	-497.28	-7.19	3.621E+10	43.262
12	93.38	-30.85	-85.27	-38.30	3.971E+10	11.372
13	16717.20	-628.76	-601.98	-20.76	1.971E+10	9.642
14	26636.00	-614.09	-893.93	-8.84	1.971E+10	9.642
15	29561.50	-582.08	-975.38	0.85	1.971E+10	9.642

Beam Info.(Concrete)-Beam Group 1-Load Step 6(2 of 2)

Node Number	Distance to N.A. (in)	Crack Width (in)	Crack Depth (in)
1	7.960	0.00	8.100
2	7.960	0.00	8.107
3	7.960	0.00	8.149
4	7.960	0.00	8.387
5	2.283	0.00	17.173
6	1.584	0.00	7.896
7	1.584	0.00	7.877
8	1.584	0.00	7.860
9	1.584	0.00	7.917
10	1.440	0.00	8.024
11	2.297	0.00	17.041

Figure 8-23 – Results Generator – Load Step 15 Beam Stresses

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 9

May 2008

This page intentionally left blank

9. CANDE TEST PROBLEM 9

9.1 Problem Definition

Analyze a 217-inch span (82-inch rise) 3-segment type corrugated steel long span arch supported on spread footings with 3 ft of fill over the top of the arch, using Working Stress (service) analysis. The problem is shown schematically in Figure 9-1. The analysis will be with Level 2, using an automated finite element arch mesh for a trench installation having interface elements. The automated finite element mesh will be modified using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch.

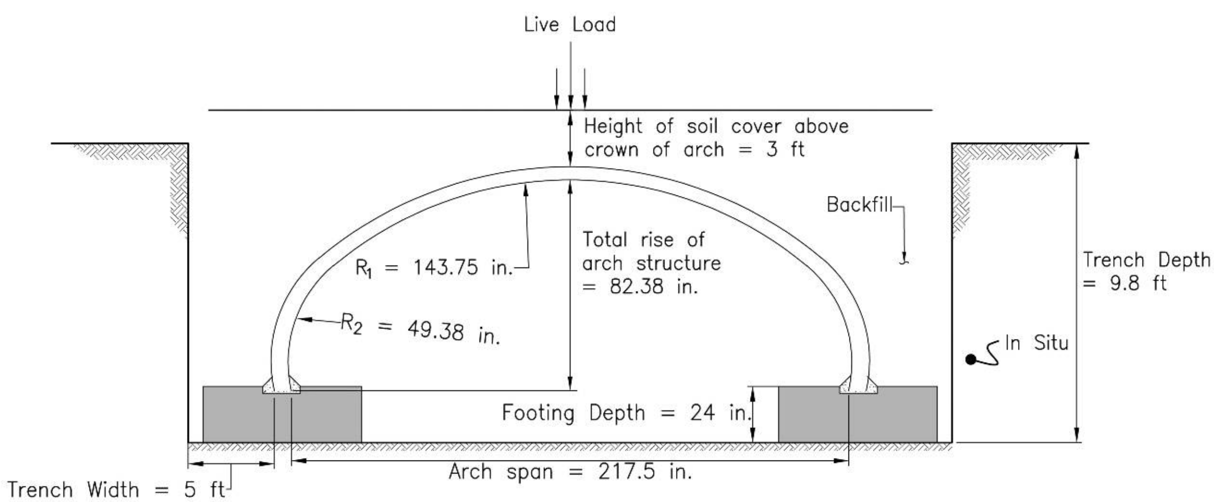


Figure 9-1 Details of Problem 9

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - Service

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Arch mesh

Soil mesh pattern - Trench

MOD-Make changes to the basic mesh - check on (problem requires using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch).

Number of new loading/boundary conditions to be added - 8 (for Live Load calculation see Figure 9-12).

Pipe material type - Steel

Joint slip - No joint slip

Soil parameters - Canned Duncan/Selig soil models except as noted:

In situ soil - ML 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Footing - Canned Linear Elastic - Young's modulus = 3,600,000 psi, Poisson's ratio = 0.17

Backfill soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils and 150 lb/ft^3 for the footing

Density of steel - 0.284 $lb/in.^3$

Culvert material behavior - Bilinear stress/strain

Analysis mode - Large deformation/buckling

Area of pipe wall unit length - 0.267 $in.^2/in.$ (Assumes 0.218 in. thick with 6 in. x 2 in. corrugations)

Moment of inertia of pipe wall/unit length - 0.127 $in.^4/in.$ (Assumes 0.218 in. thick with 6 in. x 2 in. corrugations)

Section modulus of pipe wall/unit length - 0.115 $in.^3/in.$ (Assumes 0.218 in. thick with 6 in. x 2 in. corrugations)

Number of construction steps - See *User Manual*, Chapter 5, C-2 – The default mesh provides 8 construction steps to the top of the trench. Use an additional construction step to obtain a height of fill of 3 ft. Use an additional 4 steps for the live load (applying load in steps results in fewer convergence issues), making a total of 13 construction steps.

Height of soil cover above crown of arch - 3 ft

Density of soil above truncated mesh - 120 lb/ft^3

Trench depth - 9.8 ft

Trench width - 5 ft

Total rise of arch structure - 82.38 in.

One-half of arch span at footing level - 108.75 in.

Footing depth - 24 in.

Outside footing width - 30 in.

Inside footing width - 30 in.

Radius of top arch (segment 1) - R1 - 143.75 in.

Angle for R1 segment - 40 degrees

Radius of second segment - R2 - 49.38 in.

Angle for R2 segment - 62.87 degrees

Interface material properties - The following summarizes material property definitions for the interface elements:

Interface number	**Angle from x-axis to normal interface (degrees) ***	**Coefficient of friction between nodes i and j**	**Tensile breaking force of contact nodes**
1	90	0.3	10
2	86.67	0.3	10
3	83.33	0.3	10
4	80.00	0.3	10
5	76.67	0.3	10
6	73.33	0.3	10
7	70.00	0.3	10
8	66.67	0.3	10
9	63.33	0.3	10
10	60.00	0.3	10
11	56.67	0.3	10
12	53.33	0.3	10
13	50.00	0.3	10
14	41.02	0.3	10
15	32.04	0.3	10
16	23.06	0.3	10
17	14.07	0.3	10
18	5.09	0.3	10
19	-3.89	0.3	10

* *For an example calculation depicting how these interface angles were computed and/or an explanation of the shortcut method for defining interface material properties, see CANDE Tutorial Problem 6.*

9.2 Creating the CANDE Input Document

Figures 6-2 through 6-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking "Next" on Screen 3, CANDE will display a screen indicating that an input path is initiated - click "Finish." The user is prompted for a filename and directory.

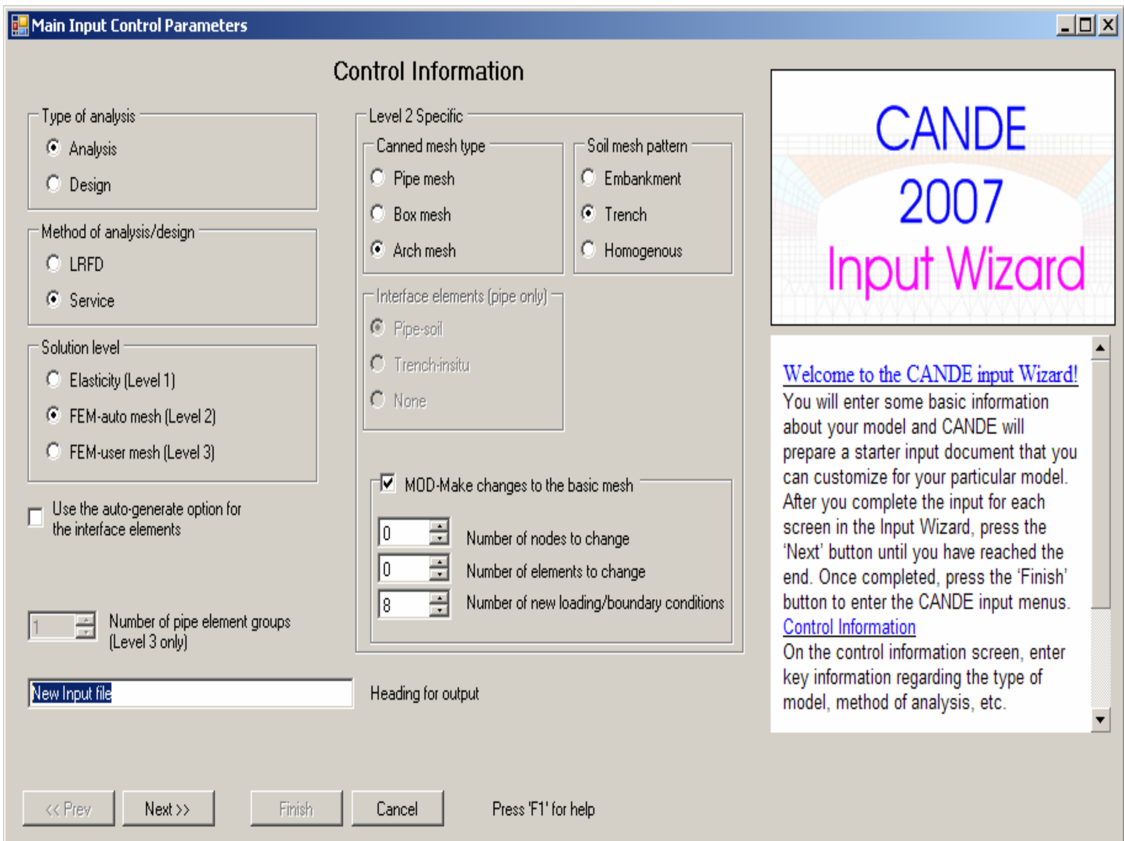


Figure 9-2 – Input Wizard, Screen 1

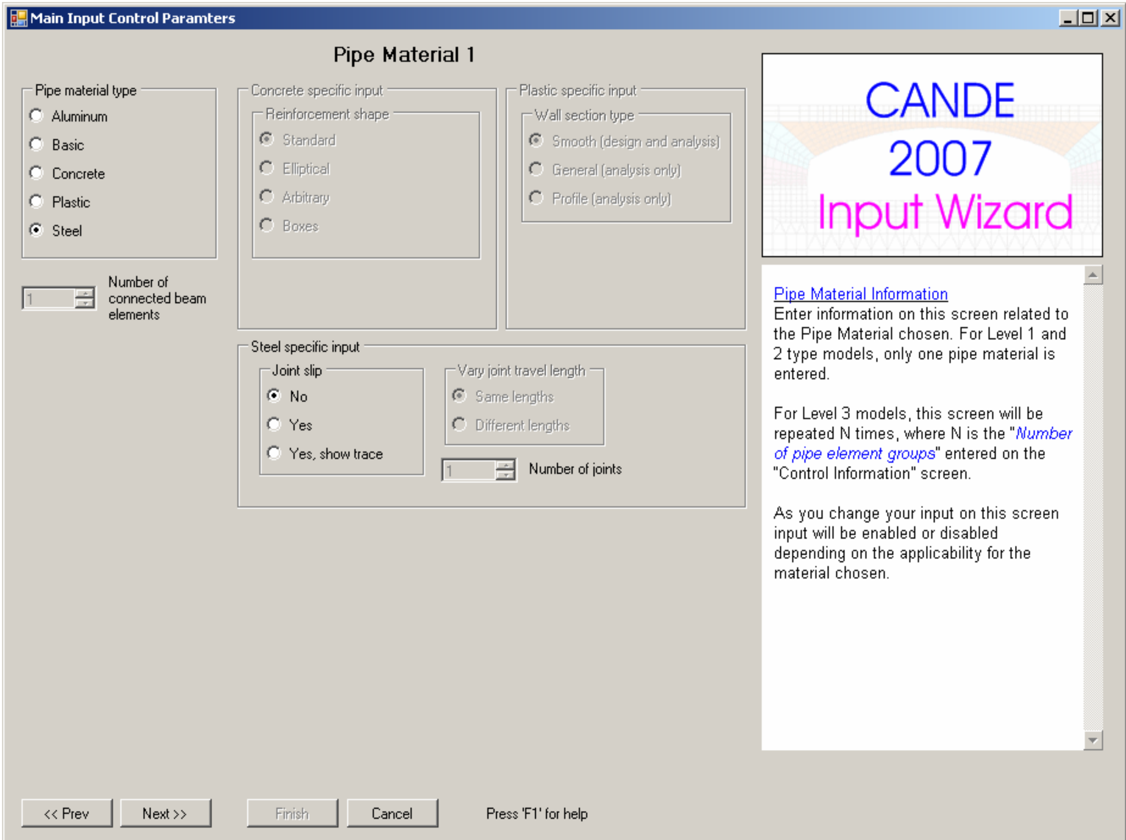


Figure 9-3 – Input Wizard, Screen 2

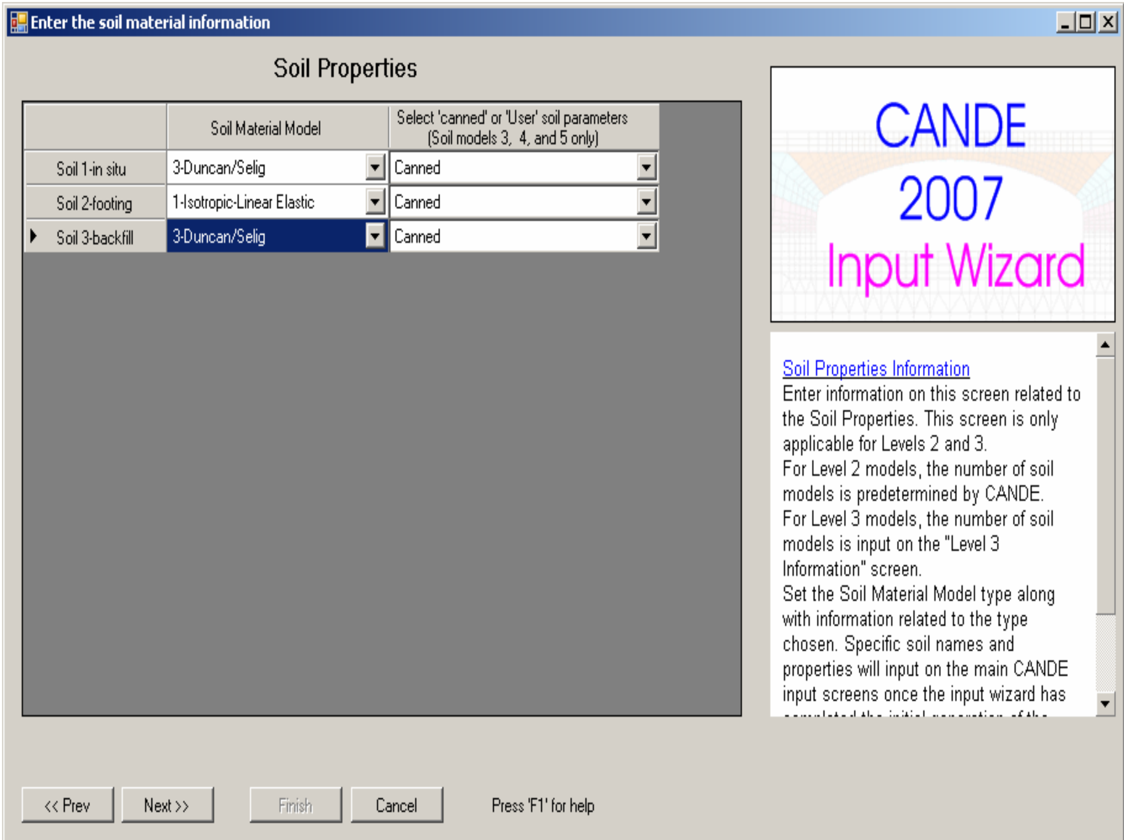


Figure 9-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 9-5. Enter an appropriate heading for output and click “Accept Input.”

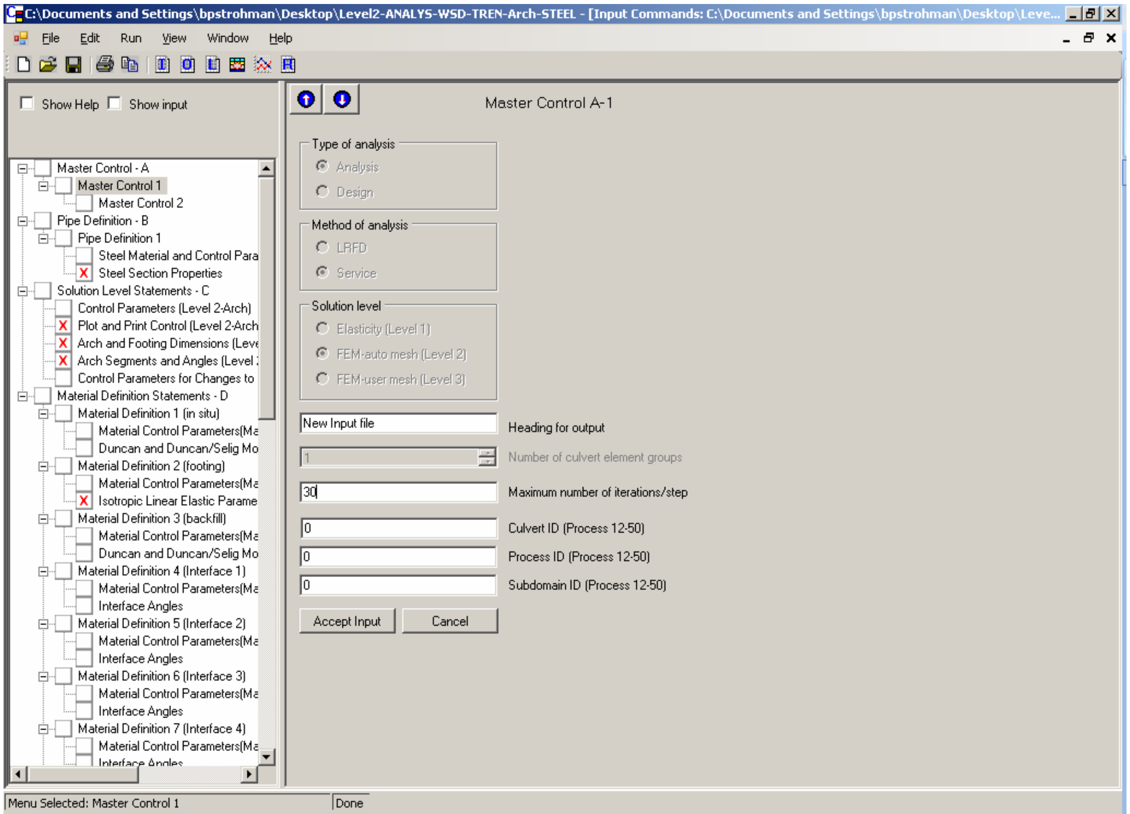


Figure 9-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 9-5 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 9-6 through 9-10 and Figures 9-14 through 9-16 show the completed input for the standard screens requiring data for the tutorial, except that only one material definition and interface material definition screen are shown. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data. Figures 9-11 through 9-13 show the live load calculation and use of the Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch.

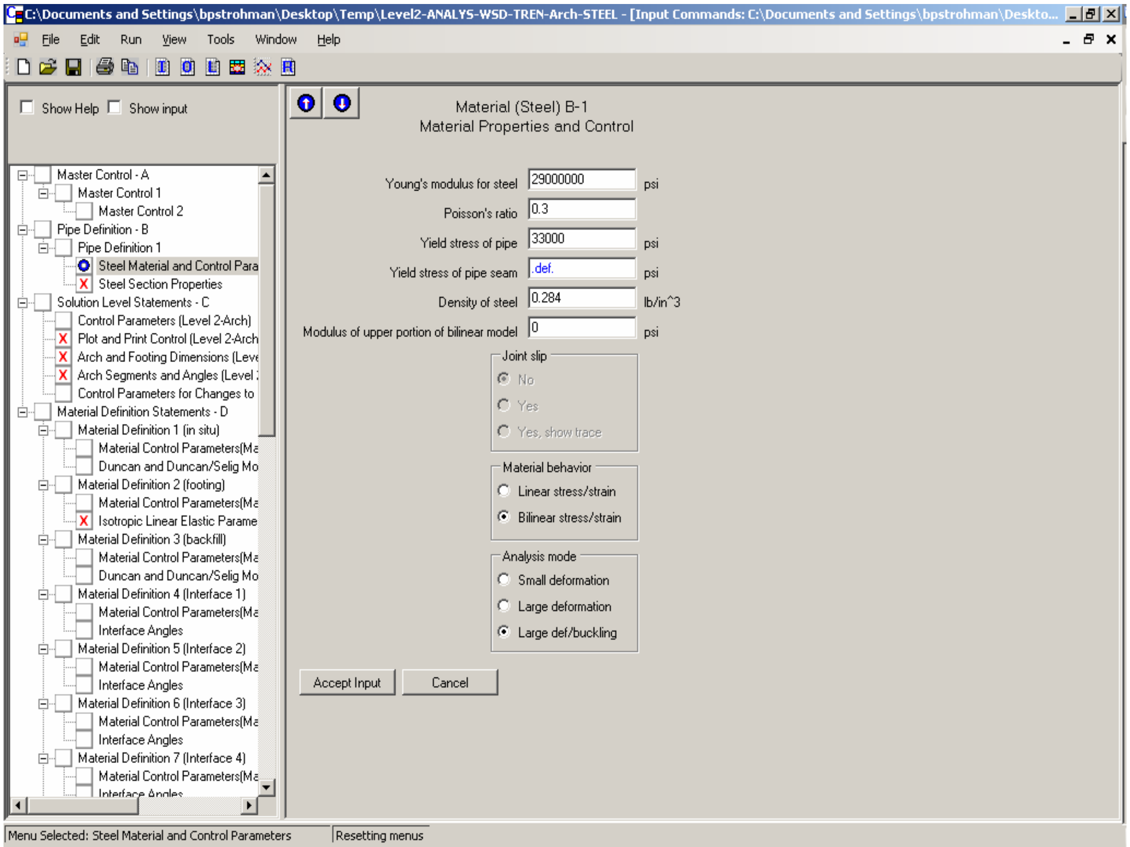


Figure 9-6 – Input Screen B-1

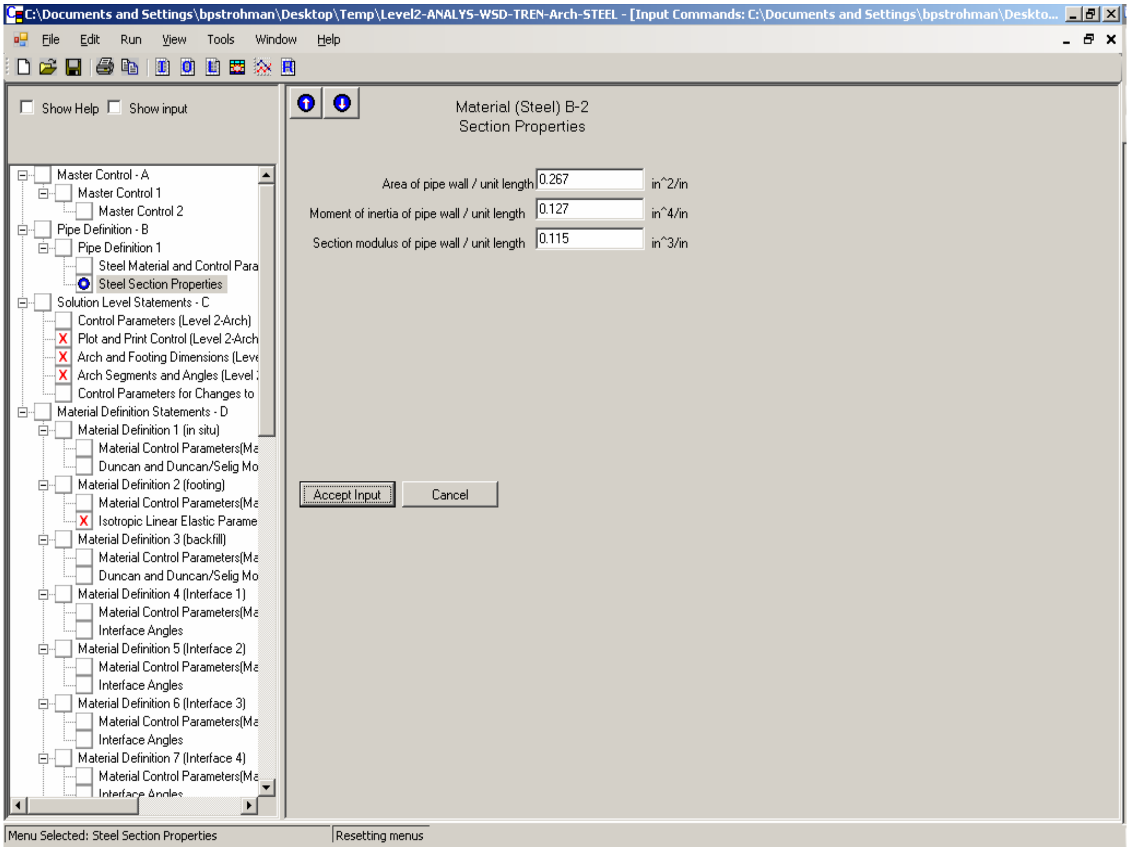


Figure 9-7 – Input Screen B-2

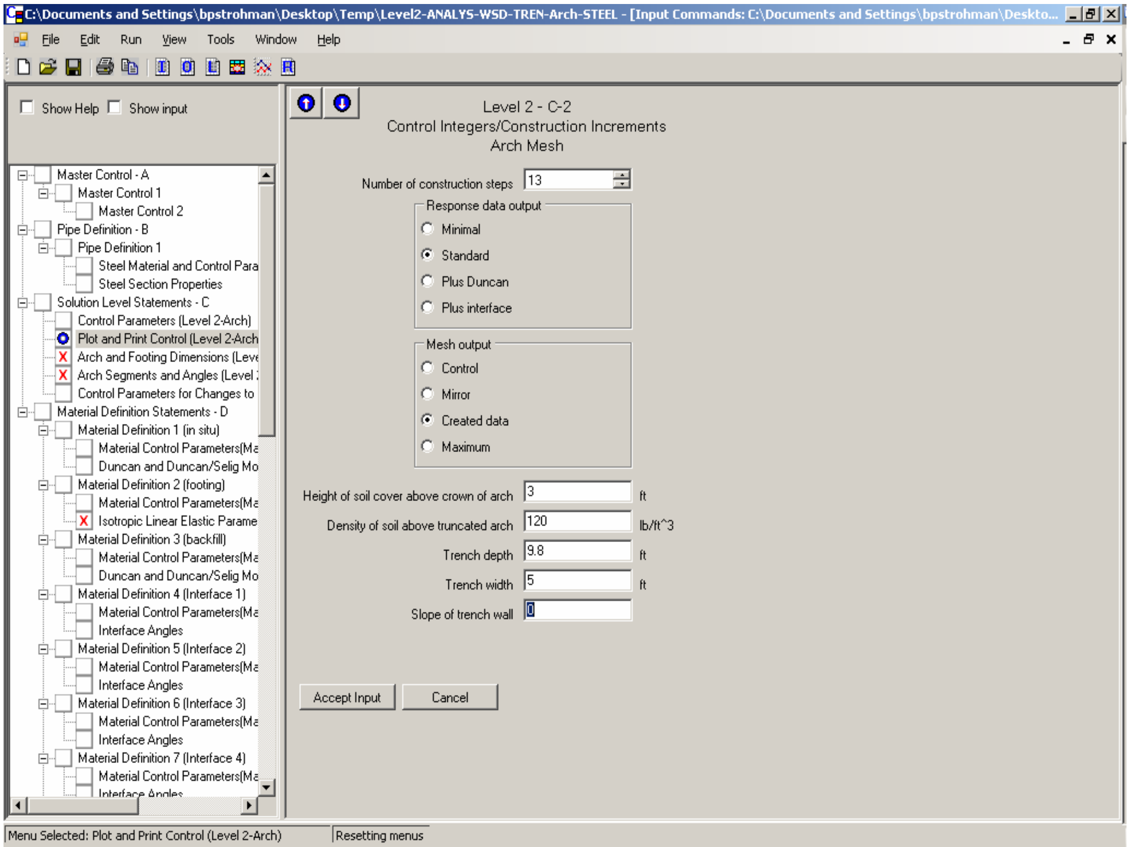


Figure 9-8 – Input Screen C-2

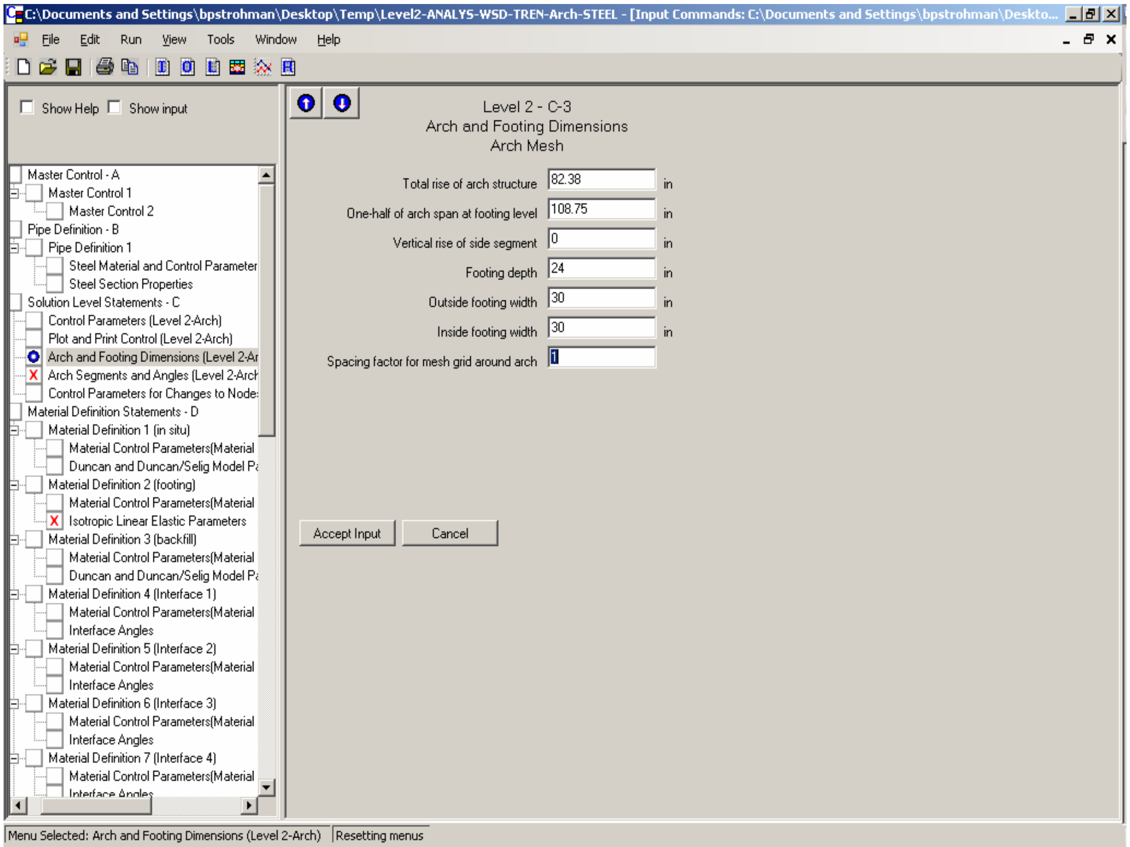


Figure 9-9 – Input Screen C-3

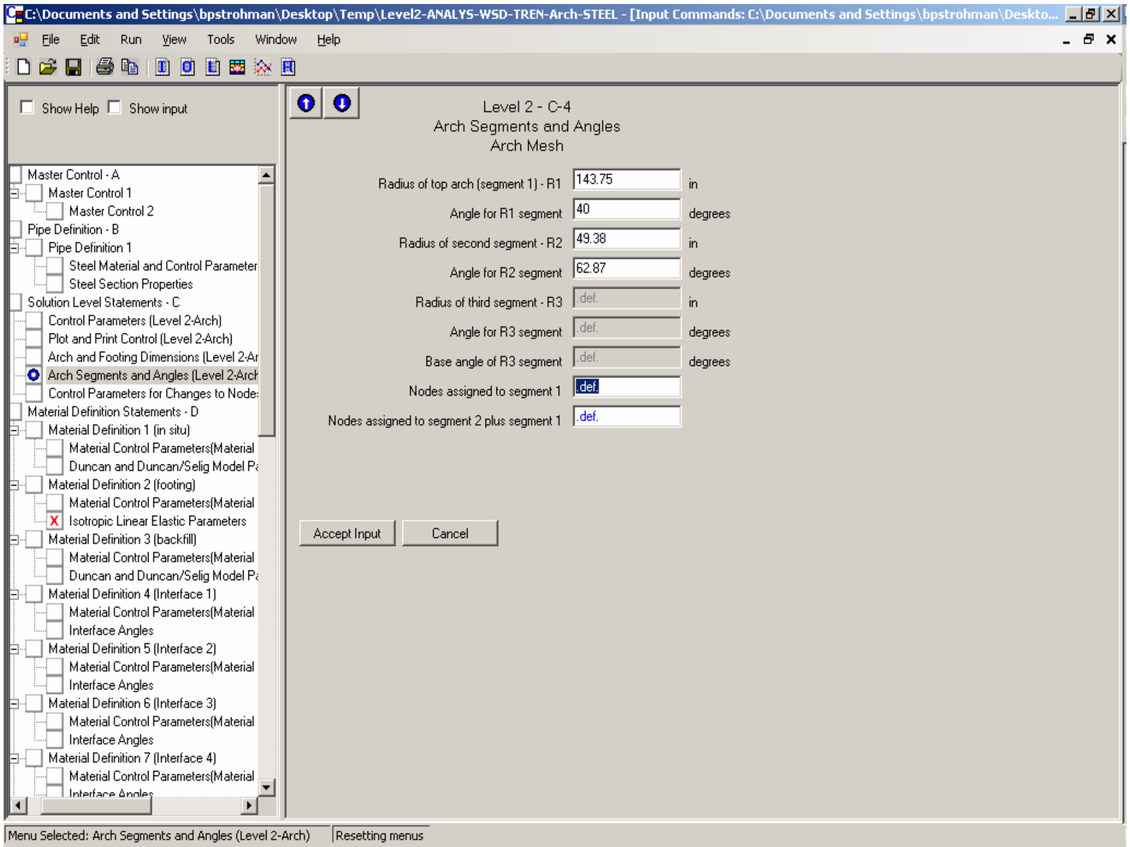


Figure 9-10 – Input Screen C-4

The automated finite element mesh will be modified using Level 2-extended to apply point loads depicting a LRFD design truck at the ground surface above the crown of the arch. Figures 9-11 and 9-13 show the completed input screens for the Level 2-extended. Figure 9-12 shows the calculation of the two dimensional out-of-plane live load through soil for use in the CANDE finite element model for a Design Truck (HS-20) at the ground surface 3 ft above the crown of the arch traveling parallel to the span of the culvert.

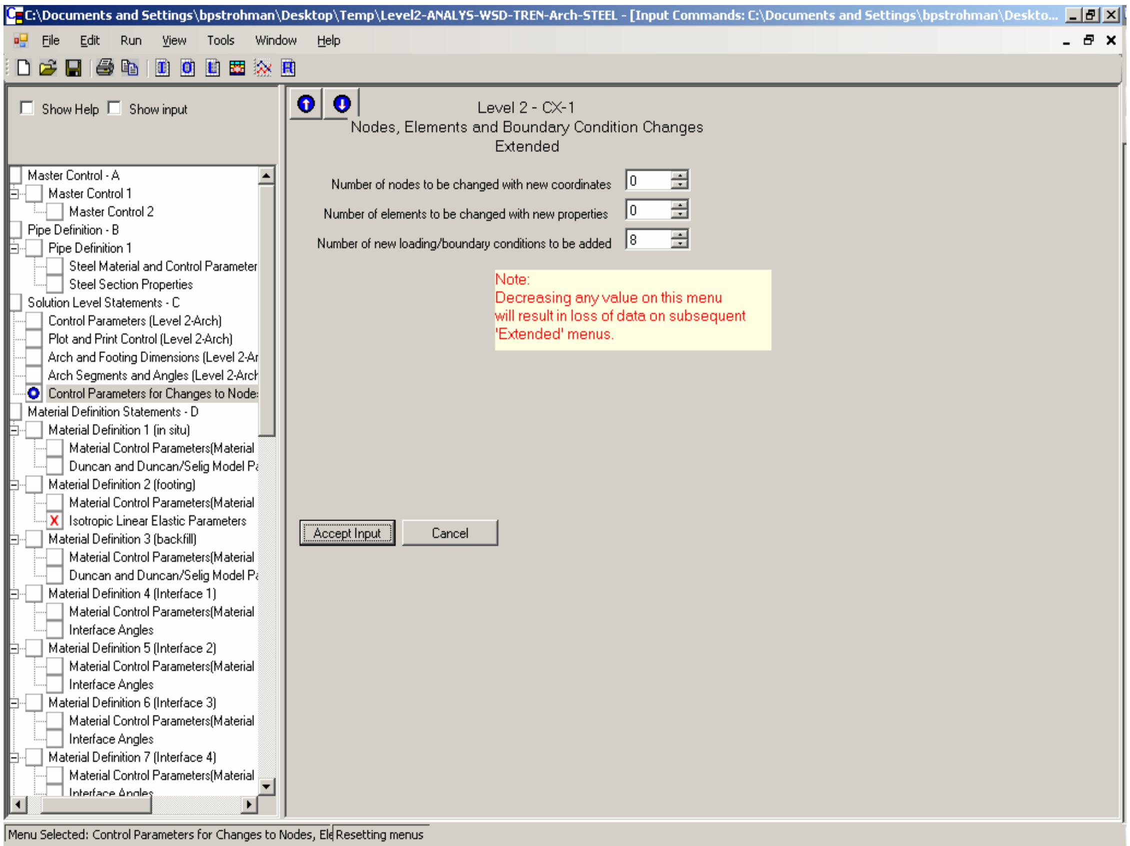


Figure 9-11 – Input Screen CX-1

Live Load Calculation

DISTRIBUTION OF LIVE LOAD THROUGH EARTH FILLS - AASHTO LRFD

Purpose:
This sheet demonstrates calculating the two-dimensional out-of-plane distribution of live load through soil for use with CANDE FEM for a Design Truck (HS-20) traveling parallel to the span of the culvert. AASHTO LRFD code references are shown in parenthesis.

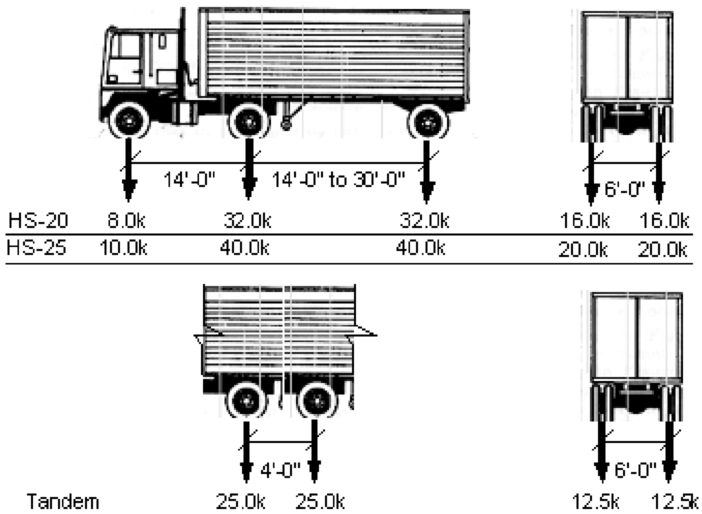


Figure 1. AASHTO Design Truck (HS-20), Design Truck plus 25% (HS-25), and Design Tandem

NOTES

1. Input values shown in grey.
2. := indicates definition of a variable.
3. = prints a previously defined variable.

GENERAL PROPERTIES

Depth of Fill/ Cover.. $H_E := 3ft$

Bridge Dimensions:

Bridge Span... $Span := 18ft + 1.5in$

Design Truck:

Axle Load... $P_{axle} := 32kips$

Tire Load.. $P_{tire} := \frac{P_{axle}}{2}$

Axle Width (distance between tires)...................... $w_{axle} := 6ft$

Tire Contact Area (LRFD 3.6.1.2.5)

Contact Width.. $w_{tire} := 20in$

Contact Length... $L_{tire} := 10in$

DESIGN FACTORS

Live Load Distribution Factor (LRFD 3.6.1.2.6)...................... $LLDF := 1.15$ (Select Granular Backfill)

Multiple Presence of Live Load (LRFD 3.6.1.1.2)................... $mpf := 1.20$

Live Load Factor (Service I).. $LL := 1.0$

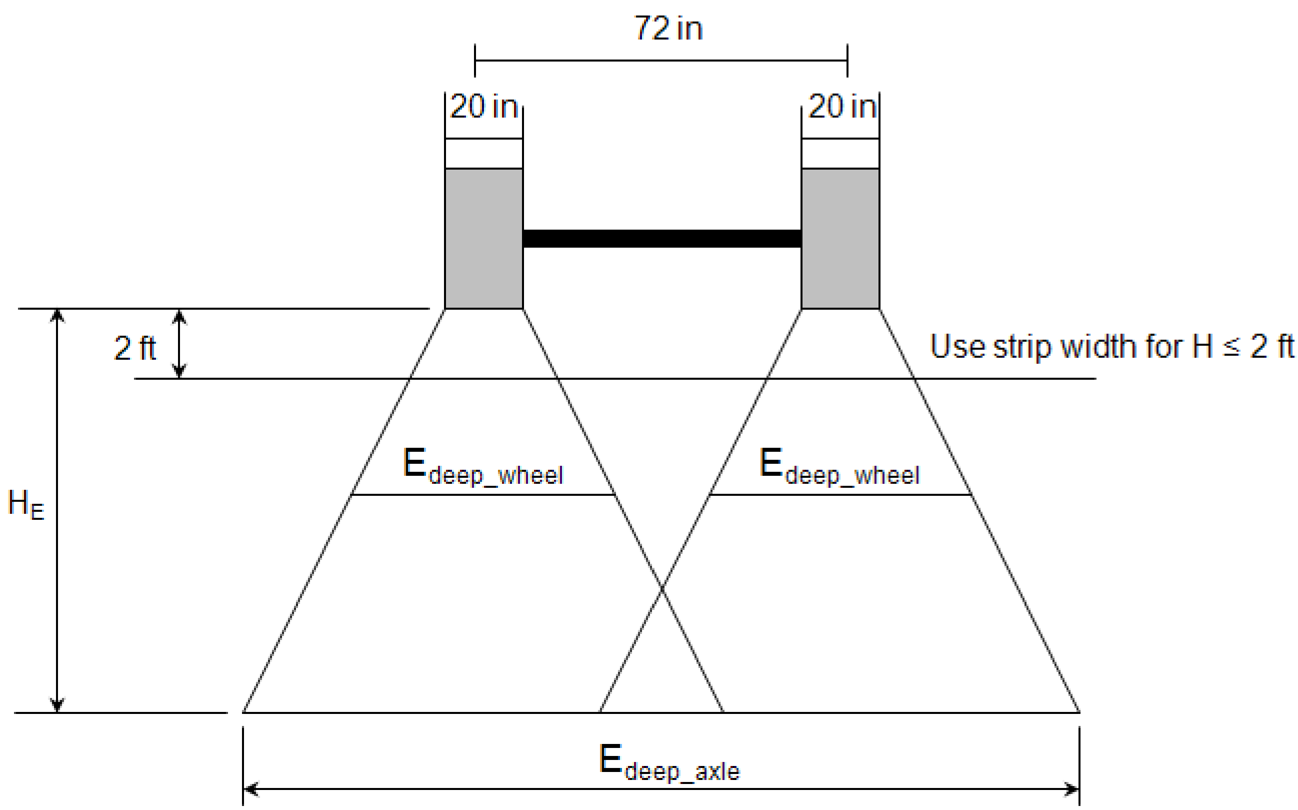
Figure 9-12 – Live Load Calculation

Impact for Buried Components (LRFD 3.6.2.2)

Dynamic Allowance.. $IM := 33 \cdot \left(1.0 - 0.125 \cdot \frac{H_E}{ft}\right)$

Impact Factor.. $I_{imp} := max\left[1.0, \left(1 + \frac{IM}{100}\right)\right]$ $\qquad I_{imp} = 1.206$

<u>FOR DEPTH OF FILL GREATER THAN 2FT(LRFD 3.6.1.2.6)</u>



Wheel Load Equivalent Distribution Width............. $E_{deep\_wheel} := w_{tire} + LLDF \cdot H_E$ $\qquad E_{deep\_wheel} = 61.4in$

Axle Load Equivalent Distribution Width................ $E_{deep\_axle} := w_{axle} + w_{tire} + LLDF \cdot H_E$ $\qquad E_{deep\_axle} = 133.4in$

Wheel Live Load Equivalent Distribution.............
(no overlap) $LL_{deep\_wheel} := \frac{P_{tire} \cdot I_{imp} \cdot mpf}{E_{deep\_wheel}}$ $\qquad LL_{deep\_wheel} = 377.2\frac{lbf}{in}$

Axle Live Load Equivalent Distribution.................
(overlap) $LL_{deep\_axle} := \frac{P_{axle} \cdot I_{imp} \cdot mpf}{E_{deep\_axle}}$ $\qquad LL_{deep\_axle} = 347.2\frac{lbf}{in}$

Determine Controlling Distribution.....................
(check wheel overlap)

$$LL_{deep} := \begin{cases} LL_{deep\_wheel} & \text{if } E_{deep\_wheel} \le \frac{E_{deep\_axle}}{2} \\ LL_{deep\_axle} & \text{otherwise} \end{cases}$$

$LL_{deep} = 377.2\frac{lbf}{in}$

Figure 9-12 – Live Load Calculation (continued)

FOR DEPTH OF FILL LESS THAN 2FT (LRFD 4.6.2.10)

Equivalent Axle Distribution Width........................... $E_{shallow} := \left(96 + 1.44\frac{Span}{ft}\right)\cdot in$ $E_{shallow} = 122.1in$

Axle Live Load Distribution.................................... $LL_{shallow} := \frac{P_{axle} \cdot I_{imp} \cdot mpf}{E_{shallow}}$ $LL_{shallow} = 379.4\frac{lbf}{in}$

TWO-DIMENSIONAL LIVE LOAD - (SERVICE I)

Two Dimensional Live .... $LL_{2D} := \begin{vmatrix} LL_{shallow} \text{ if } H_E \le 2ft \\ LL_{deep} \text{ if } H_E > 2ft \end{vmatrix}$ $LL_{2D} = 377.2\frac{lbf}{in}$

If Modeling Full Structure:

Number of Elements Over Wheel Length.................. $n_{elems} := 2$ (3 nodes)

use equal mesh spacing

Load per Interior Node.. $l_{int.node} := \frac{LL_{2D}}{n_{elems}}$ $l_{int.node} = 188.6\frac{lbf}{in}$

Load per Exterior Node.. $l_{ext.node} := \frac{l_{int.node}}{2}$ $l_{ext.node} = 94.3\frac{lbf}{in}$

Apply Wheel Load in Increments...........................
(fewer convergence issues) $n_{con.wheel} := 4$

Load per Interior Node per
Construction Increment... $l_{int.node\_per\_con\_incr} := \frac{LL_{2D}}{n_{con.wheel} \cdot n_{elems}}$

$l_{int.node\_per\_con\_incr} = 47.15\frac{lbf}{in}$

For this problem, the interior node is on the plane of symmetry.
Use half of the load above if modeling half the structure.

Load per Exterior Node per
Construction Increment... $l_{ext.node\_per\_con\_incr} := \frac{l_{int.node}}{2 \cdot n_{con.wheel}}$

$l_{ext.node\_per\_con\_incr} = 23.57\frac{lbf}{in}$

Figure 9-12 – Live Load Calculation (continued)

Figure 9-13 demonstrates adding nodal loads to 2 nodes to represent a LRFD design truck at the ground surface 3 ft above the crown of the arch. The node numbers are identified in Chapter 5, C-4 of the *User Manual* or can be obtained by making a trial run without the applied load and then using the mesh plot.

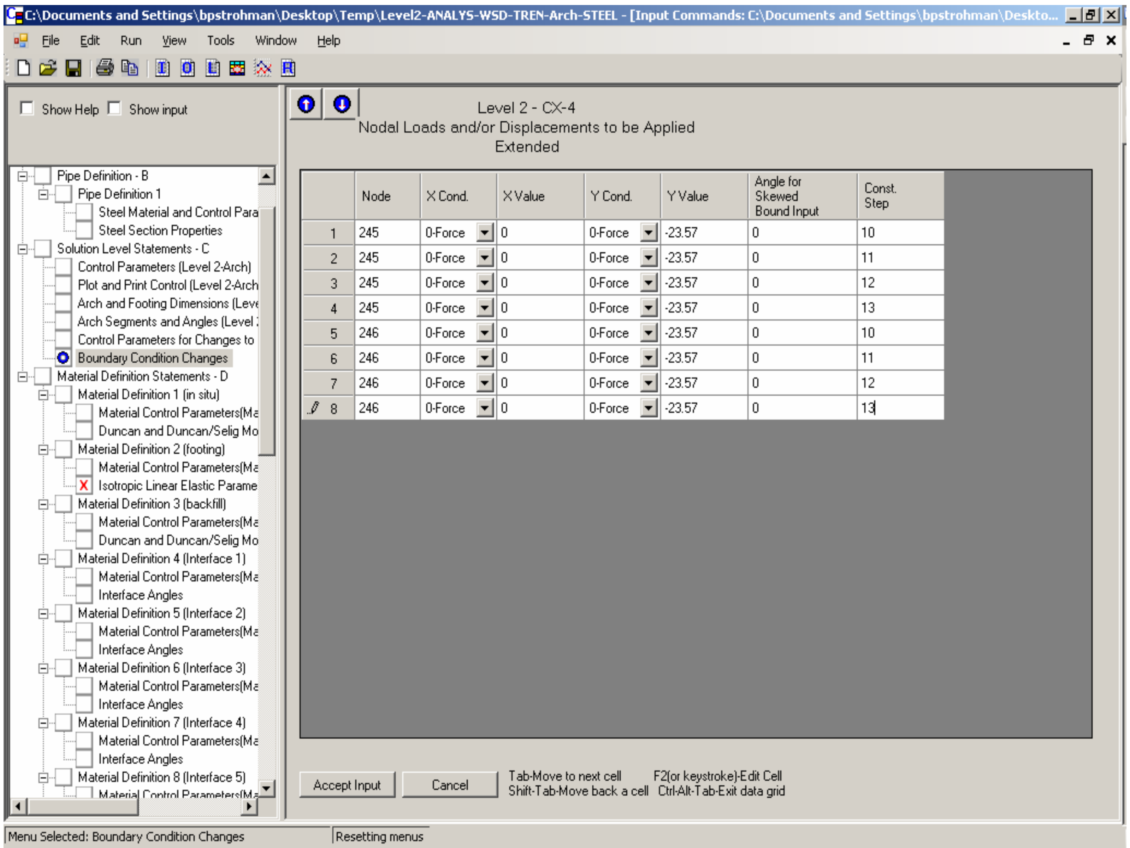


Figure 9-13 – Input Screen CX-4

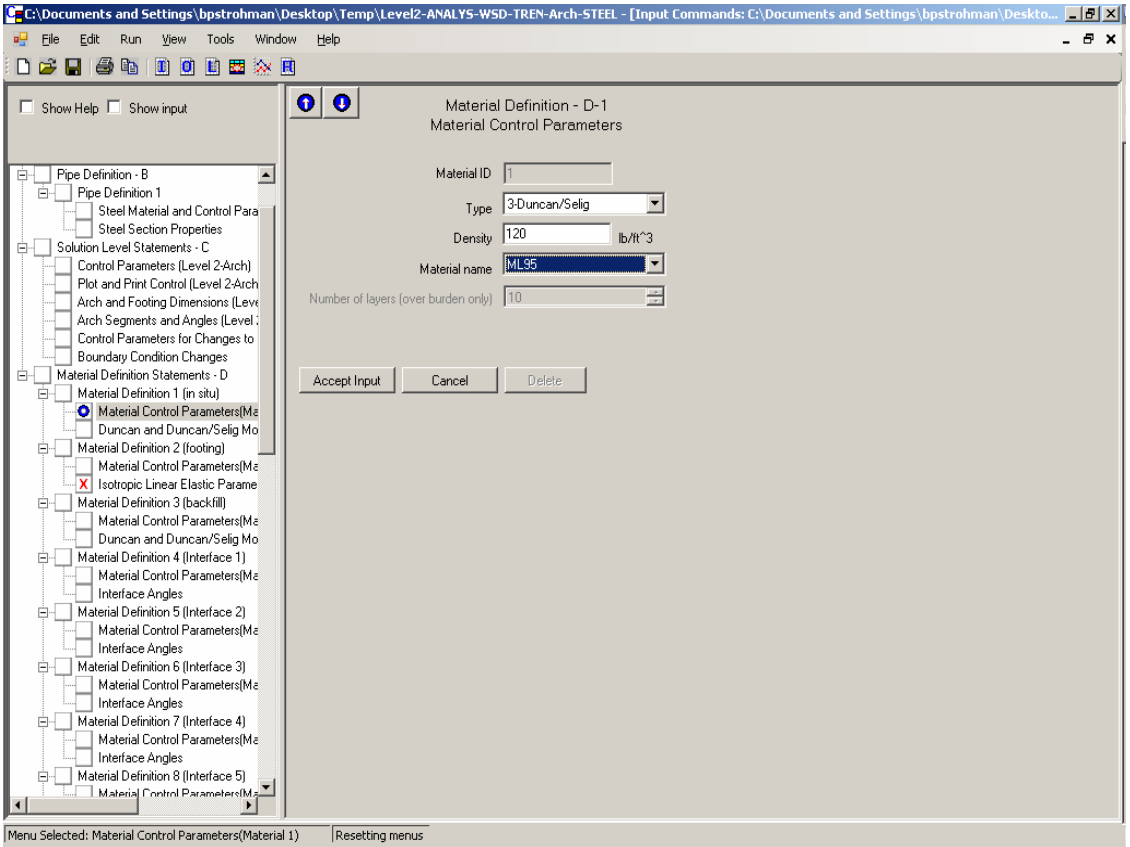


Figure 9-14 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

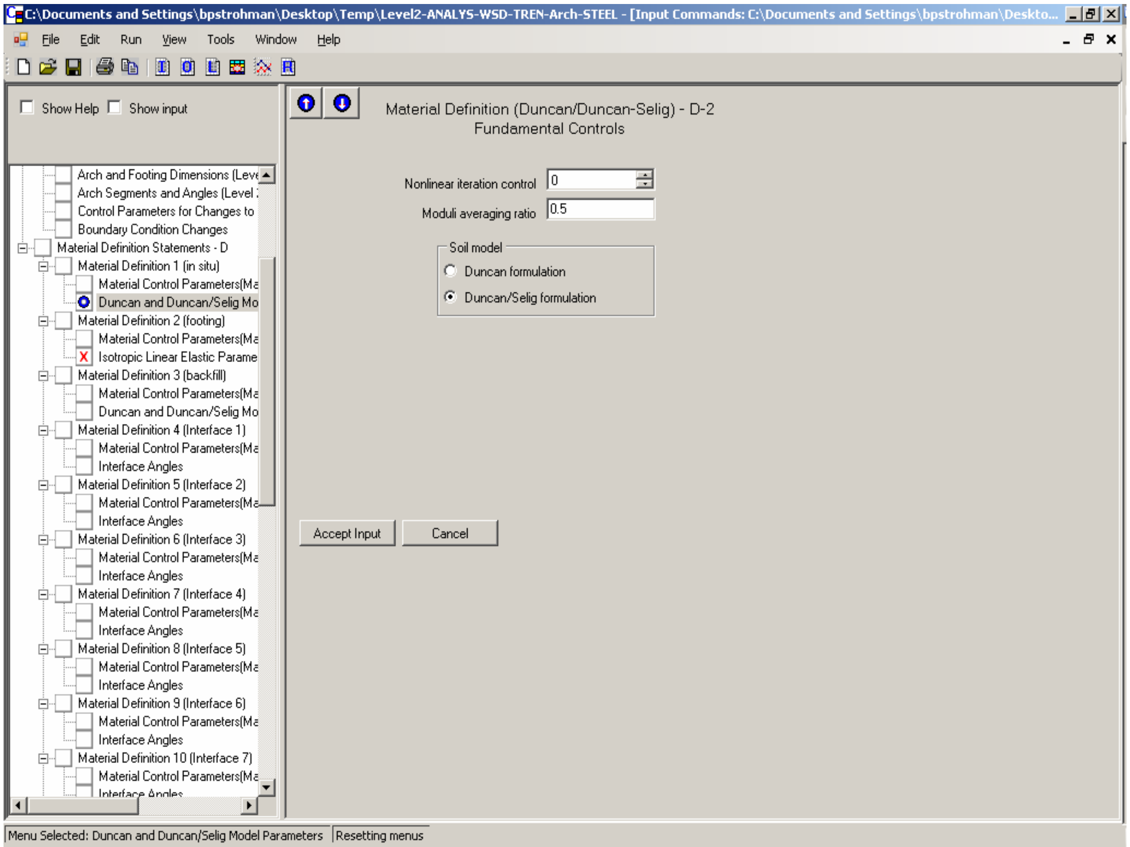


Figure 9-15 – Input Screen D-2 for Material 1 – In Situ Soil

(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

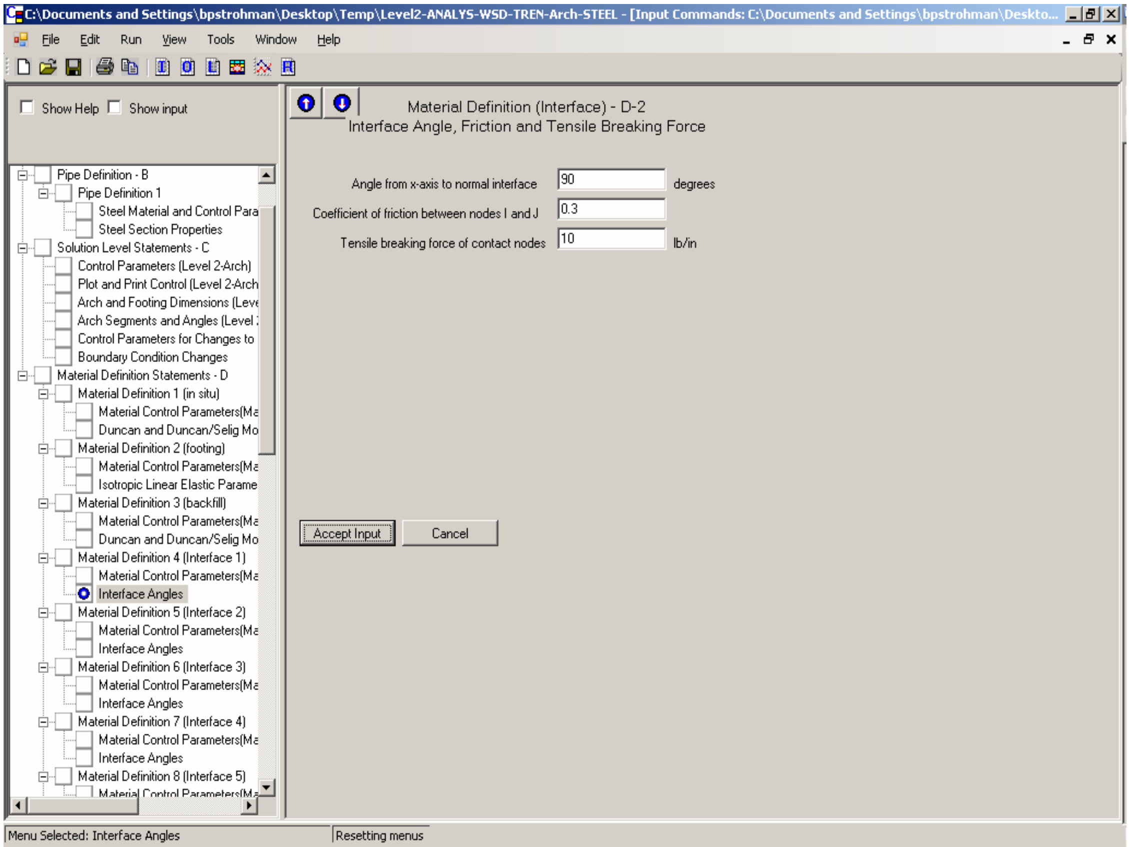


Figure 9-16 – Input Screen D-2 for Material 4 (Interface 1)
(Note: Repeat for Materials 5 through 22 with values provided in the 'Problem Definition' or use the short cut method demonstrated in Problem 6)

When all input is complete and saved, click "Run" and "CANDE-2007" on the main toolbar to execute the program. This will open the "Running CANDE" window which will allow you to monitor the run as it progresses. When the program is complete, a check box "CANDE Analysis Complete" window will appear. Click "OK" and check if the "Running CANDE" window states "Normal Exit From CANDE." If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the "Close" button in the "Running CANDE" window to return to the user interface.

9.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions and construction increments may all be added or removed from the plot. The "Plotting Parameters" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting parameters menu and click the check box "Show constr. increment numbers." Click "OK," set Load Increment to 9 to show the entire mesh and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 9-17.

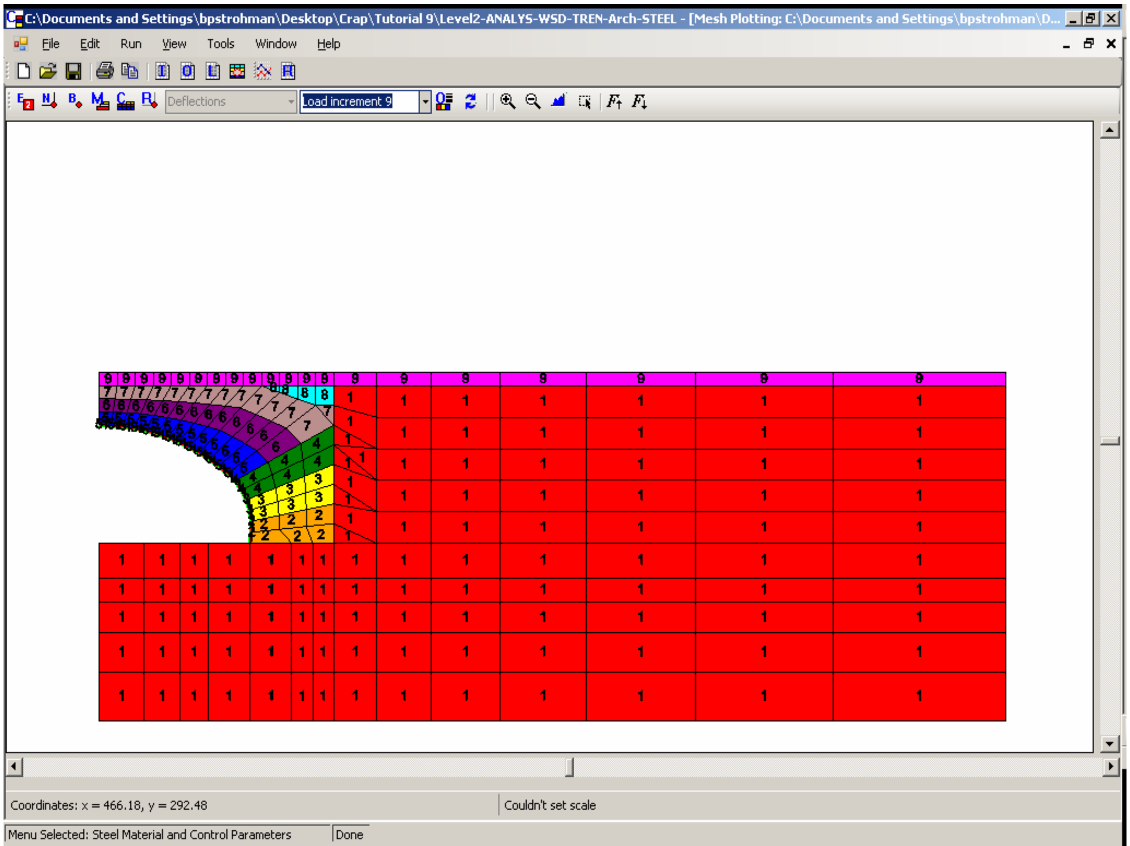


Figure 9-17 – Mesh Plot for Load Steps 1 to 9

To view shear strain, open the plotting parameters window and set the deflection magnification factor to 10 (see Figure 9-18).

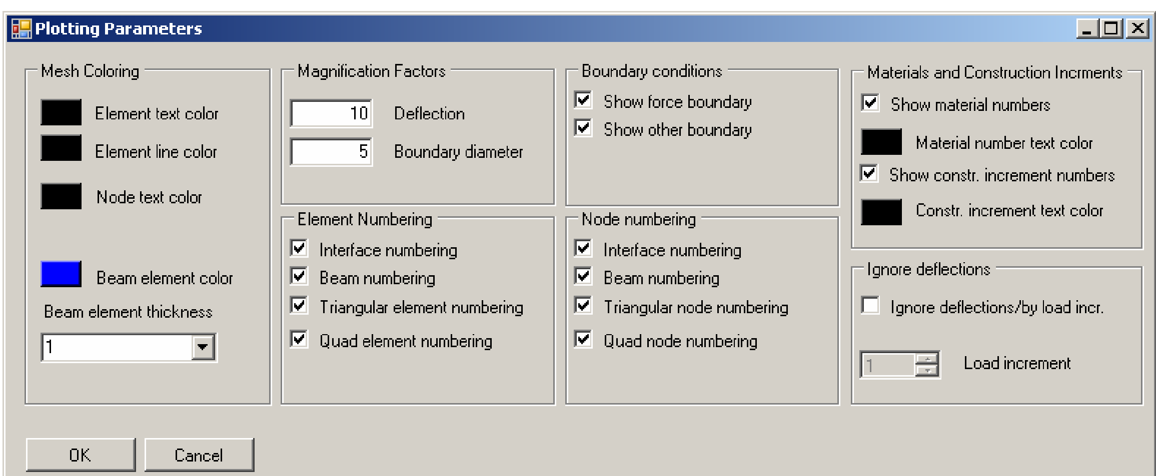


Figure 9-18 – Mesh Window Plotting Parameters

Select "OK" to close the window. Click the icon to "Turn on/off selected output results" and set the drop down box to Shear Strain. Set the increment to 13 and the screen should look like Figure 9-19, which also shows the deflected mesh geometry. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

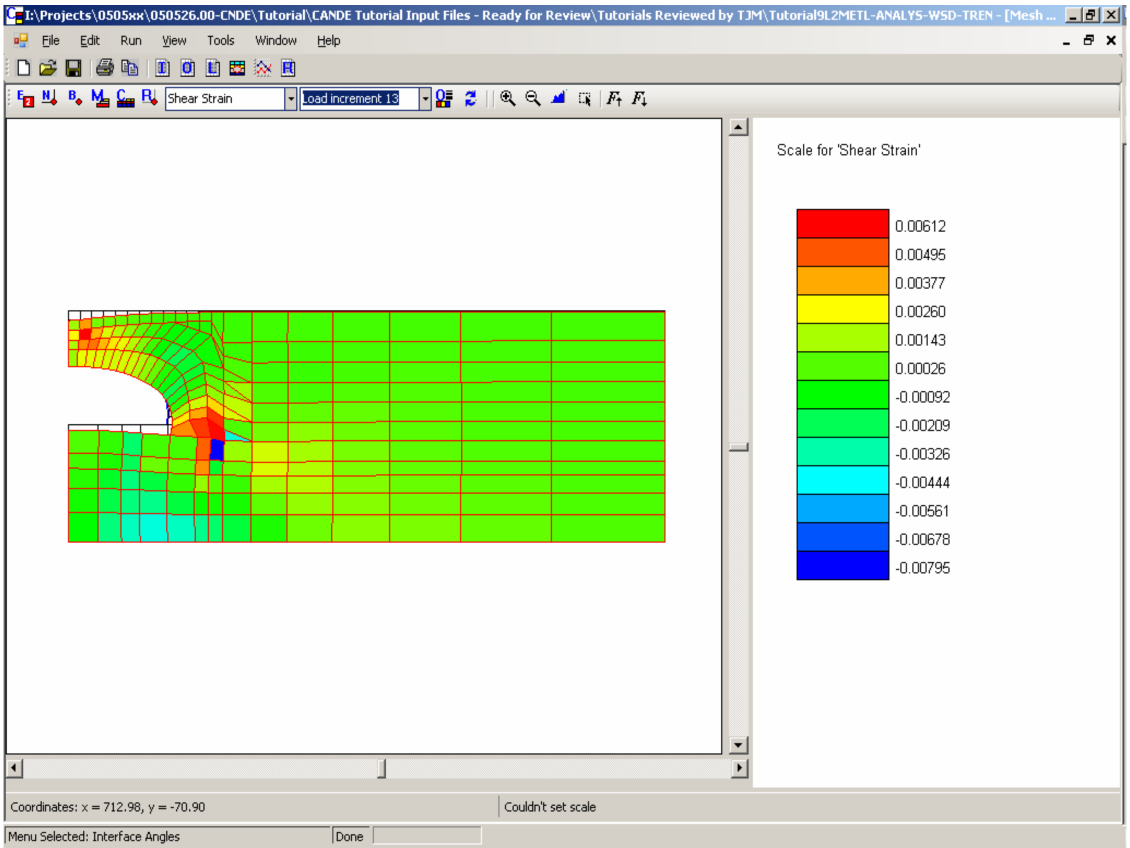


Figure 9-19 – Shear Strain Load Step 13

Similarly, using the "View CANDE Graphs" button on the toolbar or selecting "View/Graphs" the user can explore the forces in the culvert itself. Figure 9-20 compares the thrusts after three increments. This is accomplished using the "Plot multiple load steps" icon at the far left of the toolbar to uncheck "Show single load step" and check Load Steps 7, 11, and 13, then "OK." The "Turn on/off view of pipe" icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to "Thrust Force" and the screen should appear as Figure 9-20.

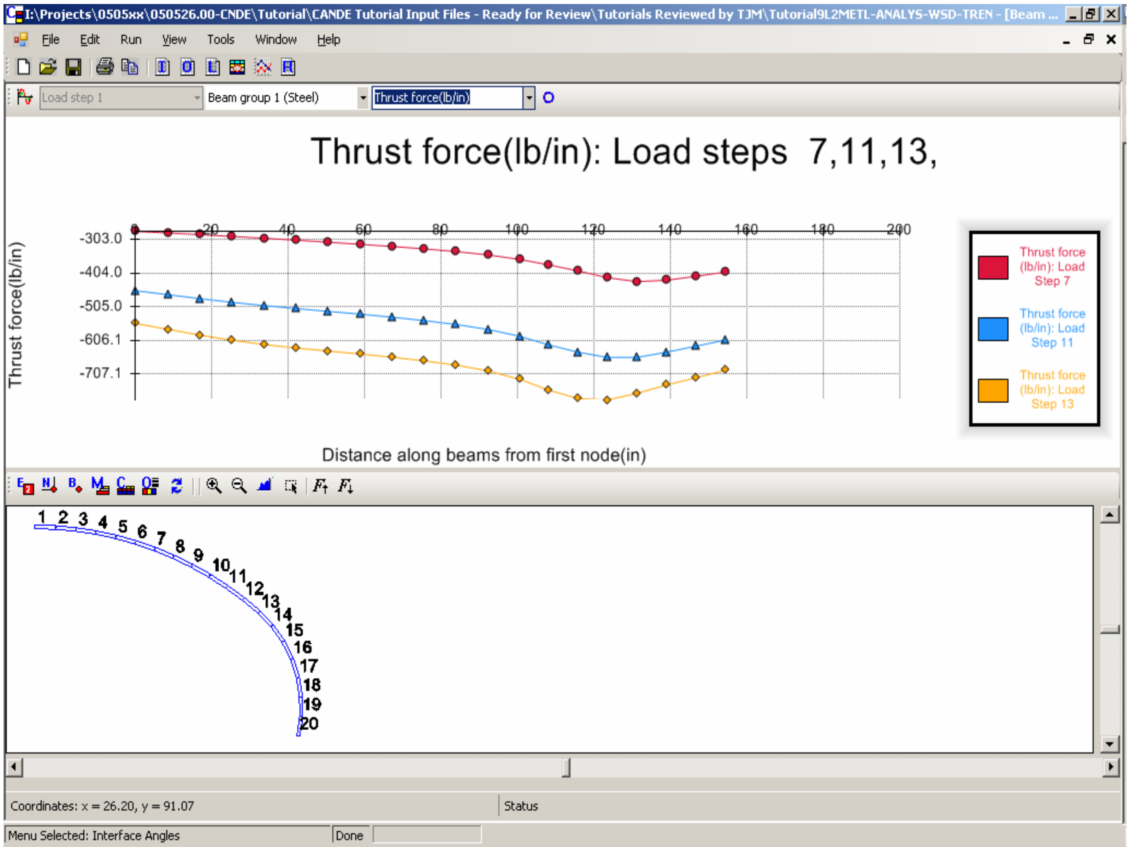


Figure 9-20 – Thrust Force for Load Steps 7, 11, and 13

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 9-21) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 9-21 shows the final assessment printed at the end of the file.

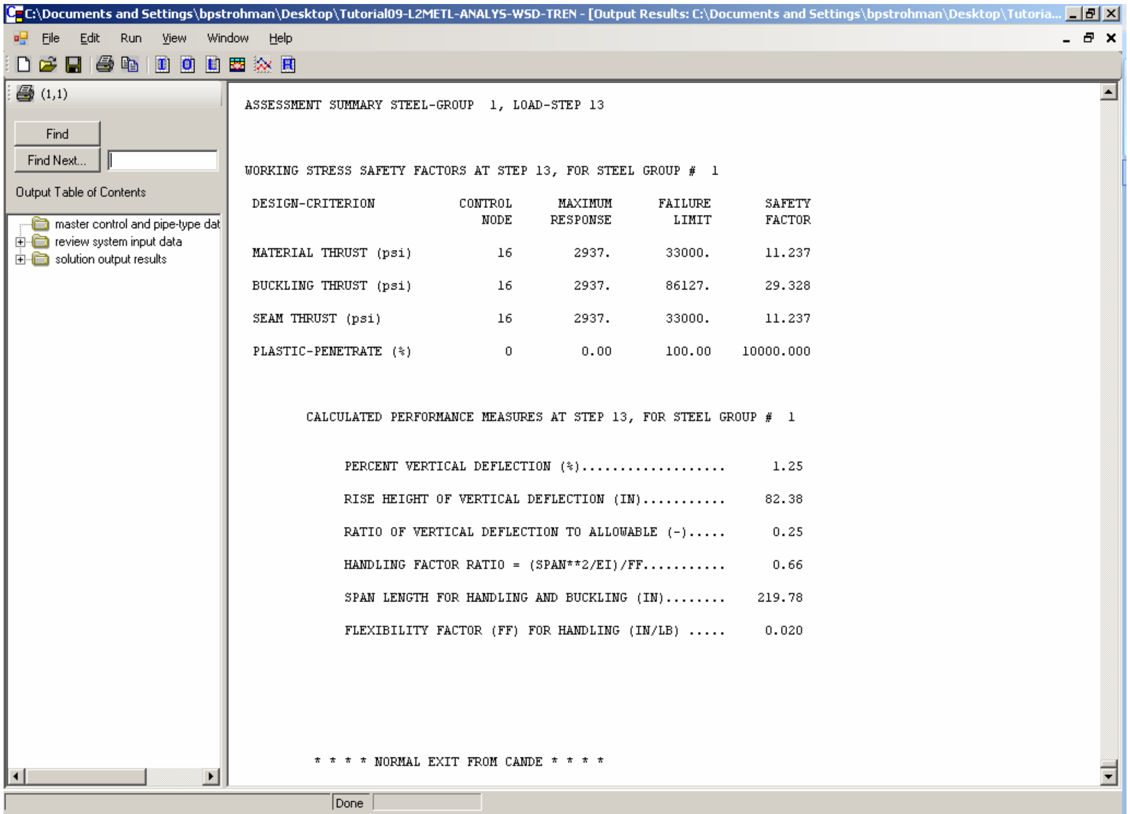


Figure 9-21 – Design Assessment Summary – Load Step 13

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 9-22 shows the Results

Generator input screen set to obtain deflection and shear data from the full output file. After selecting the desired output criteria, click the “Generate/Preview” button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 9-23. The three tabs shown in Figures 9-22 and 9-23 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

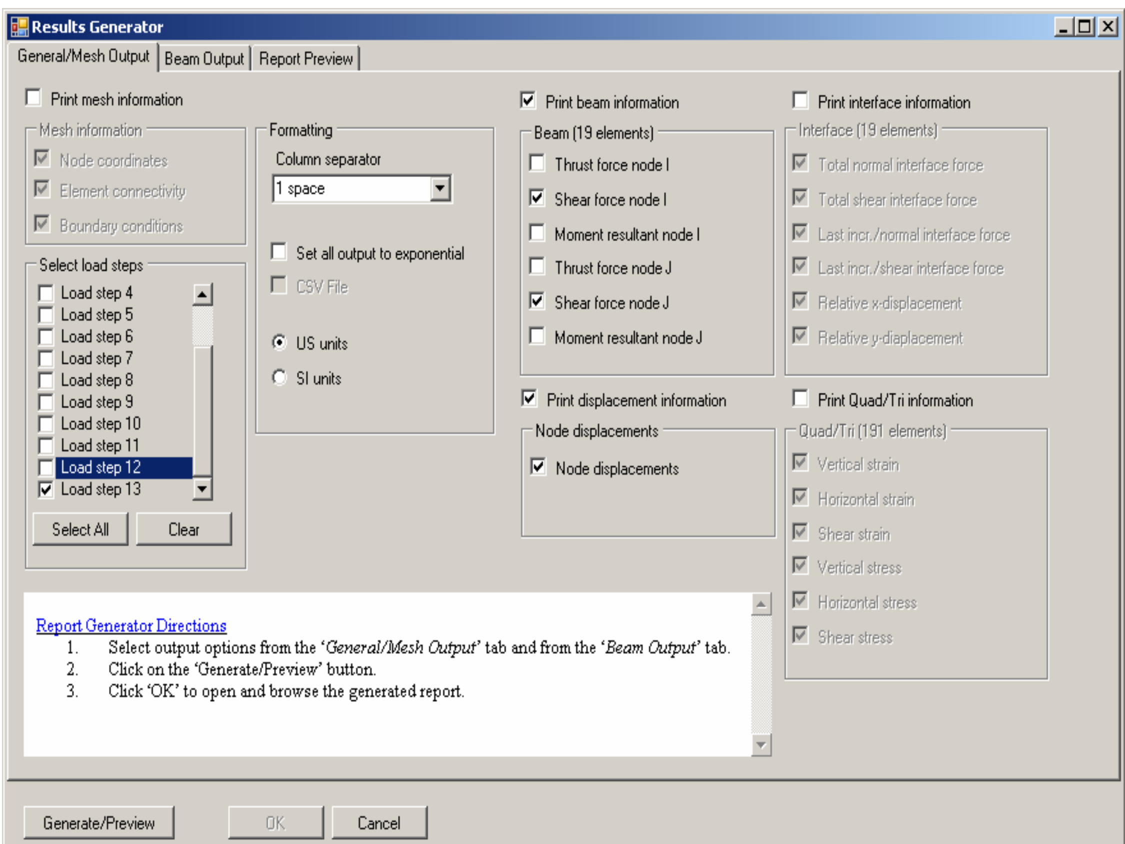
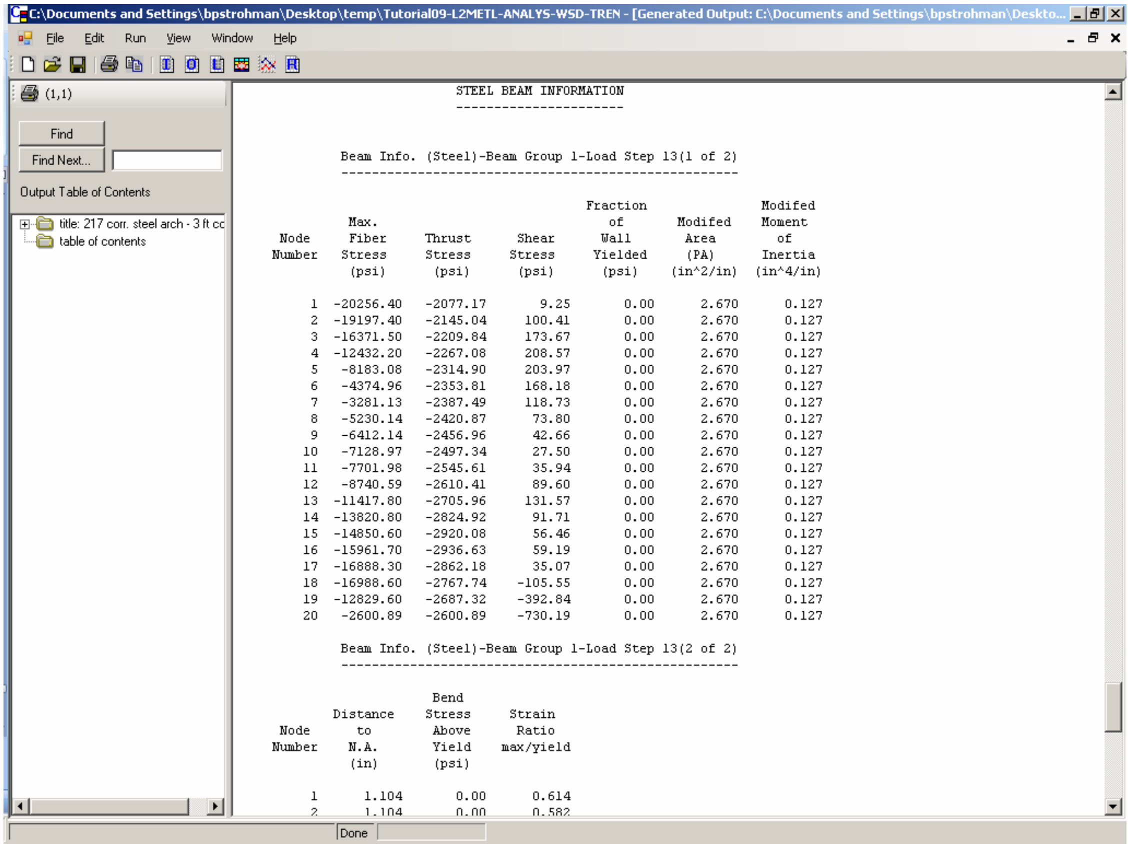


Figure 9-22 – Results Generator Input Screen – Load Step 13 Shear and Deflections



STEEL BEAM INFORMATION

Beam Info. (Steel)-Beam Group 1-Load Step 13(1 of 2)

Node Number	Max. Fiber Stress (psi)	Thrust Stress (psi)	Shear Stress (psi)	Fraction of Wall Yielded (psi)	Modifed Area (PA) (in^2/in)	Modifed Moment of Inertia (in^4/in)
1	-20256.40	-2077.17	9.25	0.00	2.670	0.127
2	-19197.40	-2145.04	100.41	0.00	2.670	0.127
3	-16371.50	-2209.84	173.67	0.00	2.670	0.127
4	-12432.20	-2267.08	208.57	0.00	2.670	0.127
5	-8183.08	-2314.90	203.97	0.00	2.670	0.127
6	-4374.96	-2353.81	168.18	0.00	2.670	0.127
7	-3281.13	-2387.49	118.73	0.00	2.670	0.127
8	-5230.14	-2420.87	73.80	0.00	2.670	0.127
9	-6412.14	-2456.96	42.66	0.00	2.670	0.127
10	-7128.97	-2497.34	27.50	0.00	2.670	0.127
11	-7701.98	-2545.61	35.94	0.00	2.670	0.127
12	-8740.59	-2610.41	89.60	0.00	2.670	0.127
13	-11417.80	-2705.96	131.57	0.00	2.670	0.127
14	-13820.80	-2824.92	91.71	0.00	2.670	0.127
15	-14850.60	-2920.08	56.46	0.00	2.670	0.127
16	-15961.70	-2936.63	59.19	0.00	2.670	0.127
17	-16888.30	-2862.18	35.07	0.00	2.670	0.127
18	-16988.60	-2767.74	-105.55	0.00	2.670	0.127
19	-12829.60	-2687.32	-392.84	0.00	2.670	0.127
20	-2600.89	-2600.89	-730.19	0.00	2.670	0.127

Beam Info. (Steel)-Beam Group 1-Load Step 13(2 of 2)

Node Number	Distance to N.A. (in)	Bend Stress Above Yield (psi)	Strain Ratio max/yield
1	1.104	0.00	0.614
2	1.104	0.00	0.582

Figure 9-23 – Results Generator – Load Step 13 Beam Stresses

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 10

May 2008

This page intentionally left blank

10. CANDE TEST PROBLEM 10

10.1 Problem Definition

Design a 72 in. inside diameter concrete pipe set on gravel bedding with 60 ft of fill over the top of the pipe using LRFD design. The problem is shown schematically in Figure 10-1. The analysis will be with Level 2, using an automated finite element pipe mesh for an embankment installation having a 6 in. layer of soft backpacking soil around the circumference of the pipe and no interface elements. The desired result is the required inner and outer reinforcement.

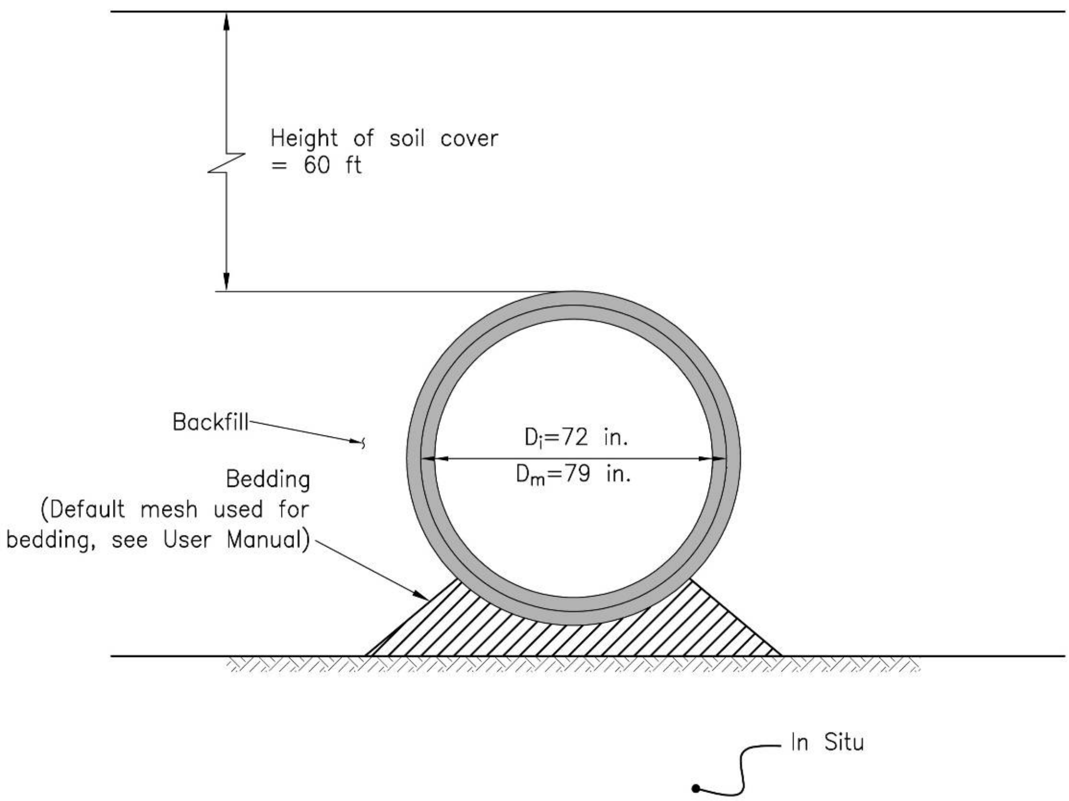


Figure 10-1 Details of Problem 10

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Design

Method of analysis/design - LRFD

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Pipe mesh

Soil mesh pattern - Embankment

Interface elements (pipe only) - None

Pipe material type - Concrete

Reinforcement shape - Standard

Soil Parameters - Canned Duncan/Selig soil models except as noted:

In situ soil - Linear Elastic, Young's modulus = 5,000 psi, Poisson's ratio = 0.40

Bedding soil - SW 90, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Backfill soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Backpack soil - ML 90, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils

Compressive strength of concrete (f'c) - 5,000 lb/in.^{2}

Shear strength equation - Pipes/arches (AASHTO 12.10.4.2.5)

Concrete strain at tension rupture - 0.0 in./in.

Note: Assuming zero tensile strength for the concrete assures the design will be for a cracked section.

Compressive strain at end of elastic range - 0.001 in./in.

Compressive strain at the initial strength limit - 0.002 in./in.

Unit weight of concrete for body weight - 150 lb/ft^3

Crack width model - Heger-McGrath (AASHTO 12.10.4.2.4d)

Analysis mode - Small deformation

Yield stress of reinforcing steel - 65,000 lb/in.^{2}

Inner surface spacing between rows of rebar - 2 in.

Outer surface spacing between rows of rebar - 2 in.

Number of inner cage layers of reinforcement - 2

Number of outer cage layers of reinforcement - 2

Type of reinforcement - Welded or deformed wire

Nonlinear behavior selection - Option 3 Plus steel yielding behavior

Concrete wall thickness - 7 in.

Concrete cover to c.l. of steel rebar cage - 1.25 in. (inside and outside reinforcement)

Average diameter of pipe - 79 in.

Height of soil cover - For embankment installations CANDE calculates height of soil cover from the springline of the pipe (see *User Manual*, Chapter 5, C-2). To achieve a total height of fill of 60 ft over the top of the pipe, set the depth to 60 ft + mean pipe radius = 63 ft (See Figure 10-1).

Density of soil above truncated mesh - 120 lb/ft^3

Number of construction steps - See *User Manual*, Chapter 5, C-3 – The default mesh provides 5 construction steps to the top of the mesh, which provides approximately 1.5 pipe diameters of fill over the arch. For the remaining depth of fill, use 15 steps, making a total of 20.

Number of backpacking elements - 10

Thickness of backpacking - 6 in.

LRFD load factor - 1.35

Load modifier - 1.05 (non-redundant for earth load)

10.2 Creating the CANDE Input Document

Figures 10-2 through 10-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 3, CANDE will display a screen indicating that an input path is initiated - click “Finish.” The user is prompted for a filename and directory.

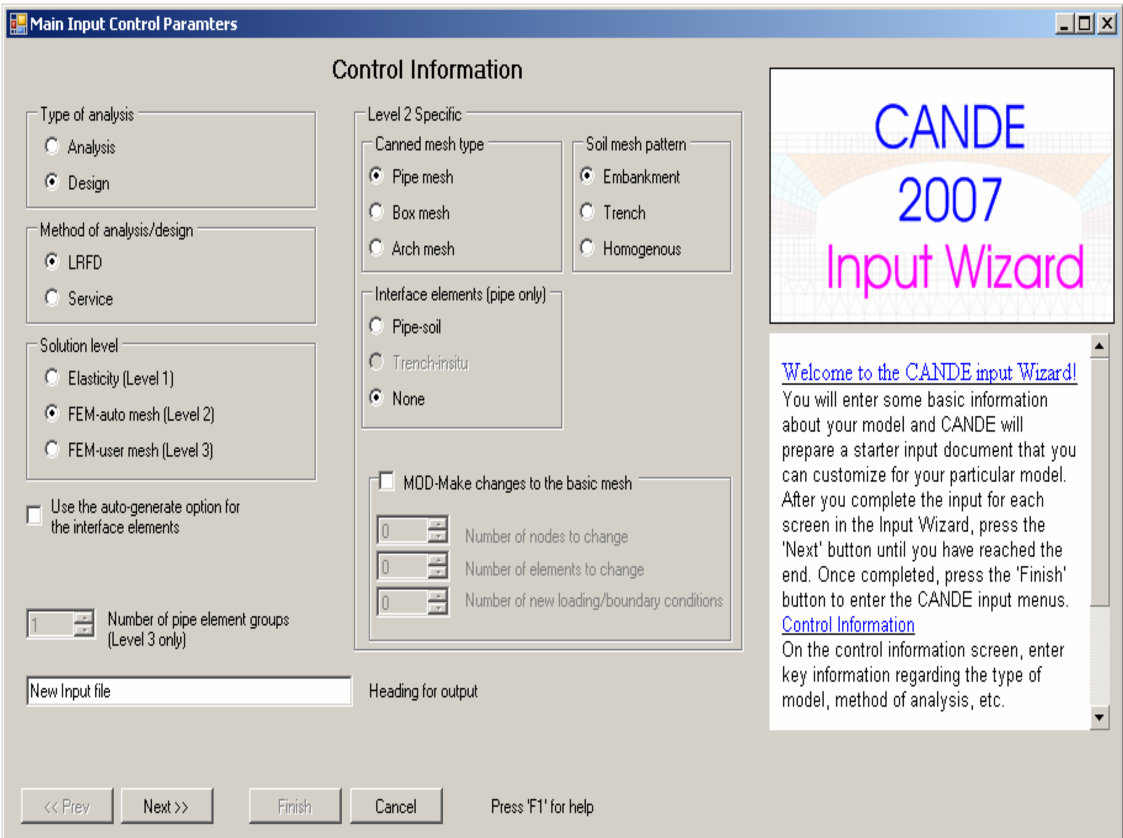


Figure 10-2 – Input Wizard, Screen 1

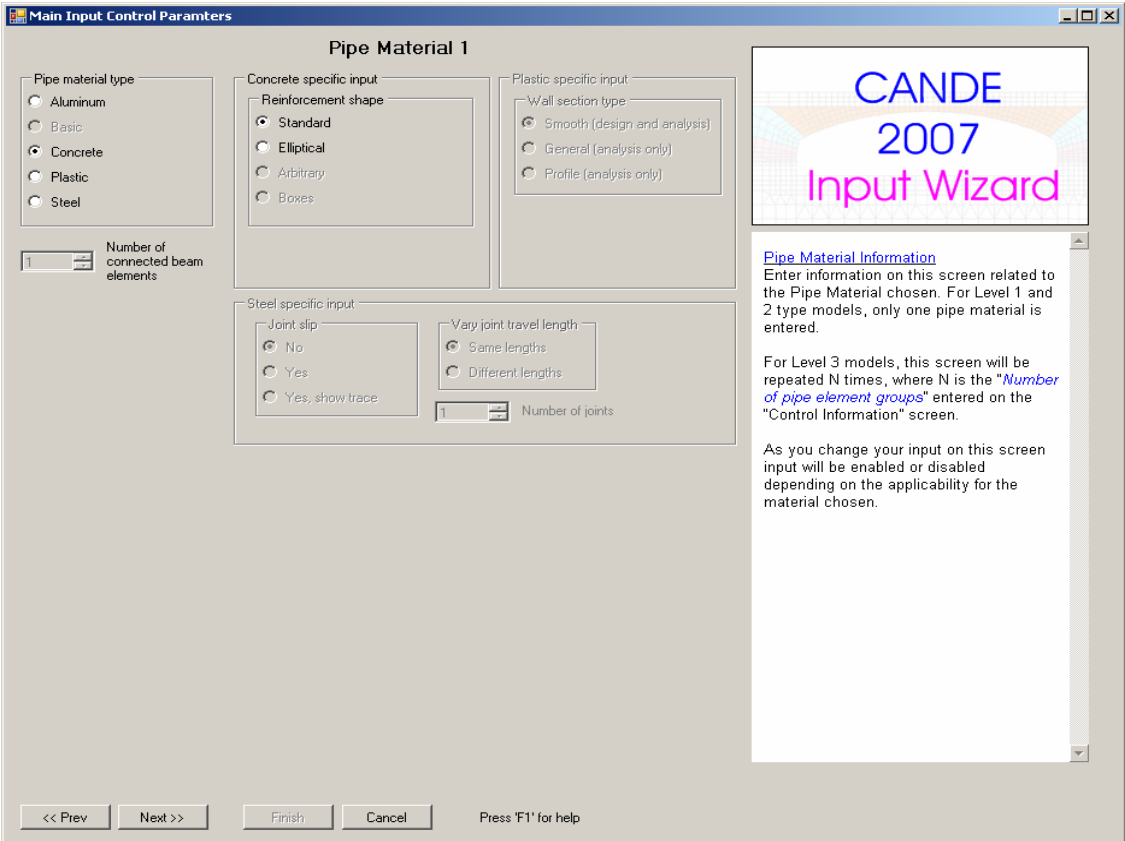


Figure 10-3 – Input Wizard, Screen 2

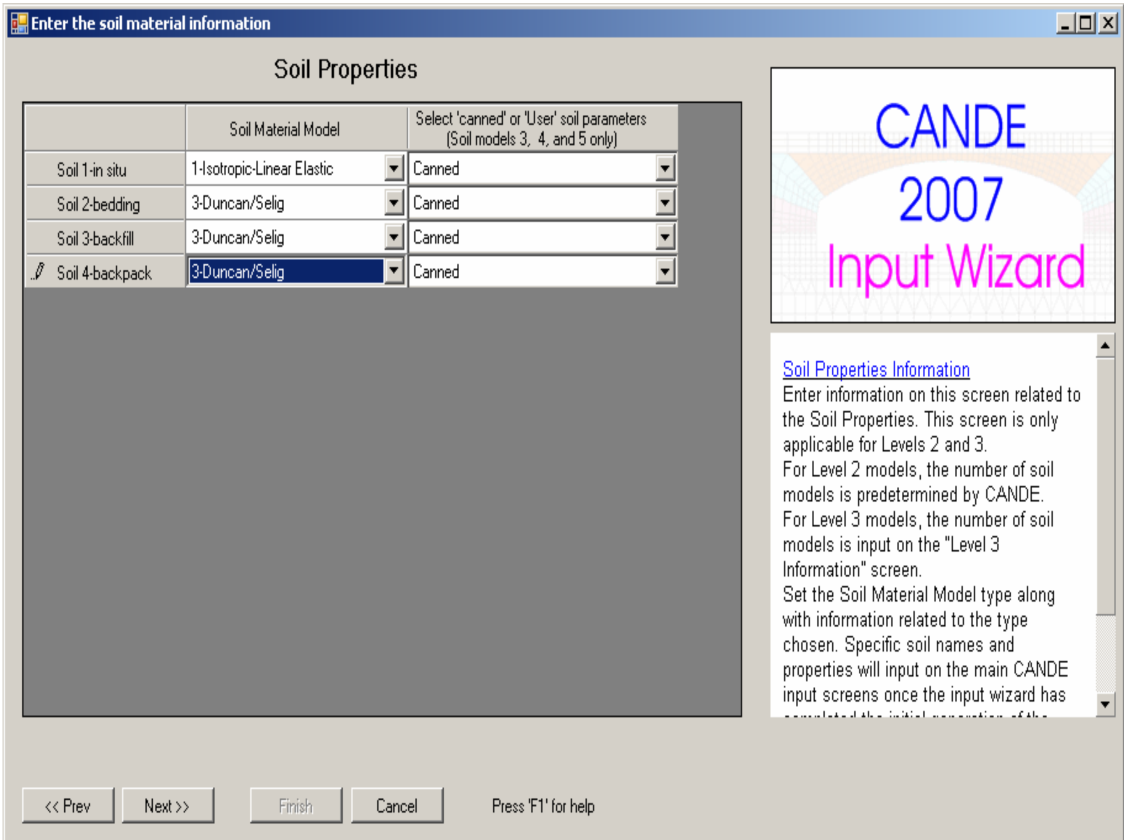


Figure 10-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 10-5. Enter an appropriate heading for output and click "Accept Input."

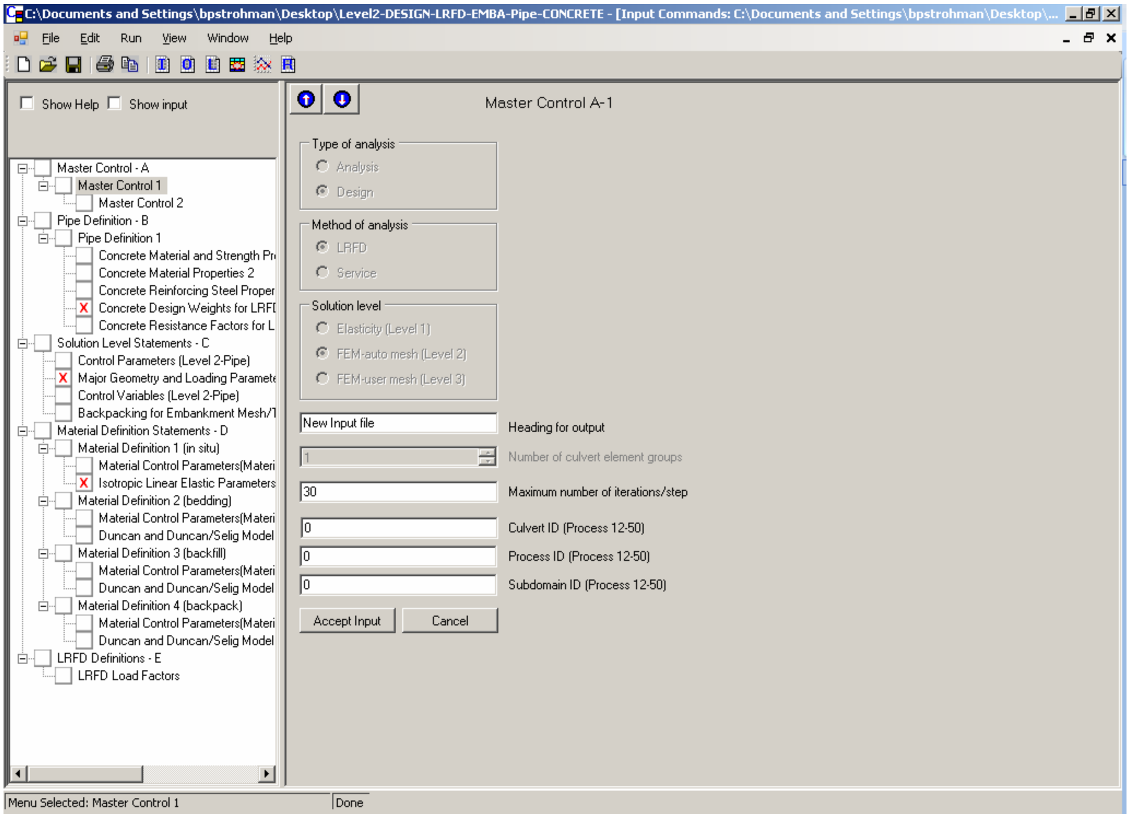


Figure 10-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 10-5 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 10-6 through 10-15 show the completed input for the screens requiring data for the tutorial, except that only one material definition is shown. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

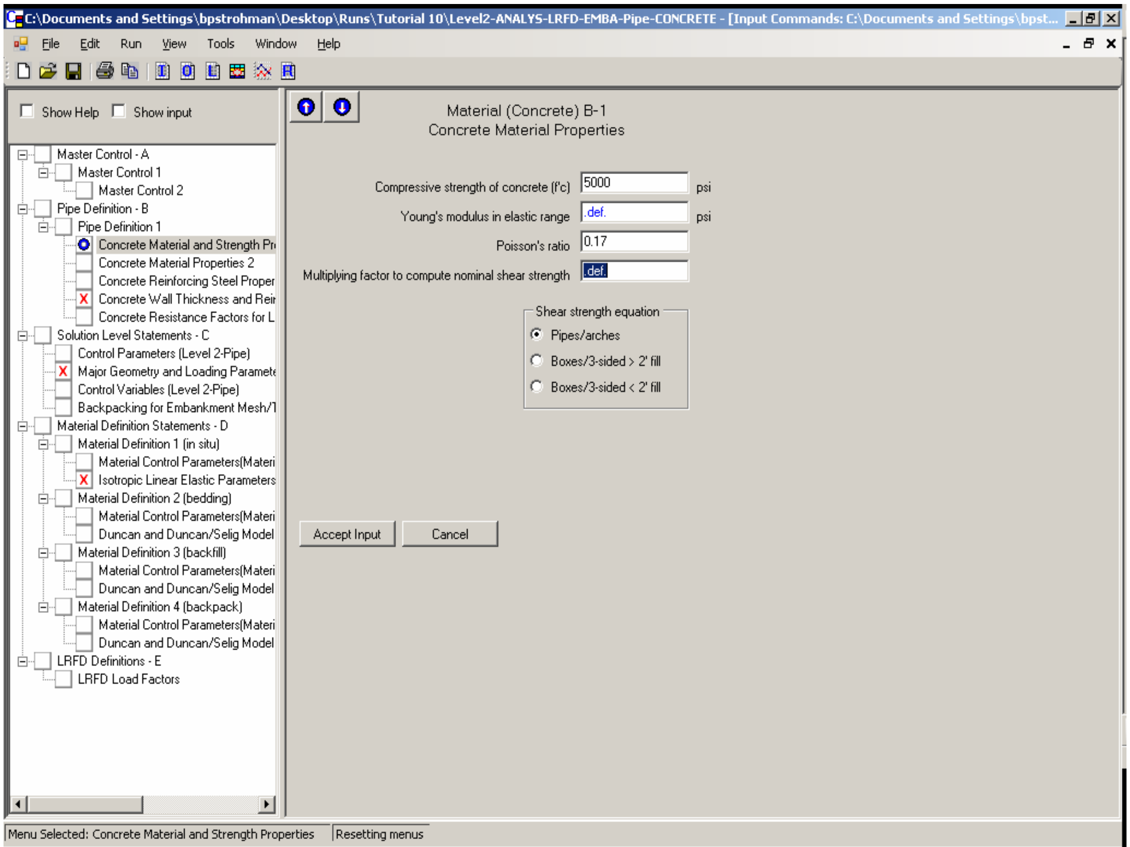


Figure 10-6 – Input Screen B-1

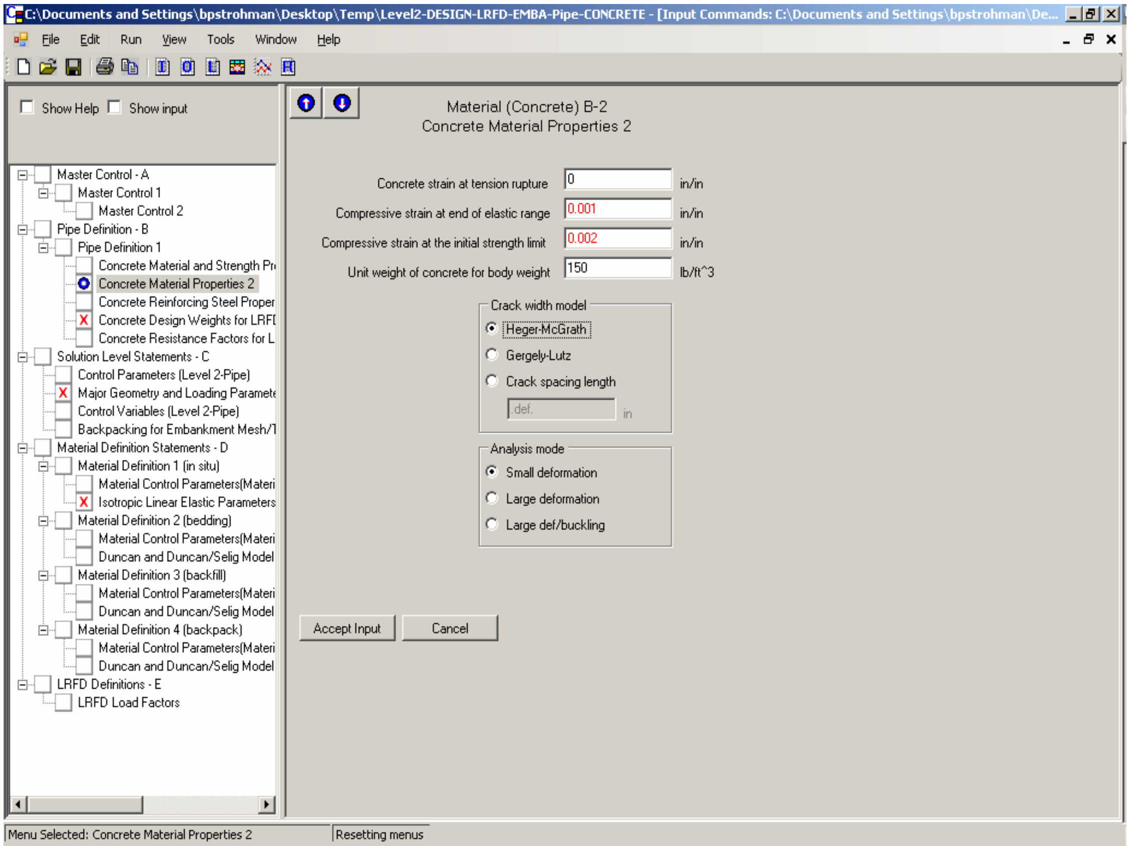


Figure 10-7 – Input Screen B-2

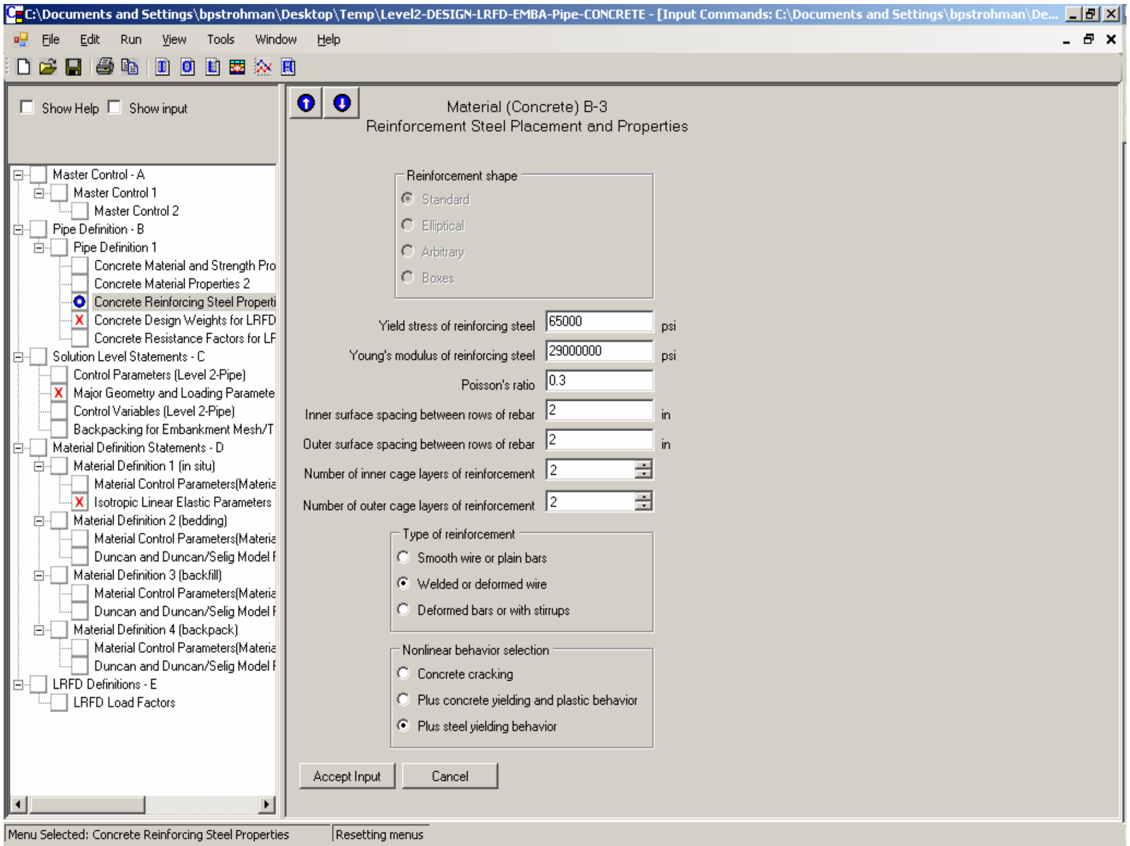


Figure 10-8 – Input Screen B-3

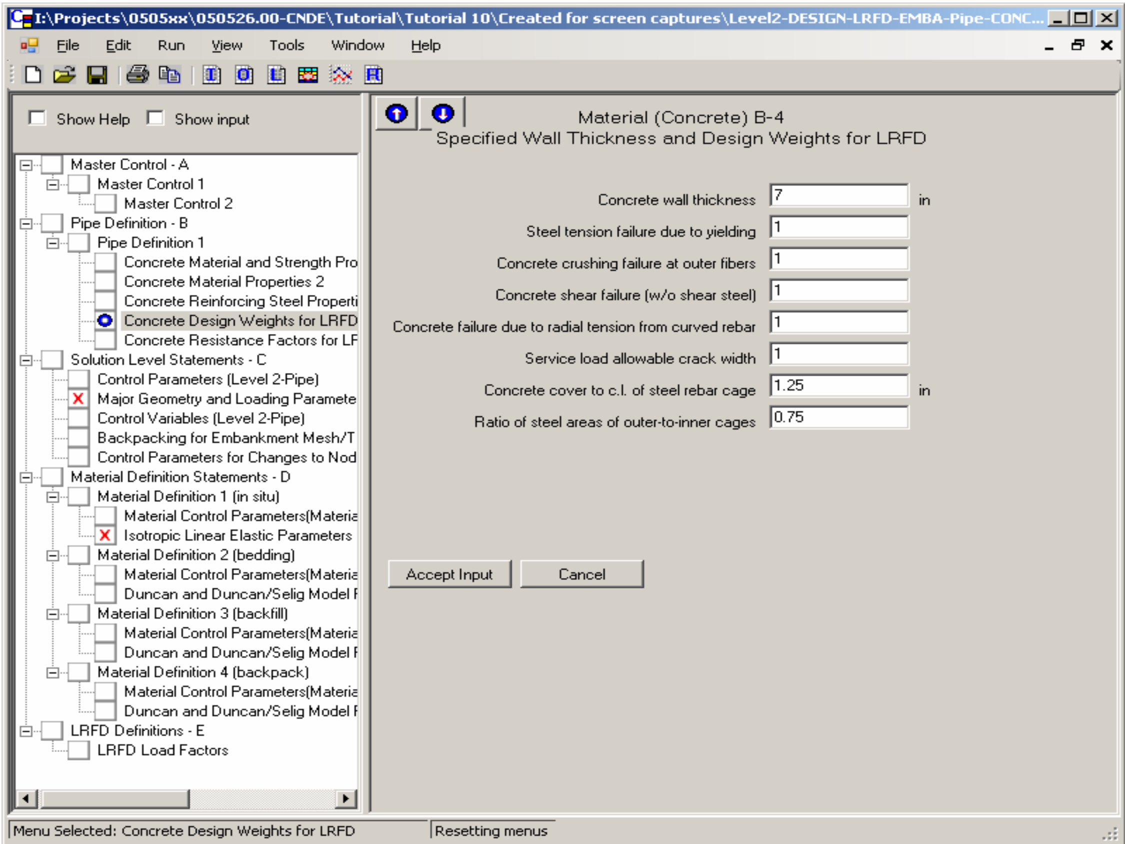


Figure 10-9 – Input Screen B-4

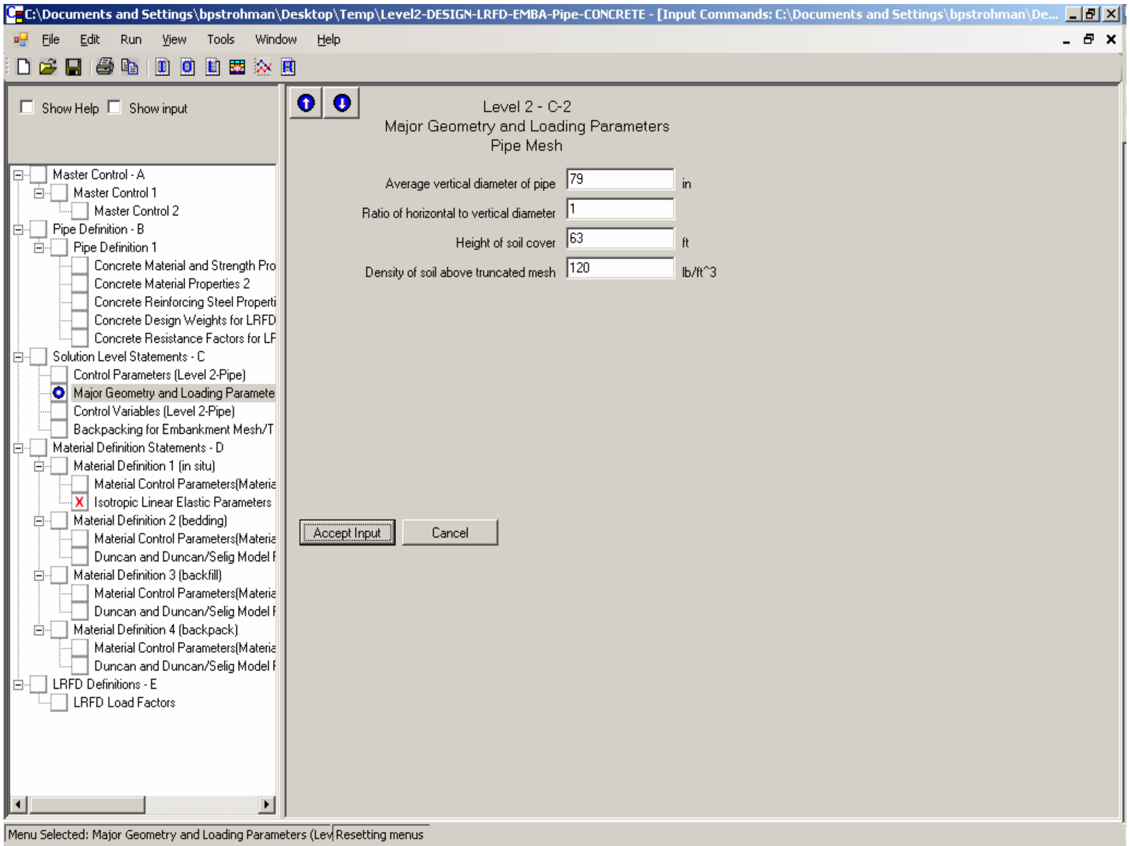


Figure 10-10 – Input Screen C-2

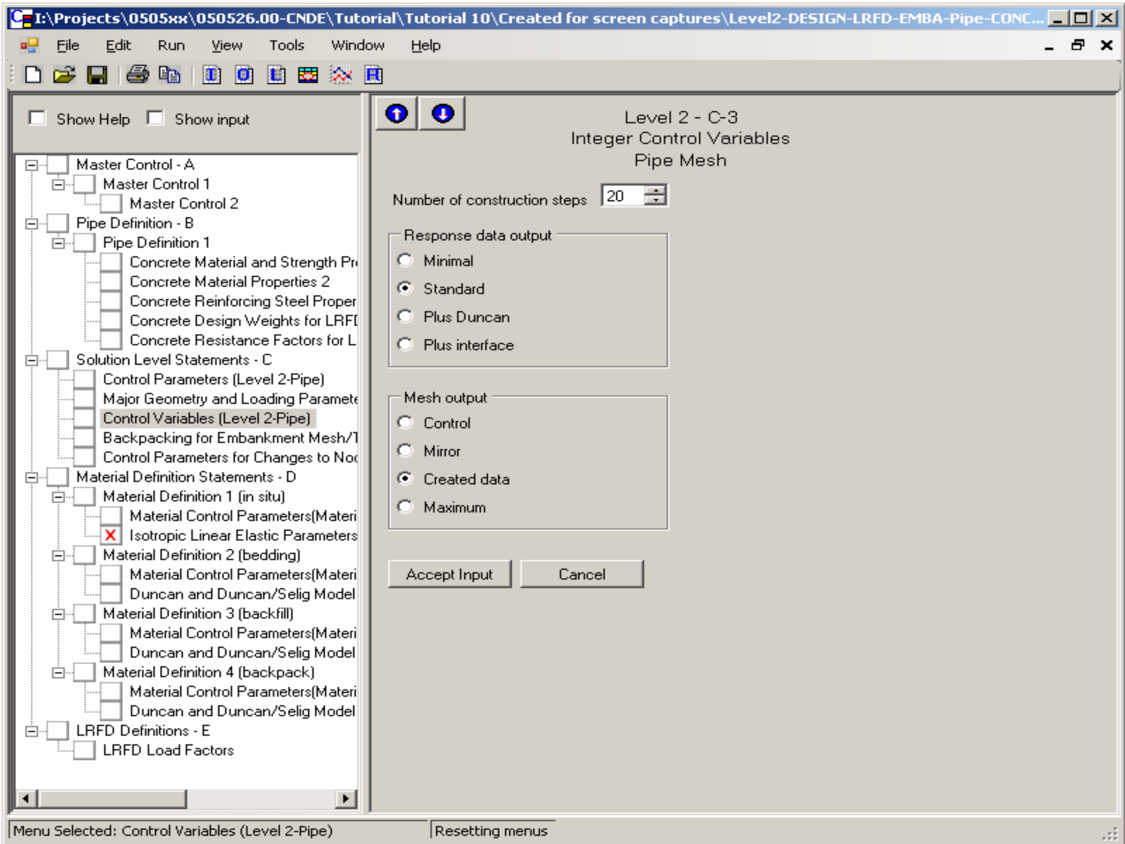


Figure 10-11 – Input Screen C-3

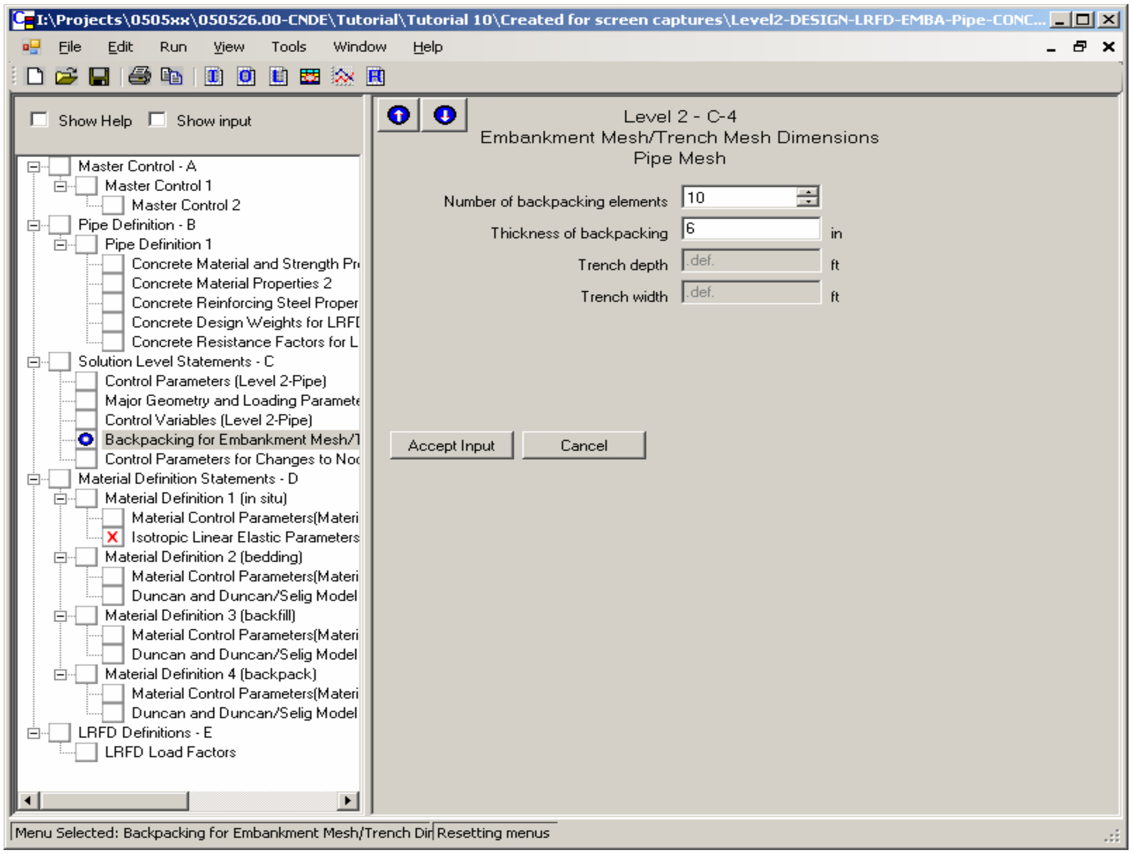


Figure 10-12 – Input Screen C-4

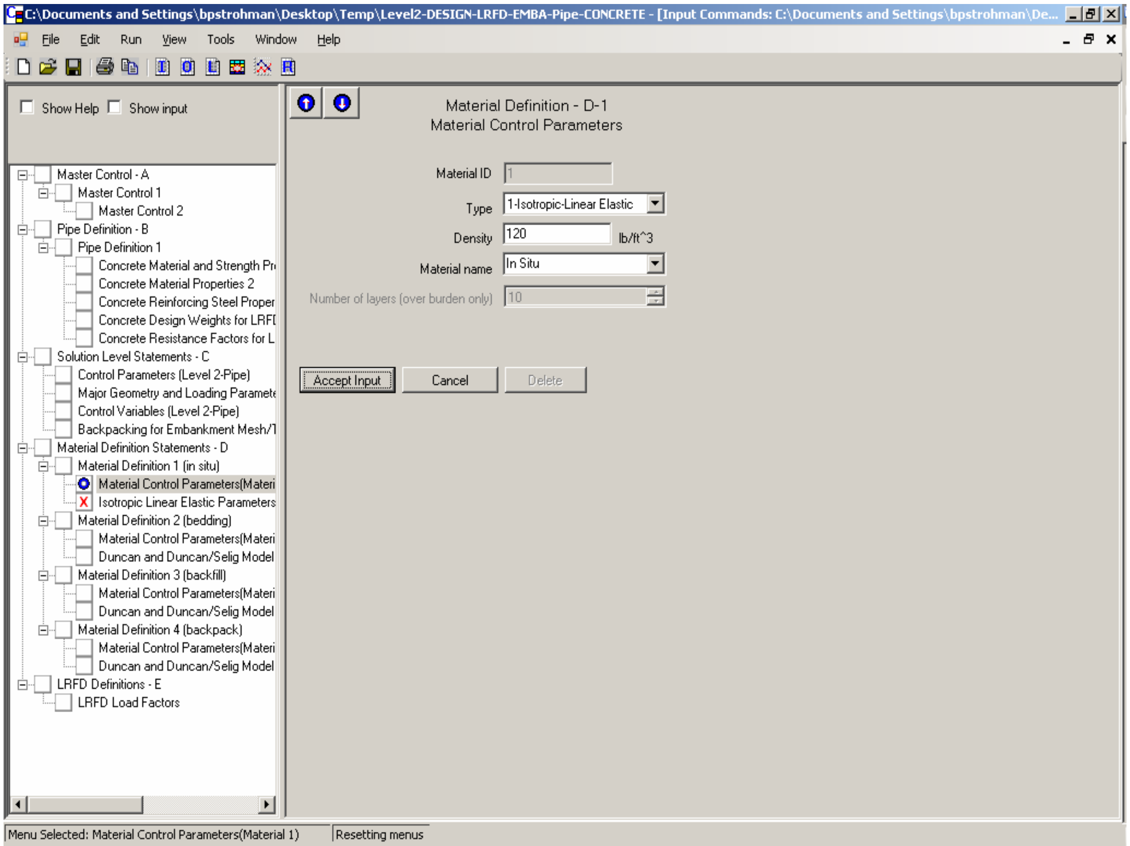


Figure 10-13 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

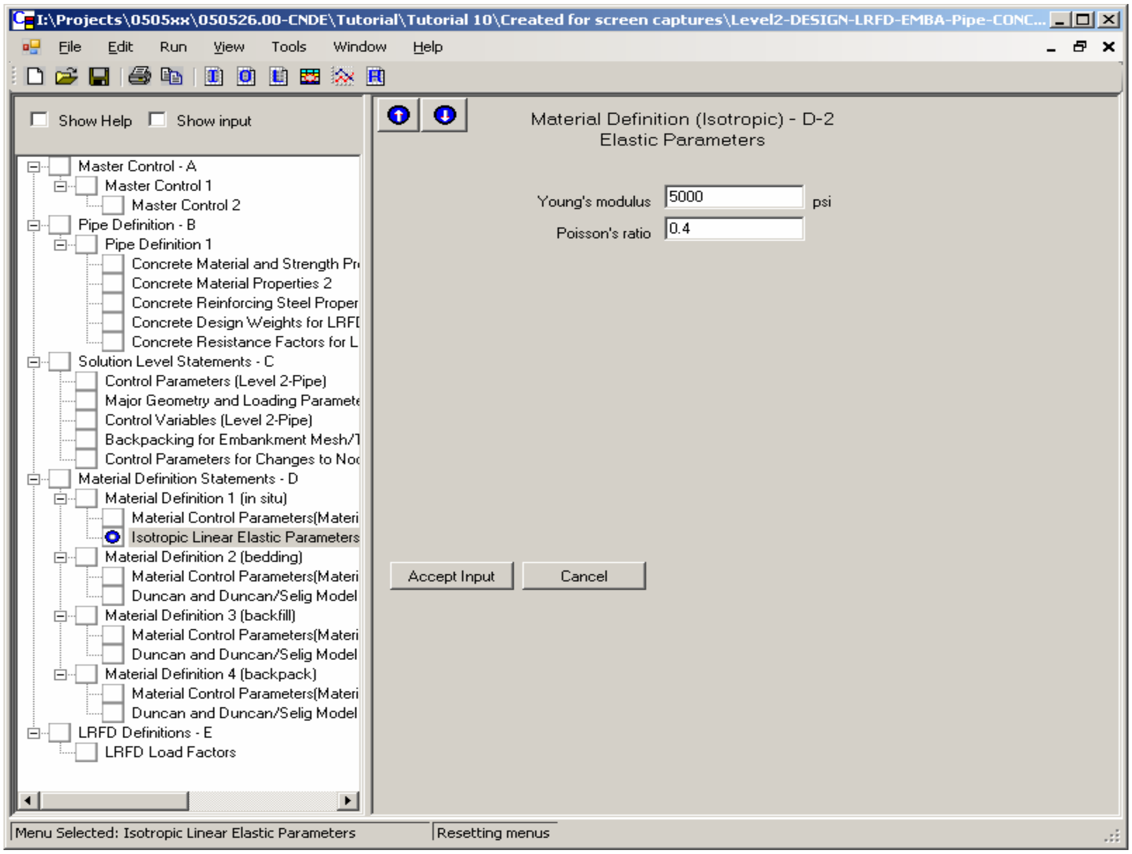


Figure 10-14 – Input Screen D-2 for Material 1 – In Situ Soil

(Note: Repeat for Materials 2, 3, and 4 with values provided in the ‘Problem Definition’)

Under LRFD load factor in Figure 10-15 enter the combined value of the load factor and load modifier, i.e. – 1.35 * 1.05 = 1.42

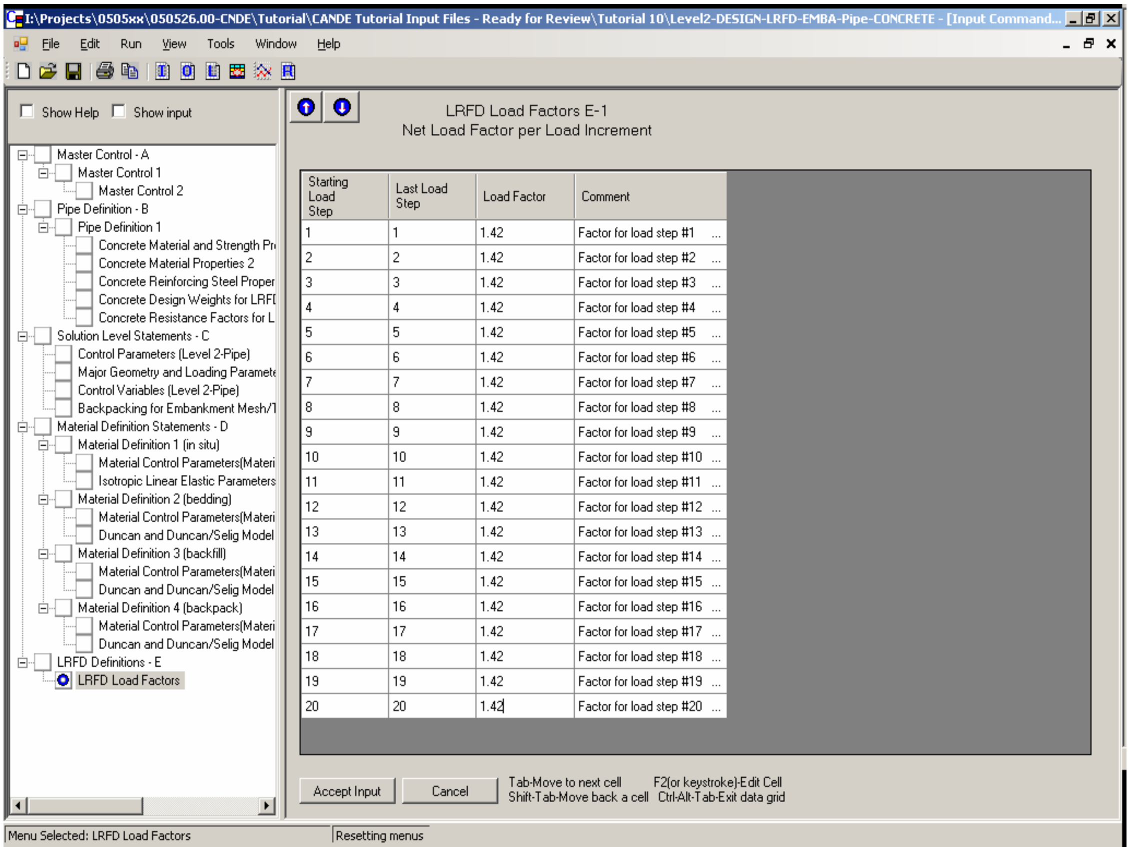


Figure 10-15 – Input Screen E-1

When all input is complete, click “Run” and “CANDE-2007” on the main tool bar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “Analysis Complete” window will appear. Click okay and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the text input option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

10.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The "Plotting Parameters" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting parameters menu and click the check box "Show constr. increment numbers." Click "OK," set Load Increment to 5 to show the entire mesh (the remaining load is placed above the mesh – see *User Manual*) and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 10-16.

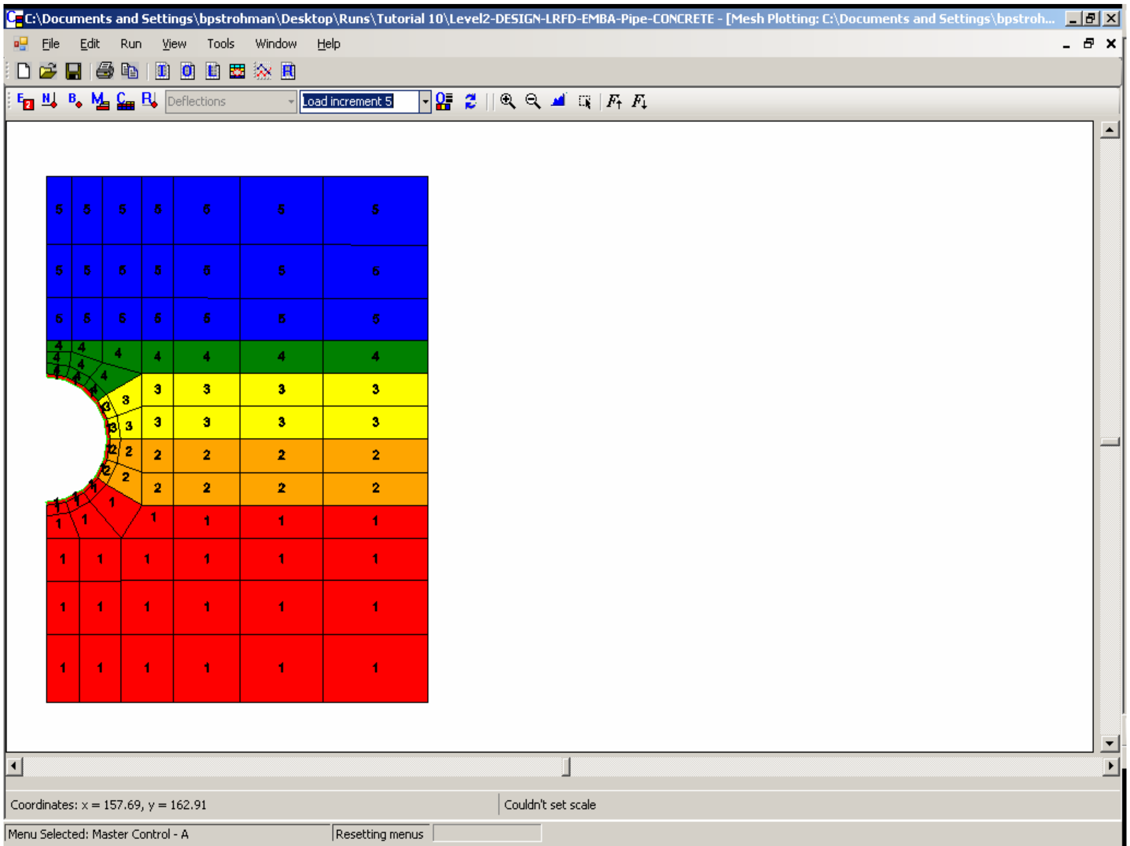


Figure 10-16 – Mesh Plot for Load Steps 1 to 5

To view vertical stress, open the plotting parameters window and set the deflection magnification factor to 10 (see Figure 10-17).

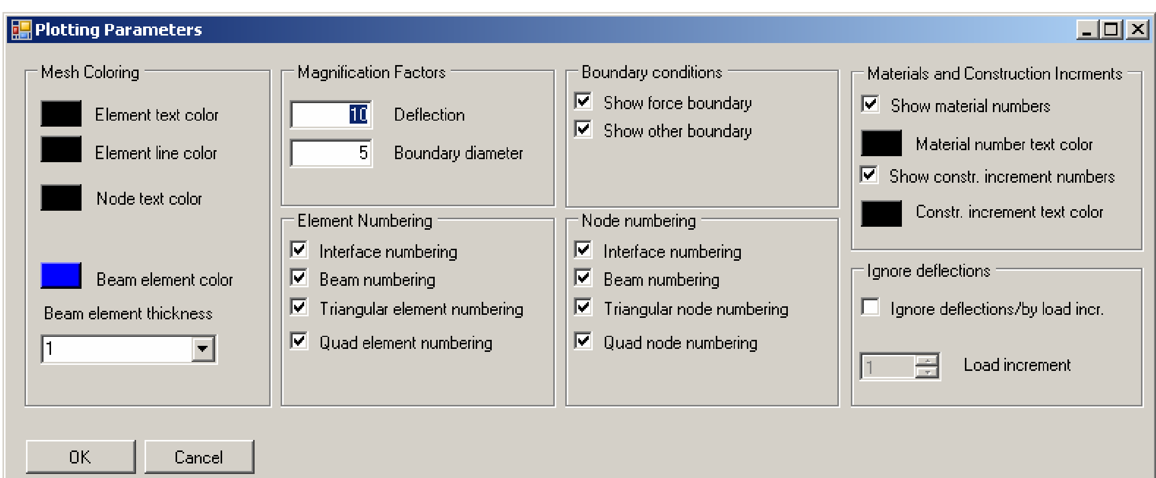


Figure 10-17 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Vertical Stress. Set the increment to 20 and the screen should look like Figure 10-18, which also shows the deflected mesh geometry. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

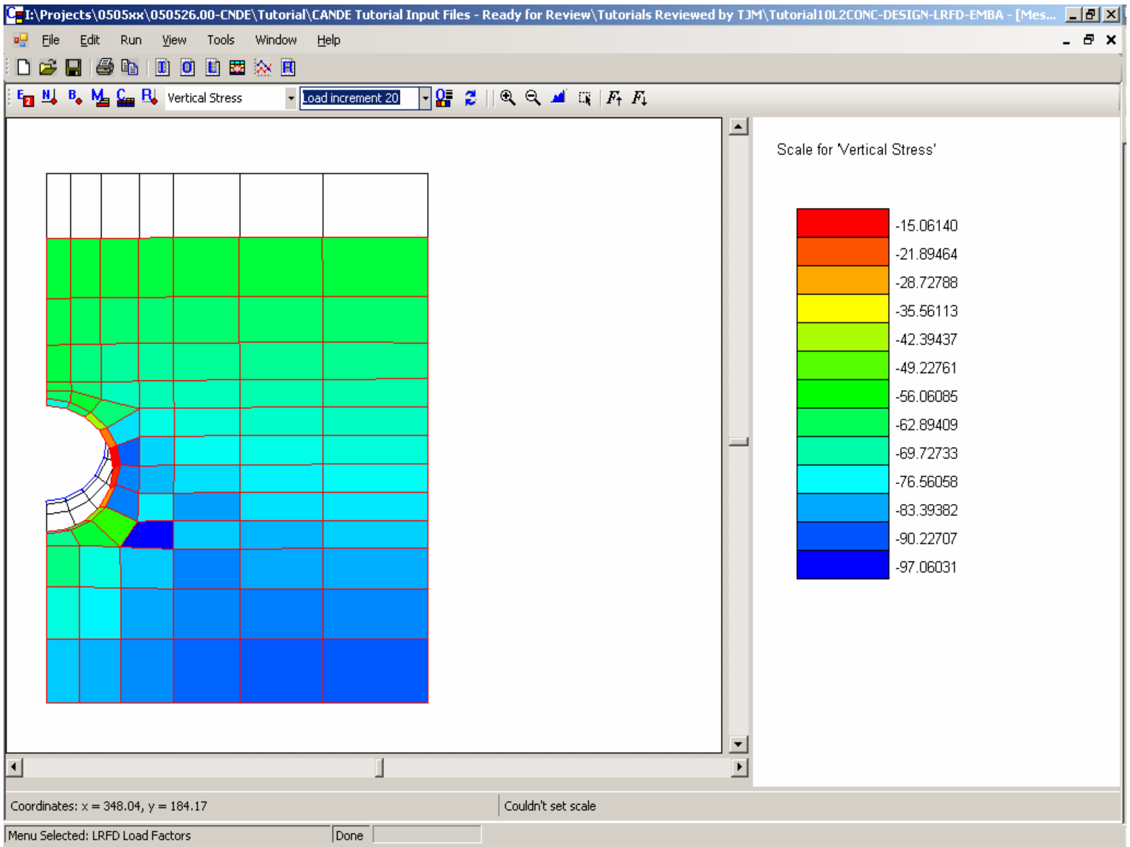


Figure 10-18 – Vertical Stress Load Step 20

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 10-19 compares the thrusts after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 5, 13, and 20, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Thrust Force” and the screen should appear as Figure 10-19.

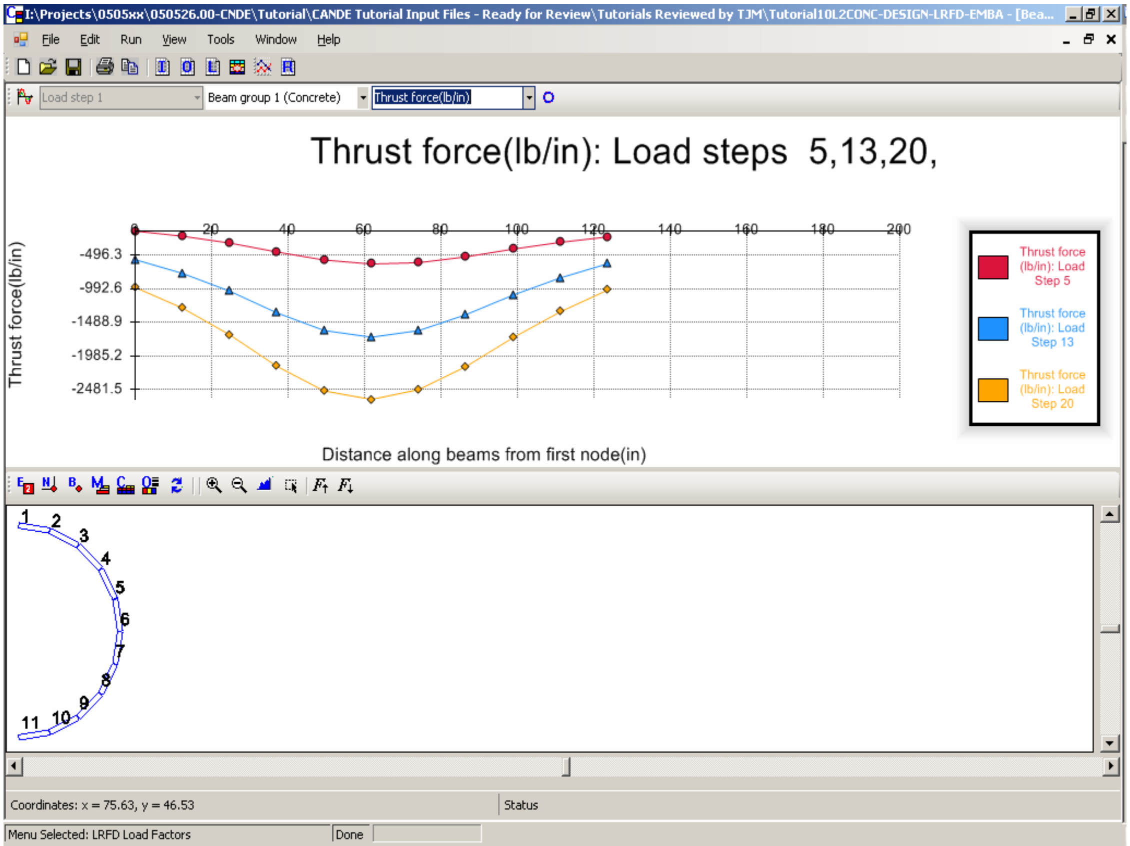


Figure 10-19 – Thrust Forces for Load Steps 5, 13, and 20

The user can also change the chart properties by right clicking on the plot and selecting "Chart Property Pages." Within the resulting window, which is shown in Figure 10-20, a user can vary the properties of the plot including changing the scale, the bounds of the plot, the legend, etc. Figure 10-21 shows the 'Chart Property Pages' window if the user would like to modify the horizontal scale to 130 as the maximum value. The updated thrust diagram is shown in Figure 10-22.

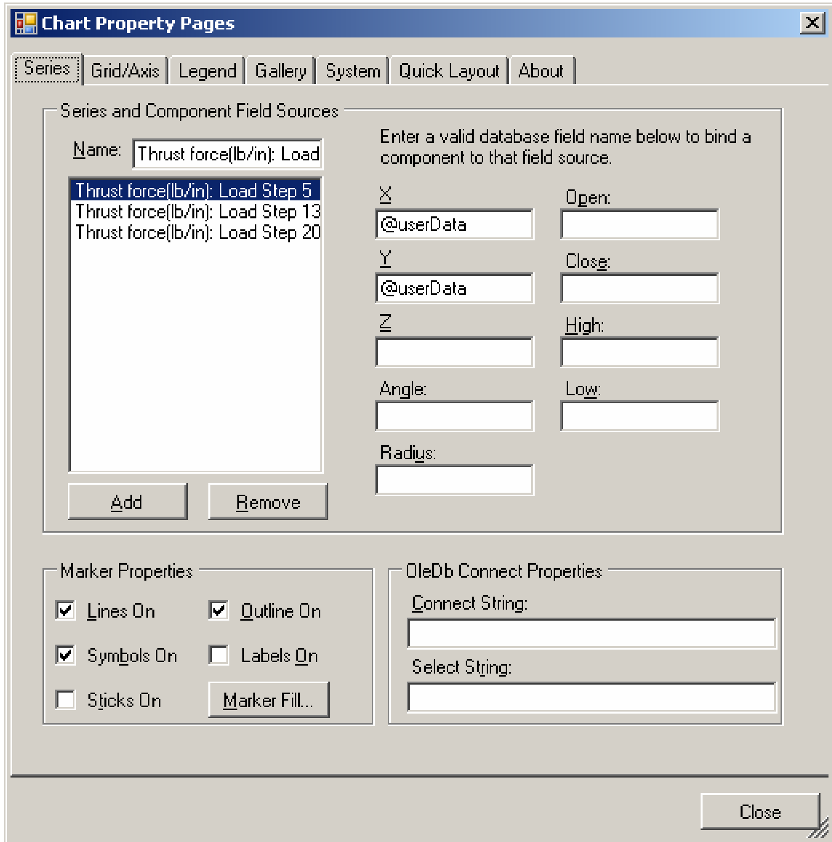


Figure 10-20 – Chart Property Pages Window

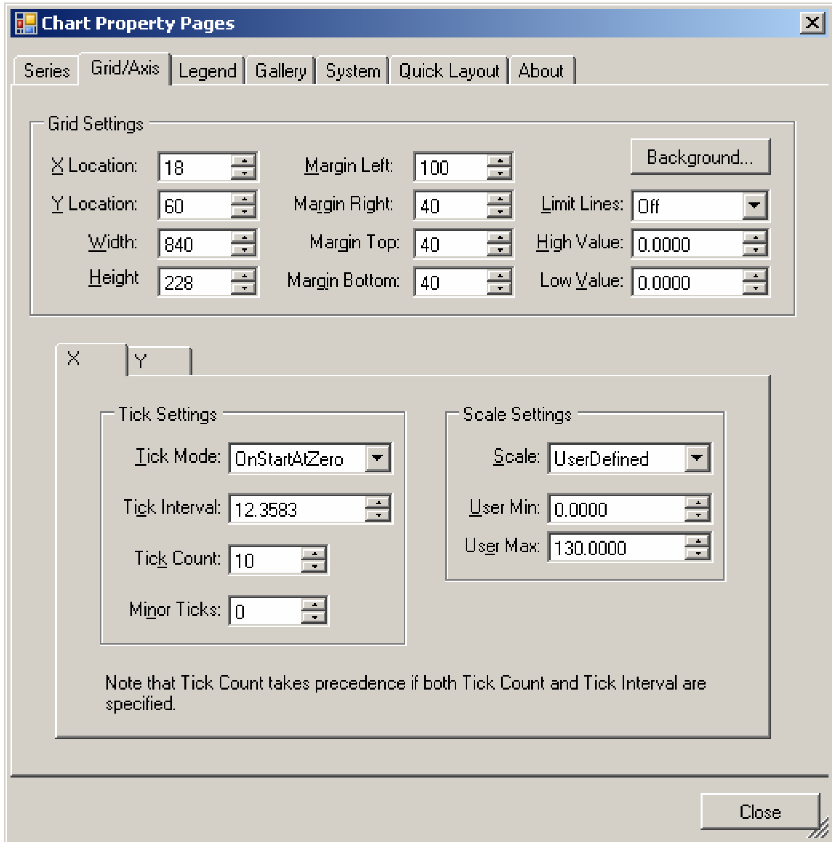


Figure 10-21 – Modified Chart Property Pages Window

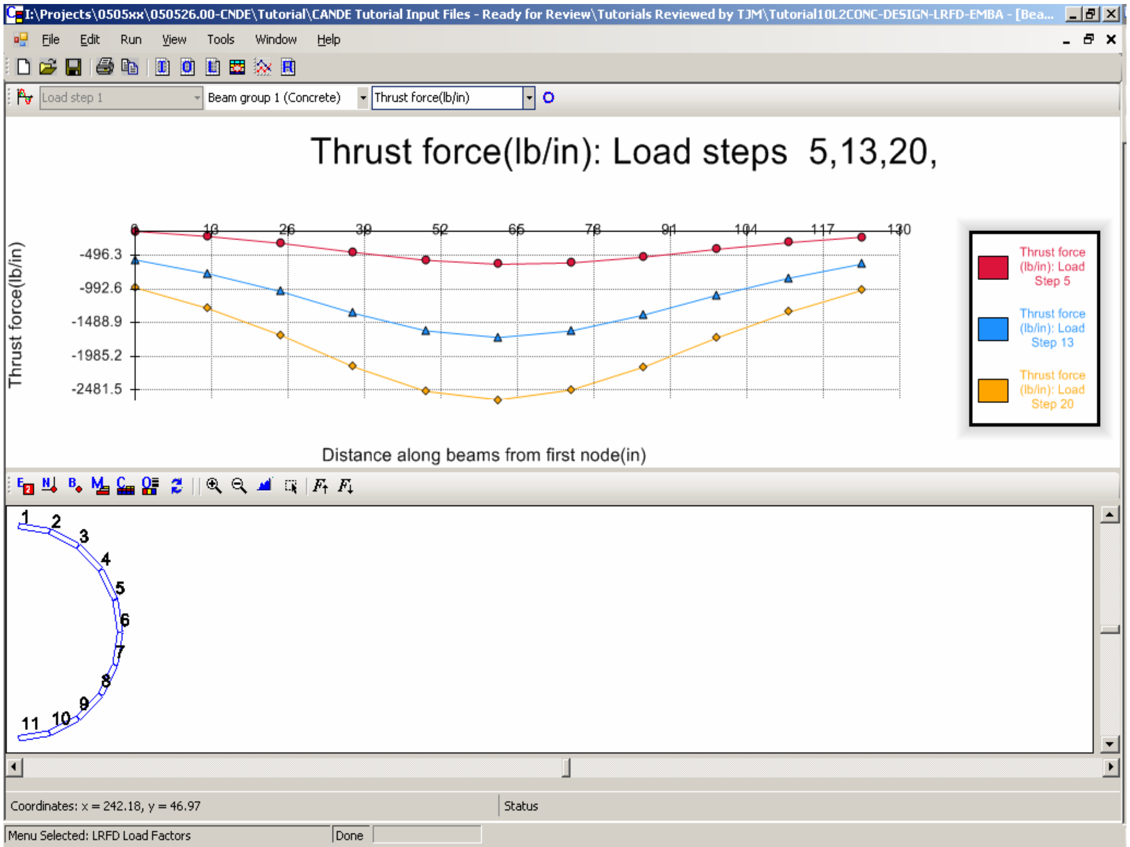


Figure 10-22 – Updated Thrust Forces for Load Steps 5, 13, and 20

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data, the design solution (Figure 10-23), or the design assessment (Figure 10-24) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 10-24 shows the final assessment printed at the end of the file.

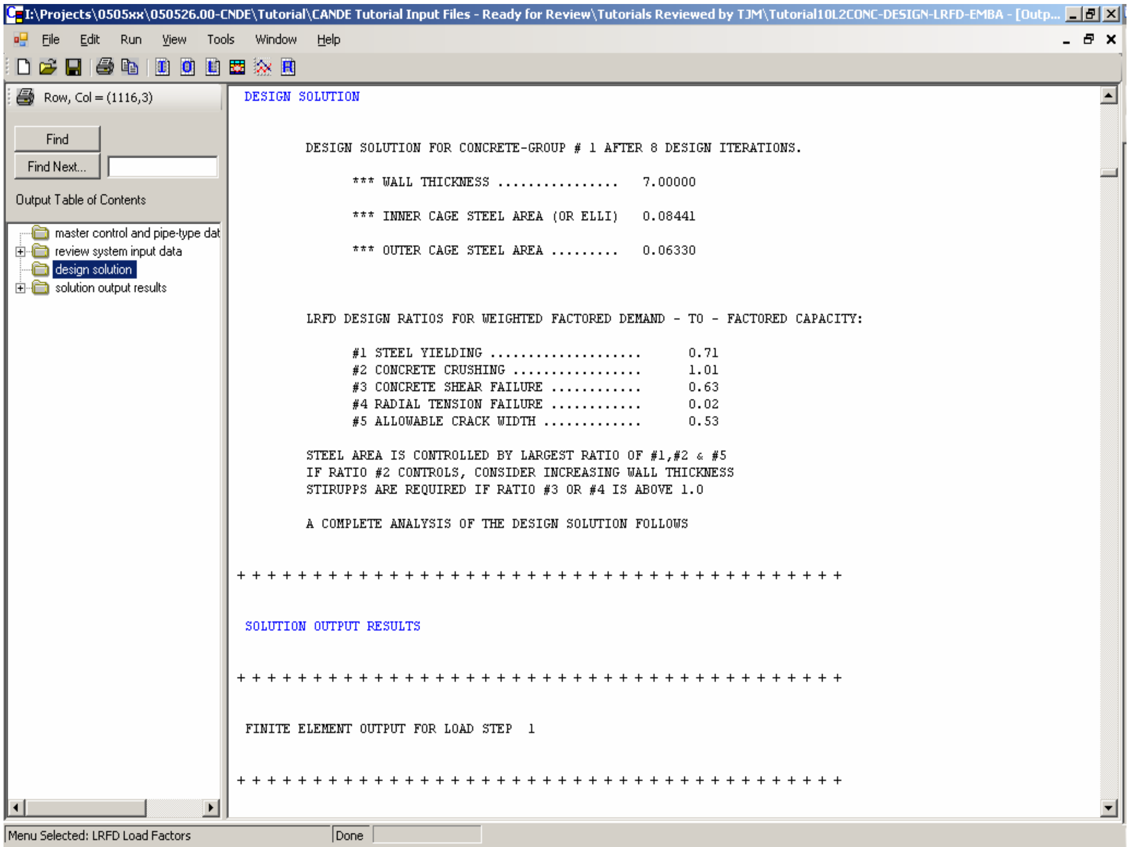


Figure 10-23 – Design Solution

In Figure 10-23 note that the LRFD design ratio for concrete crushing is greater than 1.0. Even in design mode, CANDE does not correct for these design conditions. The user must evaluate the design ratios and accept the input or make necessary design changes.

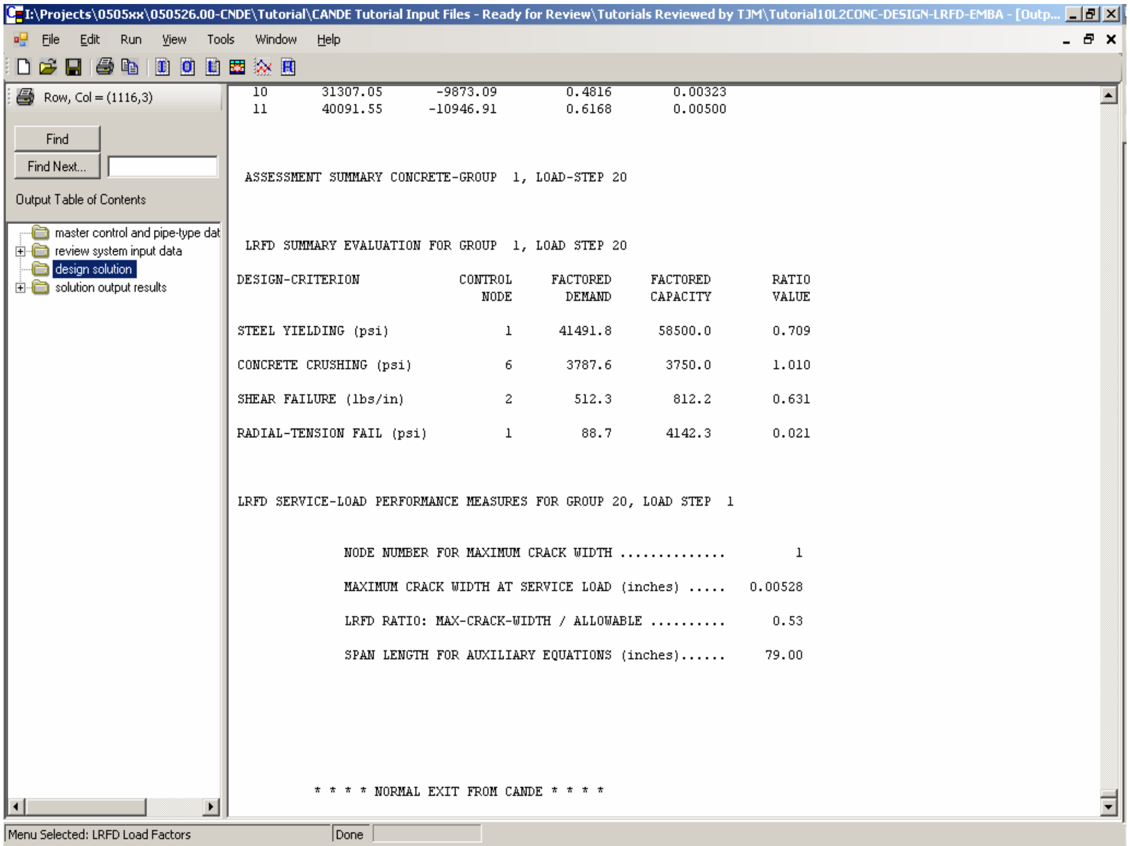


Figure 10-24 – Design Assessment Summary – Load Step 20

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 10-25 shows the Results Generator input screen set to obtain deflection and thrust data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 10-26. The three tabs shown in Figures 10-25 and 10-26 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

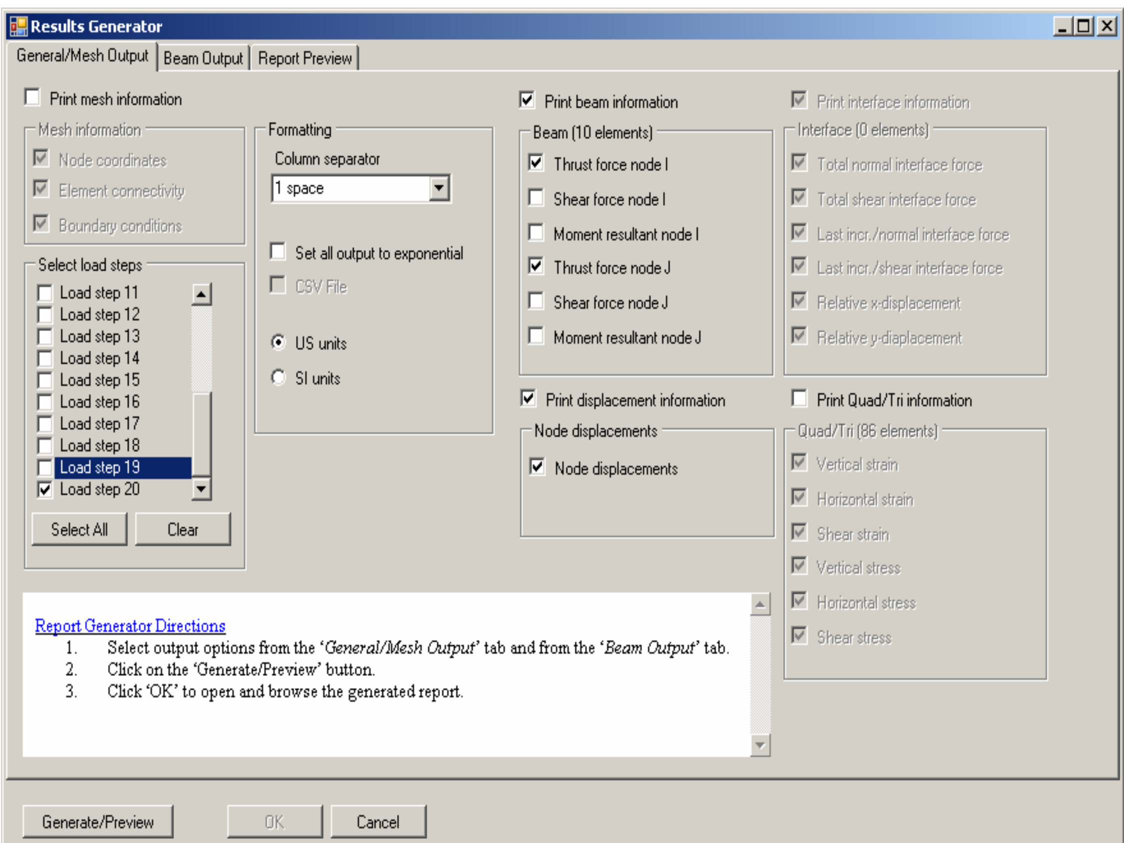
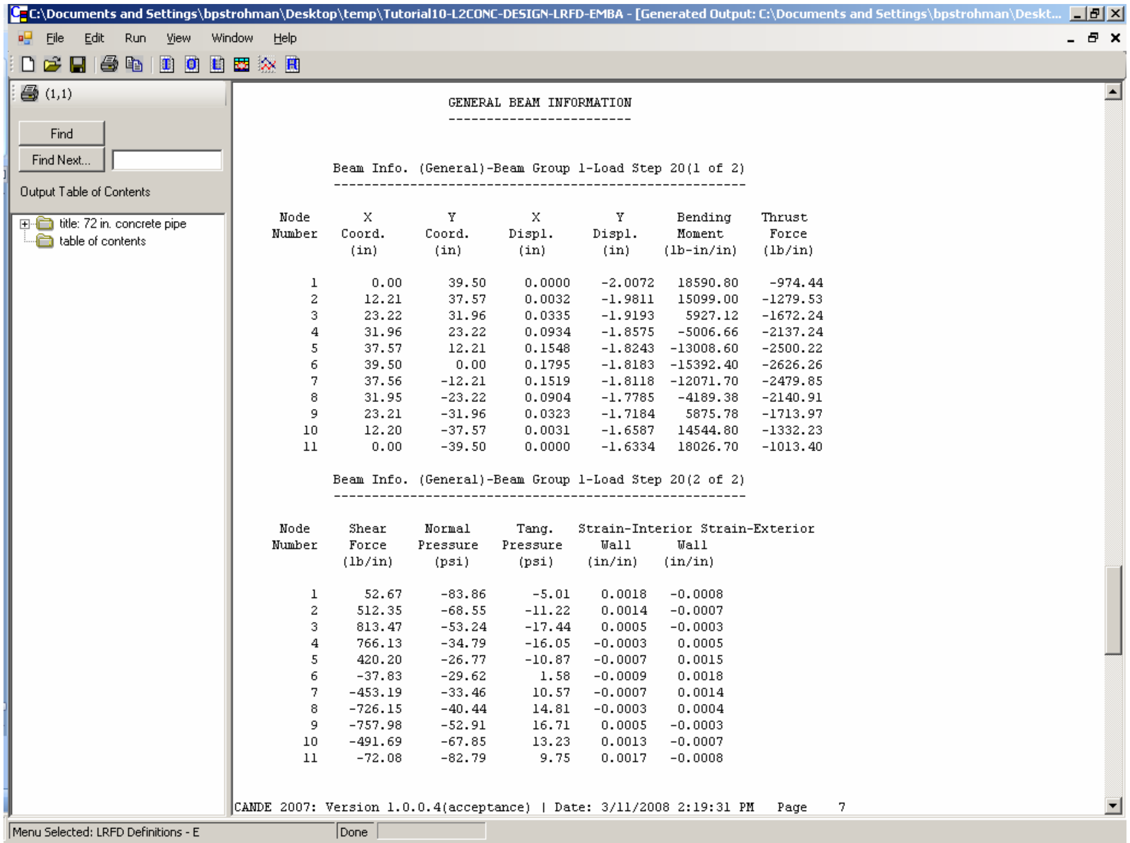


Figure 10-25 – Results Generator Input Screen – Load Step 20 Thrusts and Deflections



GENERAL BEAM INFORMATION

Beam Info. (General)-Beam Group 1-Load Step 20(1 of 2)

Node Number	X Coord. (in)	Y Coord. (in)	X Displ. (in)	Y Displ. (in)	Bending Moment (lb-in/in)	Thrust Force (lb/in)
1	0.00	39.50	0.0000	-2.0072	18590.80	-974.44
2	12.21	37.57	0.0032	-1.9811	15099.00	-1279.53
3	23.22	31.96	0.0335	-1.9193	5927.12	-1672.24
4	31.96	23.22	0.0934	-1.8575	-5006.66	-2137.24
5	37.57	12.21	0.1548	-1.8243	-13008.60	-2500.22
6	39.50	0.00	0.1795	-1.8183	-15392.40	-2626.26
7	37.56	-12.21	0.1519	-1.8118	-12071.70	-2479.85
8	31.95	-23.22	0.0904	-1.7785	-4189.38	-2140.91
9	23.21	-31.96	0.0323	-1.7184	5875.78	-1713.97
10	12.20	-37.57	0.0031	-1.6587	14544.80	-1332.23
11	0.00	-39.50	0.0000	-1.6334	18026.70	-1013.40

Beam Info. (General)-Beam Group 1-Load Step 20(2 of 2)

Node Number	Shear Force (lb/in)	Normal Pressure (psi)	Tang. Pressure (psi)	Strain-Interior Wall (in/in)	Strain-Exterior Wall (in/in)
1	52.67	-83.86	-5.01	0.0018	-0.0008
2	512.35	-68.55	-11.22	0.0014	-0.0007
3	813.47	-53.24	-17.44	0.0005	-0.0003
4	766.13	-34.79	-16.05	-0.0003	0.0005
5	420.20	-26.77	-10.87	-0.0007	0.0015
6	-37.83	-29.62	1.58	-0.0009	0.0018
7	-453.19	-33.46	10.57	-0.0007	0.0014
8	-726.15	-40.44	14.81	-0.0003	0.0004
9	-757.98	-52.91	16.71	0.0005	-0.0003
10	-491.69	-67.85	13.23	0.0013	-0.0007
11	-72.08	-82.79	9.75	0.0017	-0.0008

CANDE 2007: Version 1.0.0.4(acceptance) | Date: 3/11/2008 2:19:31 PM Page 7

Figure 10-26 – Results Generator – Load Step 10 Beam Forces

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 11

May 2008

This page intentionally left blank

11. CANDE TEST PROBLEM 11

11.1 Problem Definition

Analyze a 48 in. inside diameter corrugated plastic (profile) pipe with 40 ft of fill over the top of the pipe using Working Stress (service) analysis. The problem is shown schematically in Figure 11-1. The analysis will be with Level 2, using an automated finite element pipe mesh for a trench installation having interface elements. The automated finite element mesh will be modified using Level 2-extended to change the haunch zones to a user defined soil material and the thickness of bedding layer to 6 in.

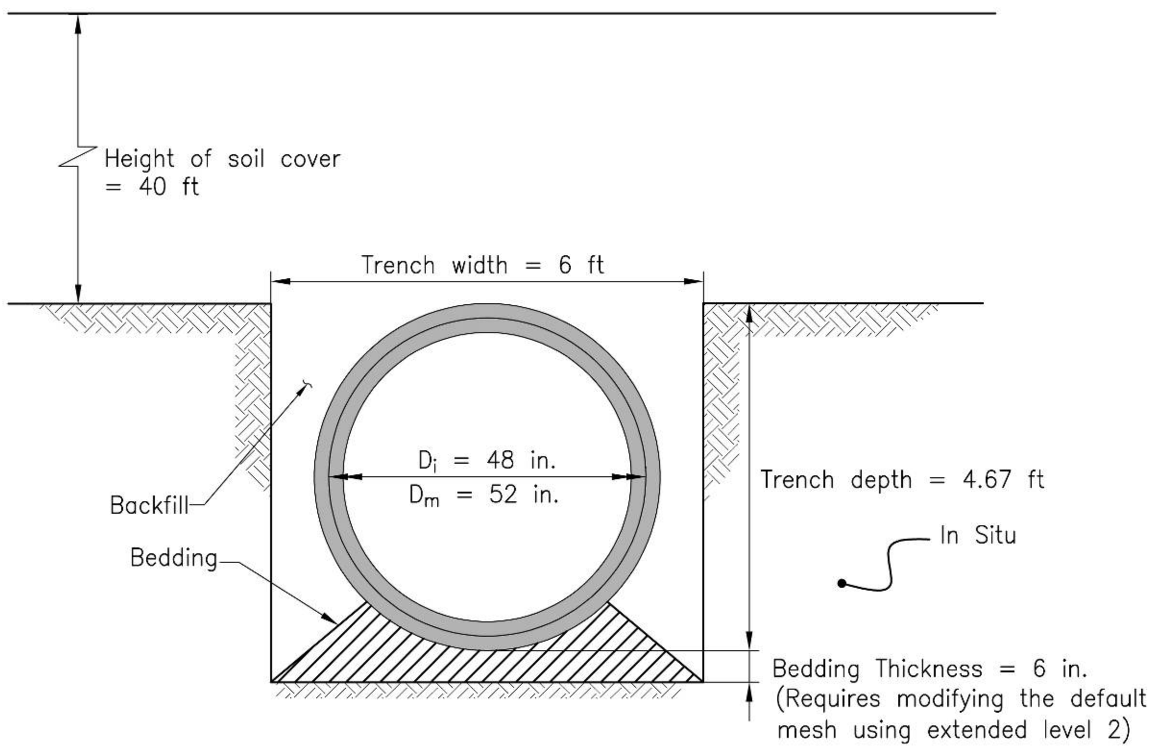


Figure 11-1 Details of Problem 11

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - Service

Solution level - FEM-auto mesh (Level 2)

Canned mesh type - Pipe mesh

Soil mesh pattern - Trench

Interface elements (pipe only) - Pipe-soil

MOD-Make changes to the basic mesh - check on (problem requires using Level 2-extended to change the haunch zones to a user defined soil material and the thickness of bedding layer to 6 in.

Number of nodes to be changed with new coordinates - 7

Number of elements to be changed with new properties - 2

Pipe material type - Plastic

Wall section type - Profile

Soil parameters - Canned overburden dependent soil models for each of the following materials:

In situ soil - Cohesive soil-good

Bedding soil - Granular soil-good

Backfill soil - Granular soil-good

Backpack soil - Mixed soil-fair

Soil density - 120 lb/ft^3 for all soils

Type of plastic - HDPE

Loading duration - Long term

Analysis mode - Small deformation

Young's modulus for long term loading - 22,000 lb/in.^{2}

Ultimate stress limit for long term loading - 900 lb/in.^{2}

Poisson's ratio - 0.4

Density of plastic material used for body weight - 0.034 lb/in.^{3}

Length of profile period - 5.4 in.

Total height of profile section - 4.0 in.

Web angle with the horizontal - 77 degrees

Web thickness - 0.24 in.

Web "k" value for edge support coefficient - 4.0

Number of horizontal elements in profile - 3

Include local buckling calculations - check on

Plastic element properties: The following summarizes the width, thickness, and edge support coefficients for the elements of the pipe profile:

	Element Identifier	**Length, in.**	**Thickness, in.**	**Edge support coefficient (k)**
1	Valley	1.50	0.28	4
2	Liner	3.88	0.12	4
3	Crest	2.31	0.20	4

Average diameter of pipe - 50 in.

Height of soil cover - For trench installations CANDE calculates height of soil cover from the top of the trench (See *User Manual*, Chapter 5, C-2). To achieve a total height of fill of 40 ft over the top of the pipe, set the height of soil cover to 40 ft (See Figure 11-1).

Density of soil above truncated mesh - 120 lb/ft^3

Number of construction steps - See *User Manual*, Chapter 5, C-3 – The default mesh provides 5 construction steps to the top of the mesh, which provides approximately 1.5 pipe diameters of fill over the pipe. For the remaining depth of fill, use 5 steps, making a total of 10 construction steps.

Trench depth - Depth from bottom of pipe to top of trench (See *User Manual,* Chapter 5, C-4). Set equal to 4.67 ft per Figure 11-1.

Trench width - 6 ft

Interface material properties - The following table summarizes the material property definitions for the interface elements:

Interface number	Angle from x-axis to normal interface (degrees) *	Coefficient of friction between nodes i and j	Tensile breaking force of contact nodes
1	-90	0.5	1
2	-72	0.5	1
3	-54	0.5	1
4	-36	0.5	1
5	-18	0.5	1
6	0	0.5	1
7	18	0.5	1
8	36	0.5	1
9	54	0.5	1
10	72	0.5	1
11	90	0.5	1

The interface angles for this problem were computed in the following manner:

Interface angle in the pipe elements, $\theta(i) = -90 + (i-1)*180 / (n-1)$

where:

i = 1, 2,…..n

n = number of nodes in the pipe

11.2 Creating the CANDE Input Document

Figures 11-2 through 11-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 3, CANDE will display a screen indicating that an input path is initiated - click “Finish.” The user is prompted for a filename and directory.

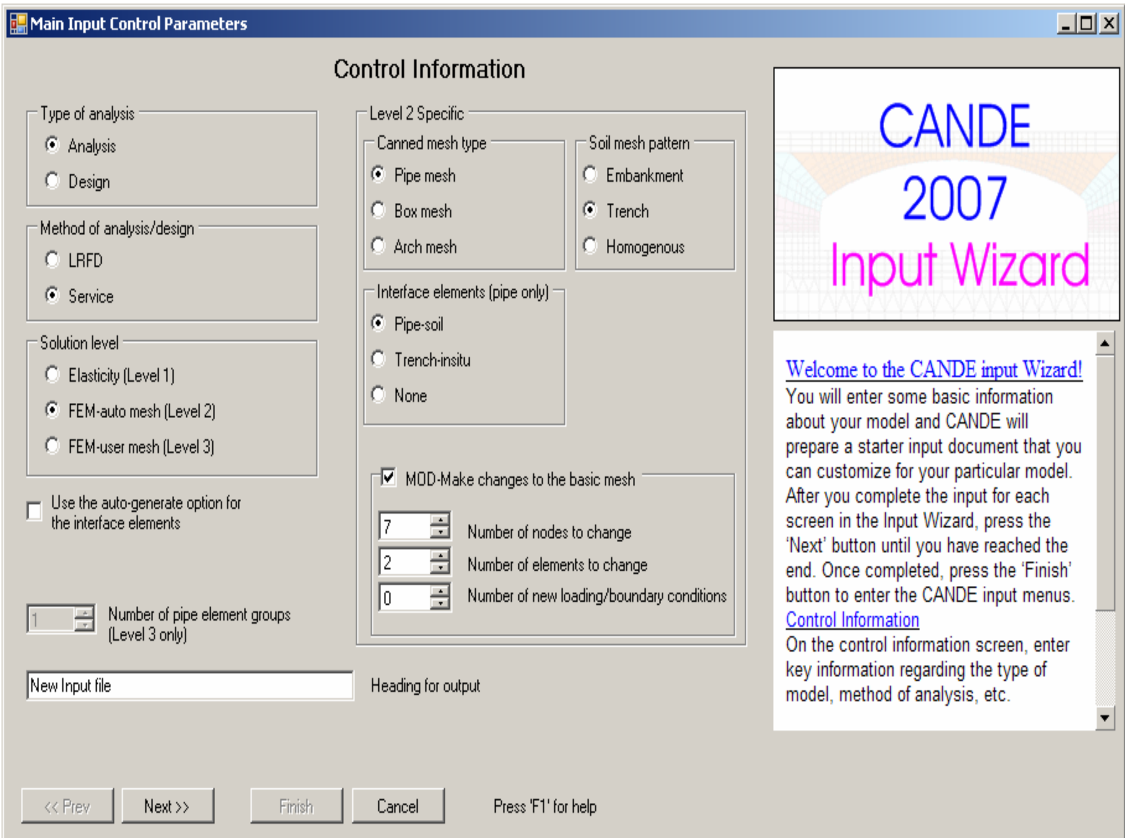


Figure 11-2 – Input Wizard, Screen 1

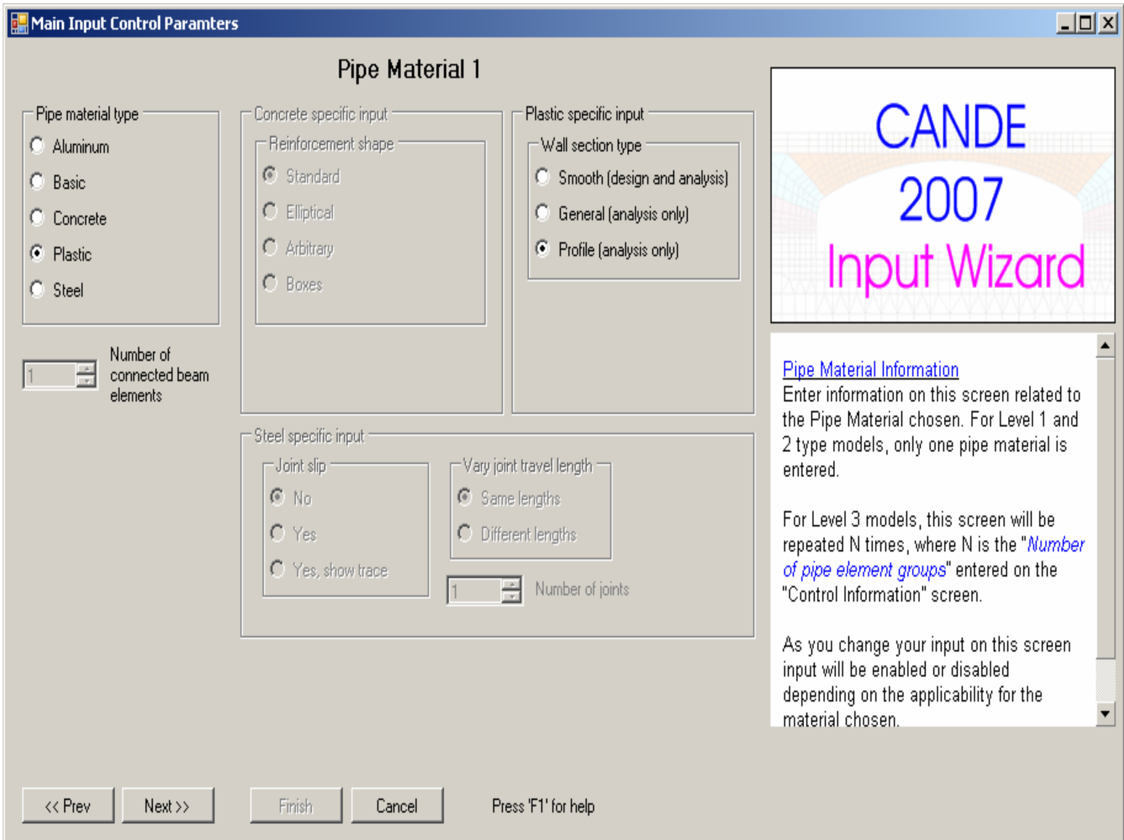


Figure 11-3 – Input Wizard, Screen 2

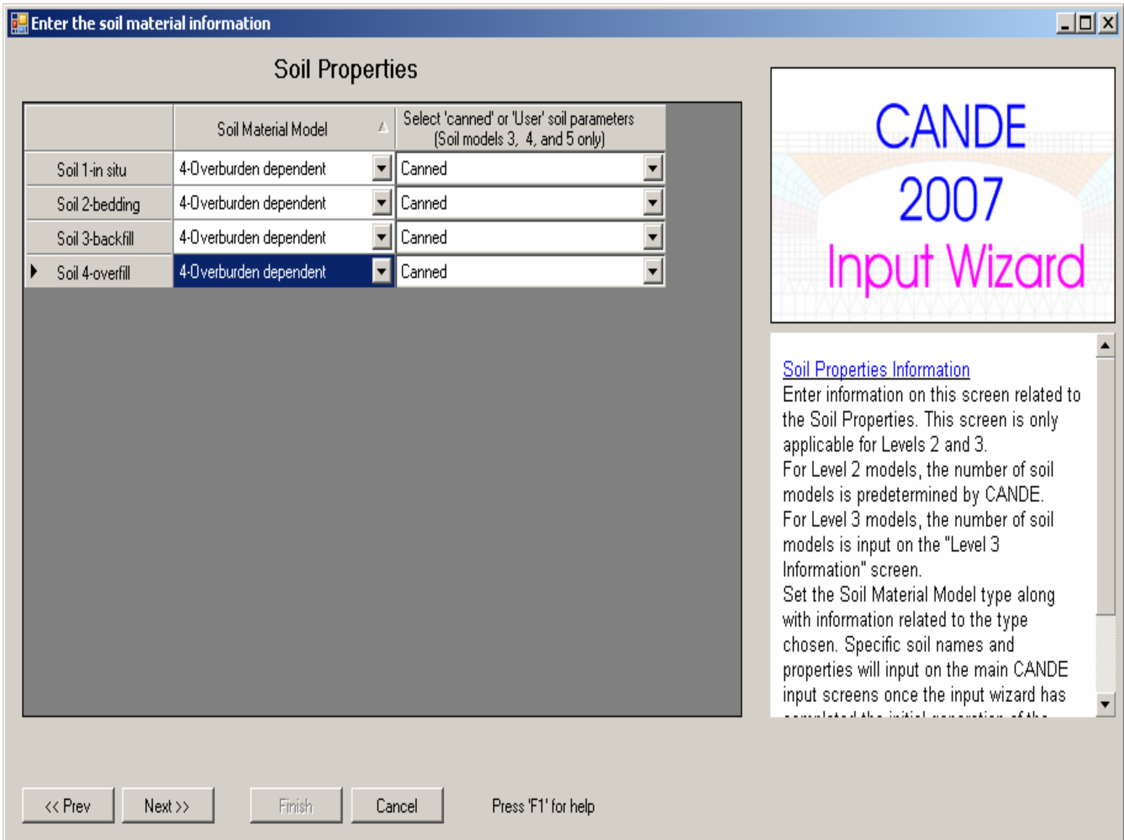


Figure 11-4 – Input Wizard, Screen 3

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 11-5. Enter an appropriate heading for output and click “Accept Input.”

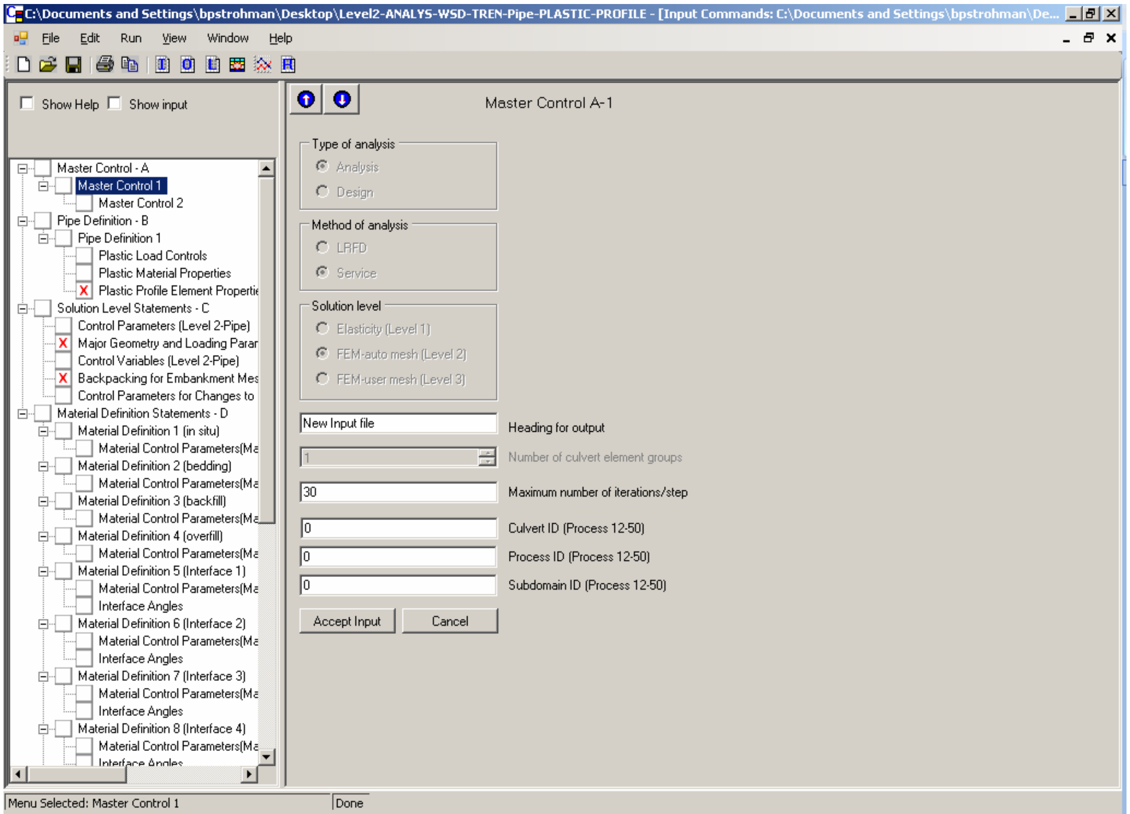


Figure 11-5 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 11-5 shows “X” for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 11-6 through 11-12 and Figures 11-16 and 11-17 show the completed input for the standard screens requiring data for the tutorial, except that only one material definition is shown. Input the remaining definition screens. As input is modified, a ‘blue circle’ icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click “Accept Input” and the blue circle in the control panel will disappear indicating all fields have data. Figures 11-13 through 11-15 and Figures 11-18 and 11-19 show the use of the Level 2-extended to change the haunch zones to a user defined soil material and the thickness of bedding layer to 6 in. Note that the user defined soil material will be input in the “file/open text input” mode after all of the other material properties and problem definitions have been defined.

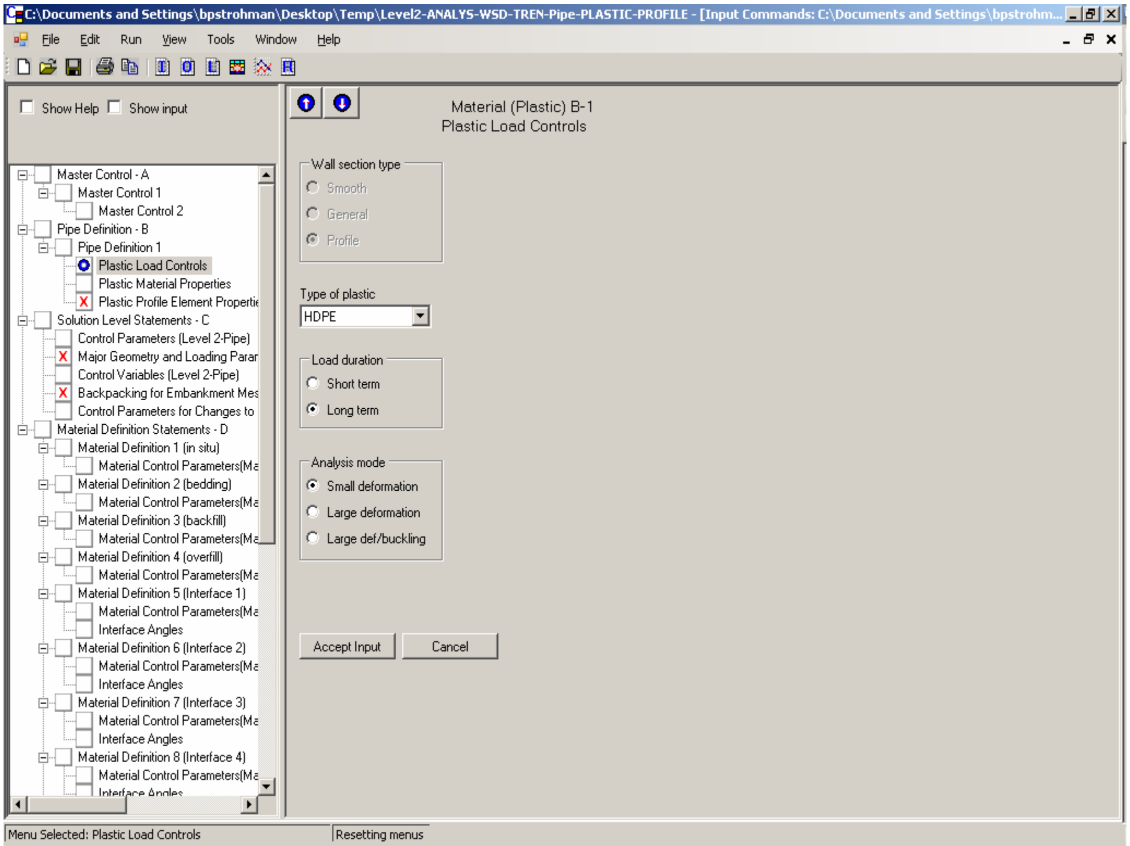


Figure 11-6 – Input Screen B-1

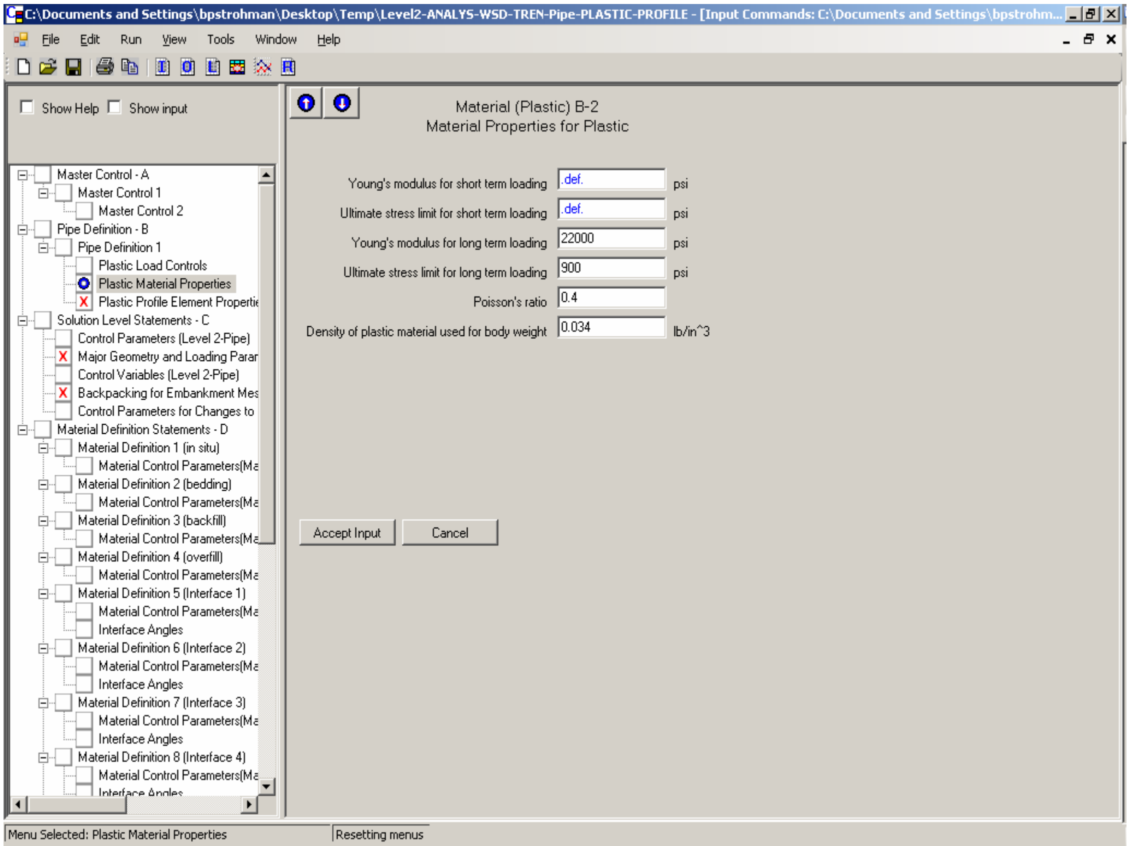


Figure 11-7 – Input Screen B-2

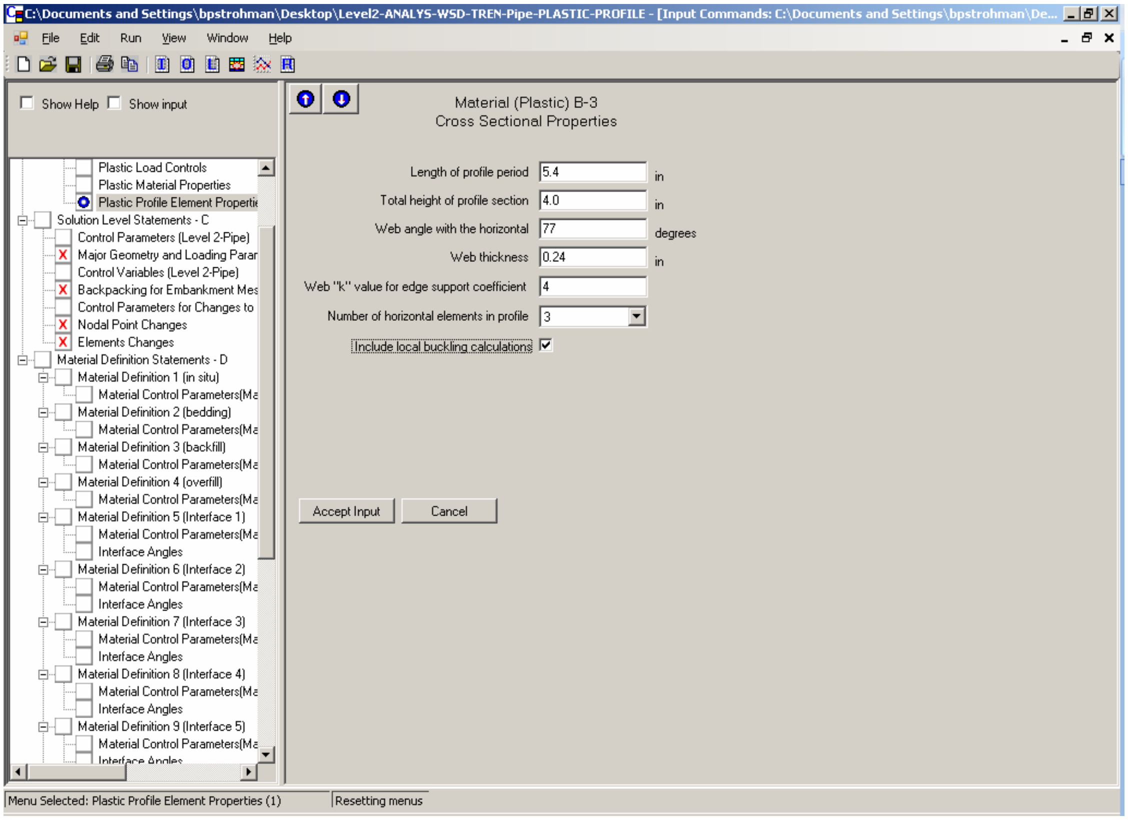


Figure 11-8 – Input Screen B-3

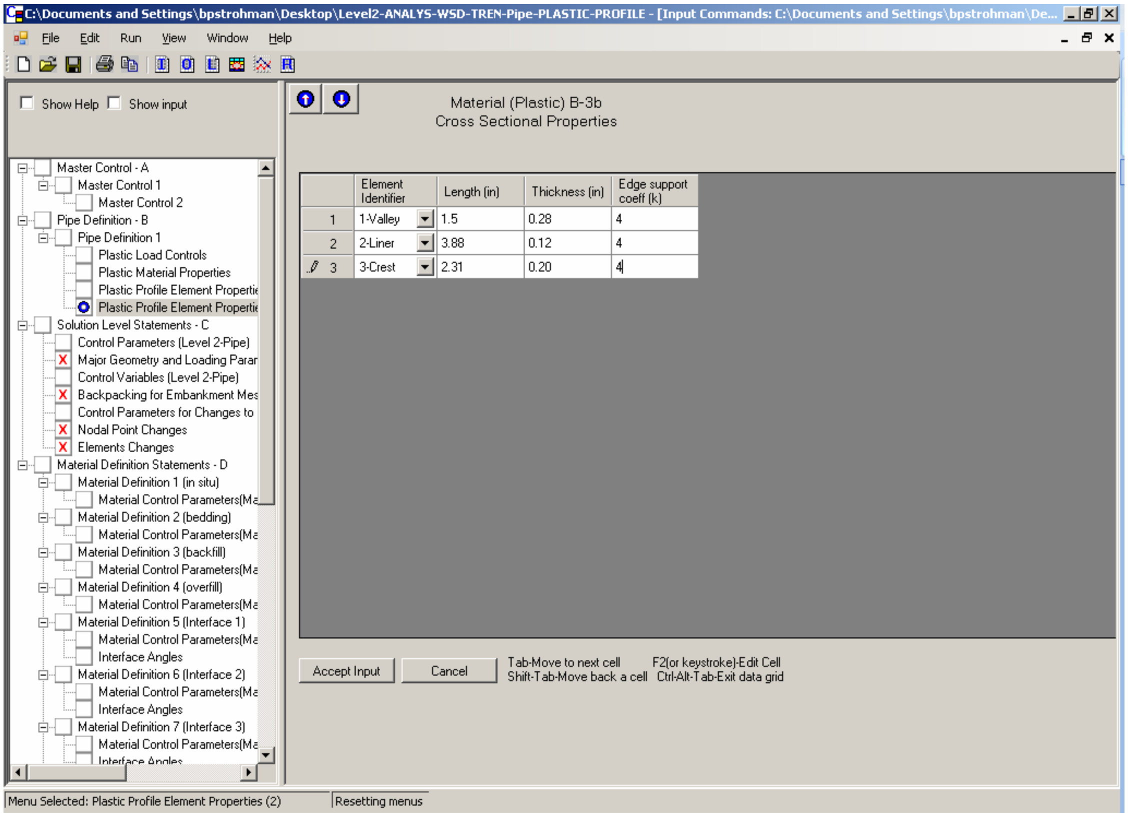


Figure 11-9 – Input Screen B-3b

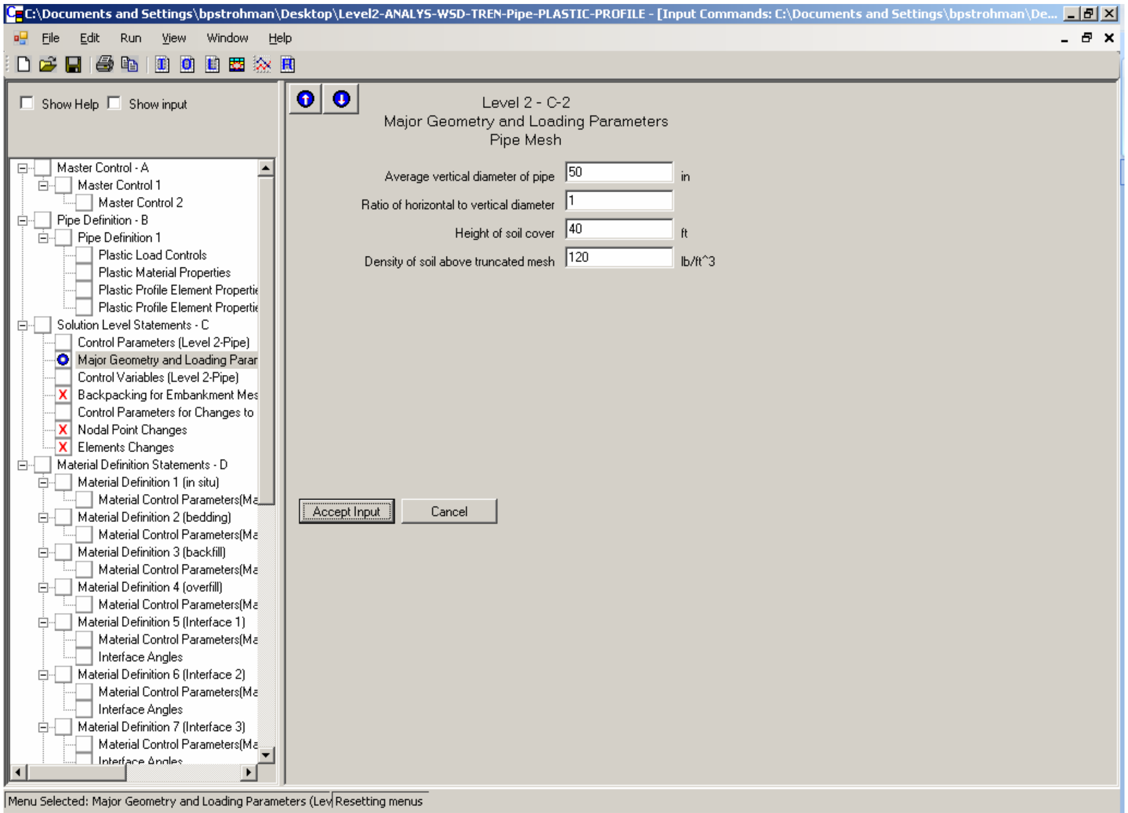


Figure 11-10 – Input Screen C-2

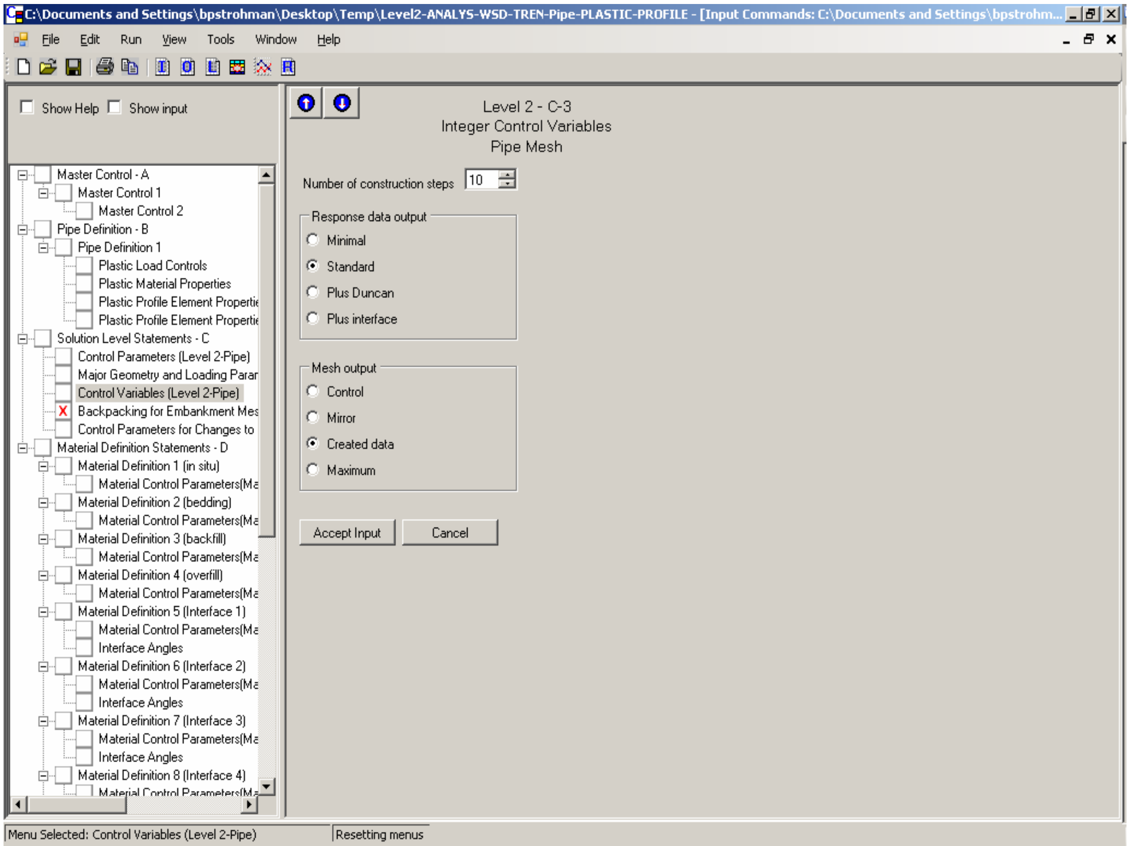


Figure 11-11 – Input Screen C-3

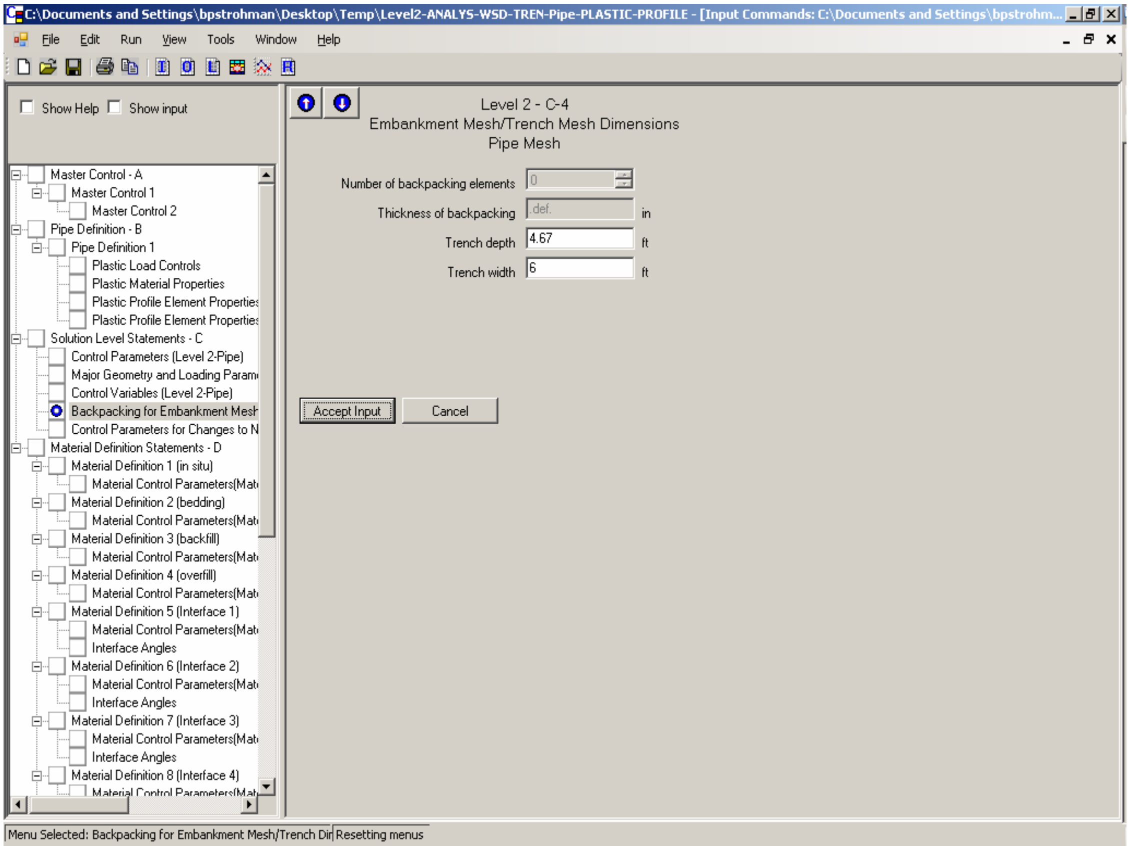


Figure 11-12 – Input Screen C-4

The automated finite element mesh will be modified using Level 2-extended to change the haunch zones to a user defined soil material and the thickness of bedding layer to 6 in. Figures 11-13 through 11-15 show the completed input screens for the Level 2-extended, which require changing the y-coordinate of 7 nodes and the material property for 2 elements. The element and node numbers are identified in Chapter 5, C-4 of the *User Manual* or can be obtained by making a trial run without the modified mesh and then using the mesh plot.

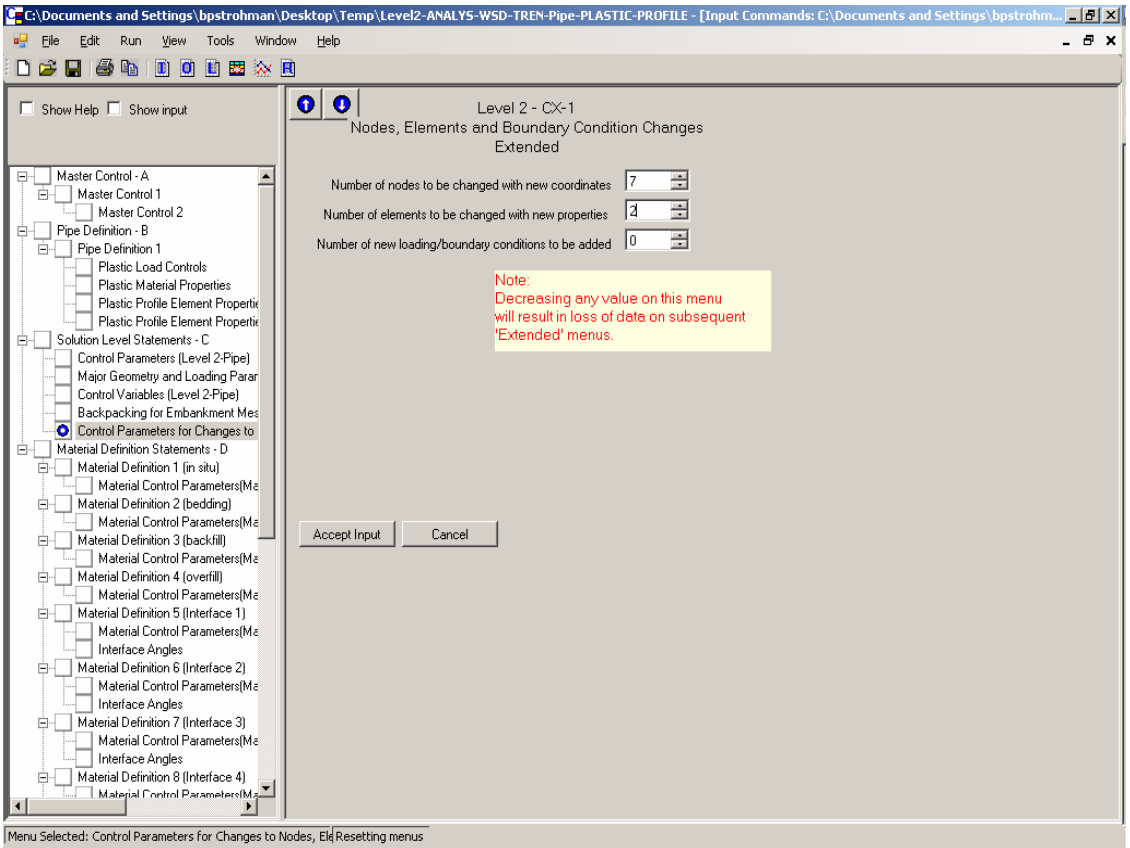


Figure 11-13 – Input Screen CX-1

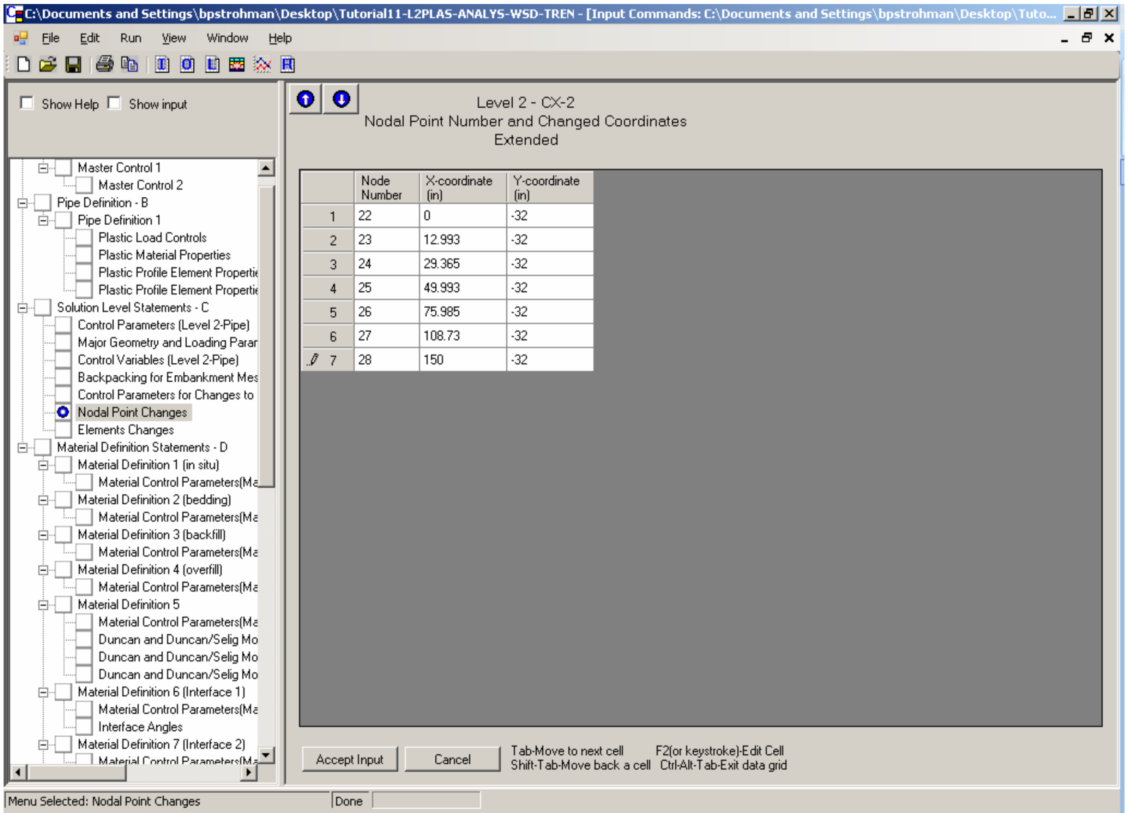


Figure 11-14 – Input Screen CX-2

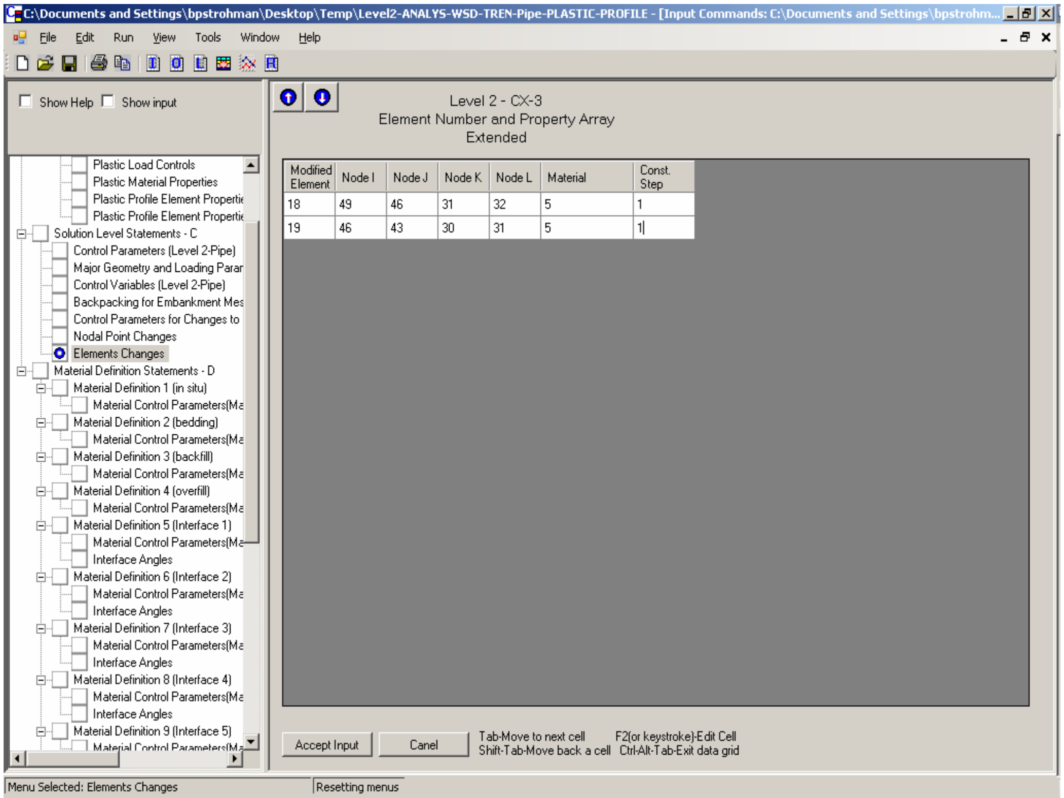


Figure 11-15 – Input Screen CX-3

Upon clicking 'Accept Input' the user will be prompted as shown in Figure 11-15a. The user has the option for CANDE to automatically add the new soil property and then define the properties within in the GUI or to select 'No' and add the properties manually through the CANDE text editor. For illustrating the use of the CANDE text editor, we will select 'No' and add the soils manually.

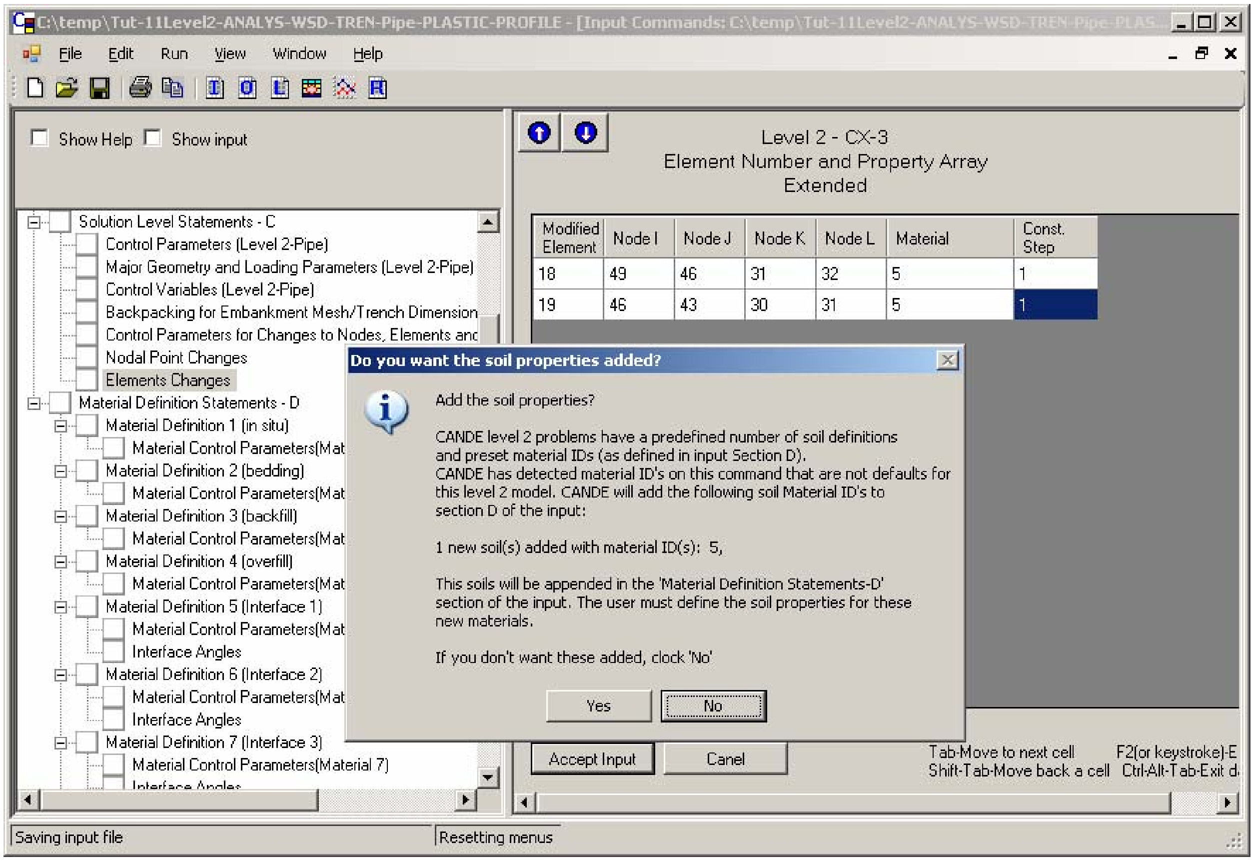


Figure 11-15b – Input Screen CX-3 – Auto-material add feature

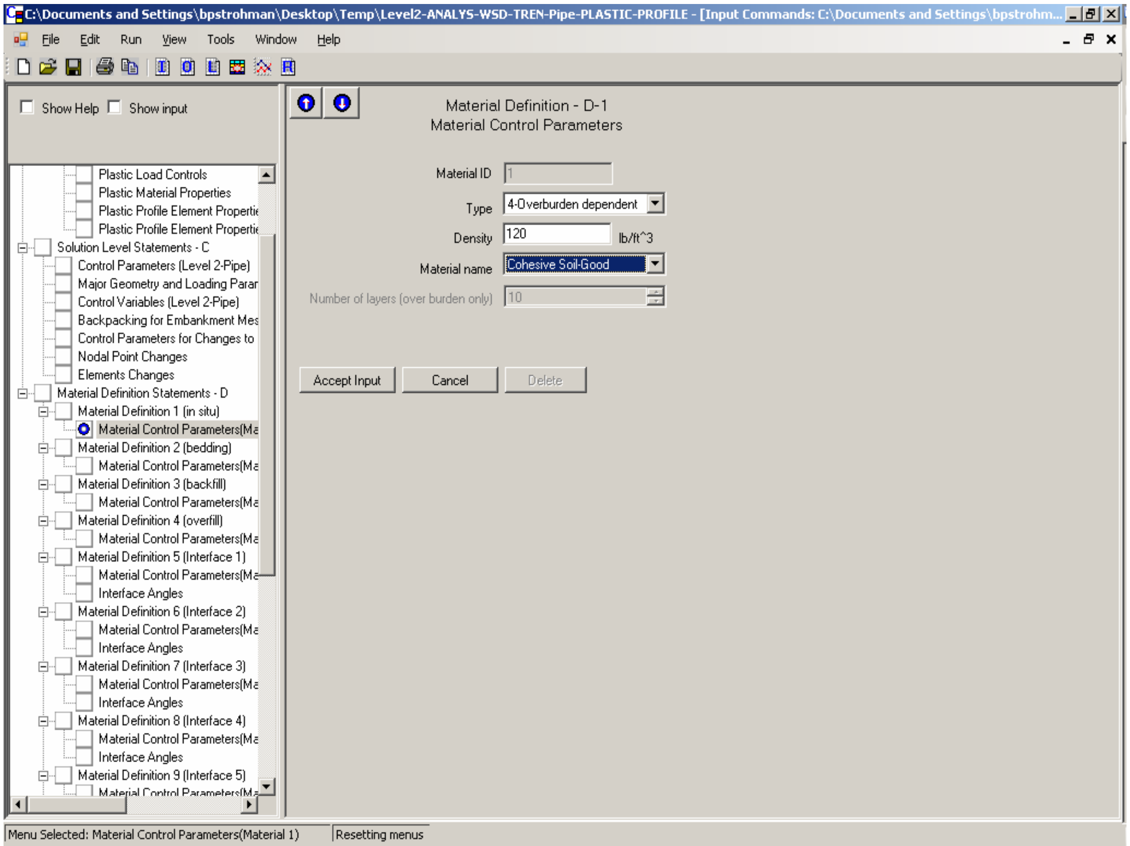


Figure 11-16 – Input Screen D-1 for Material 1 – In Situ Soil
(Note: Repeat for Materials 2, 3, and 4 with values provided in the 'Problem Definition')

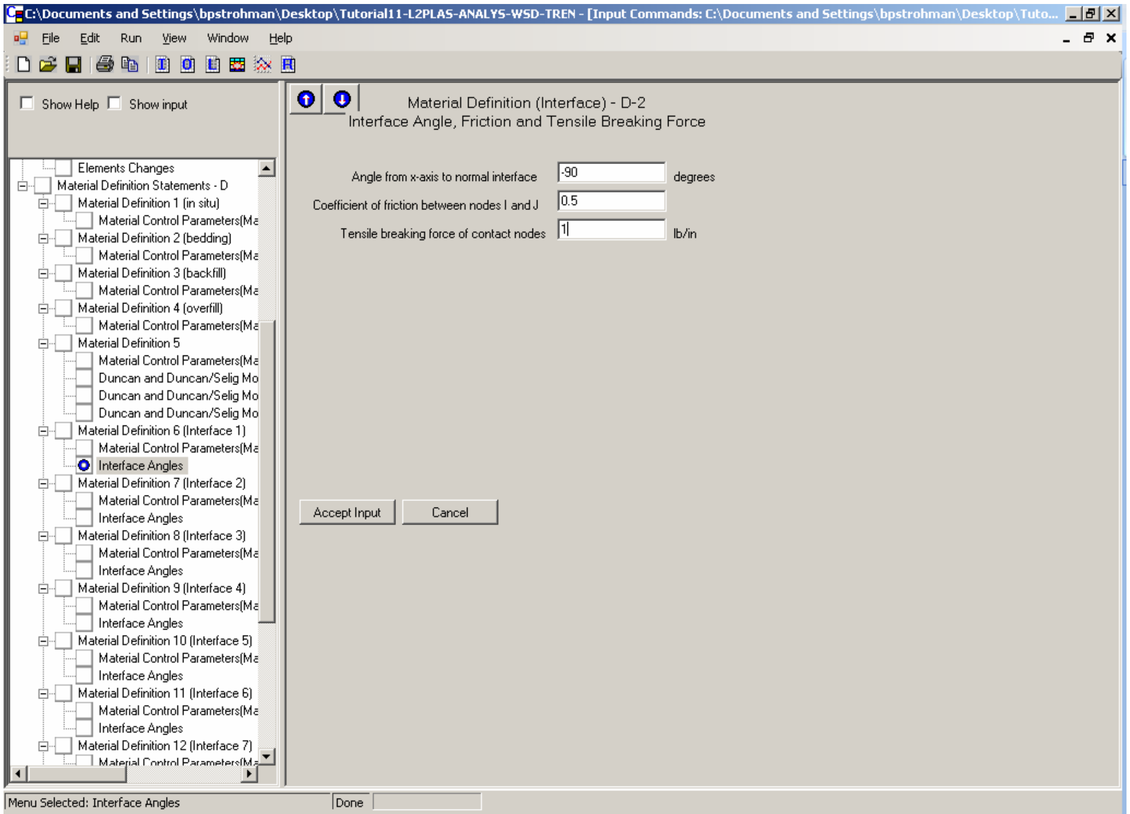


Figure 11-17 – Input Screen D-2 for Material 5 (Interface 1)

(Note: Repeat for Materials 6 through 15 with values provided in the ‘Problem Definition’)

When all input using the graphic interface is complete, save the input file and reopen in text input mode by going to “File/open text input.” The screen should look like Figure 11-18.

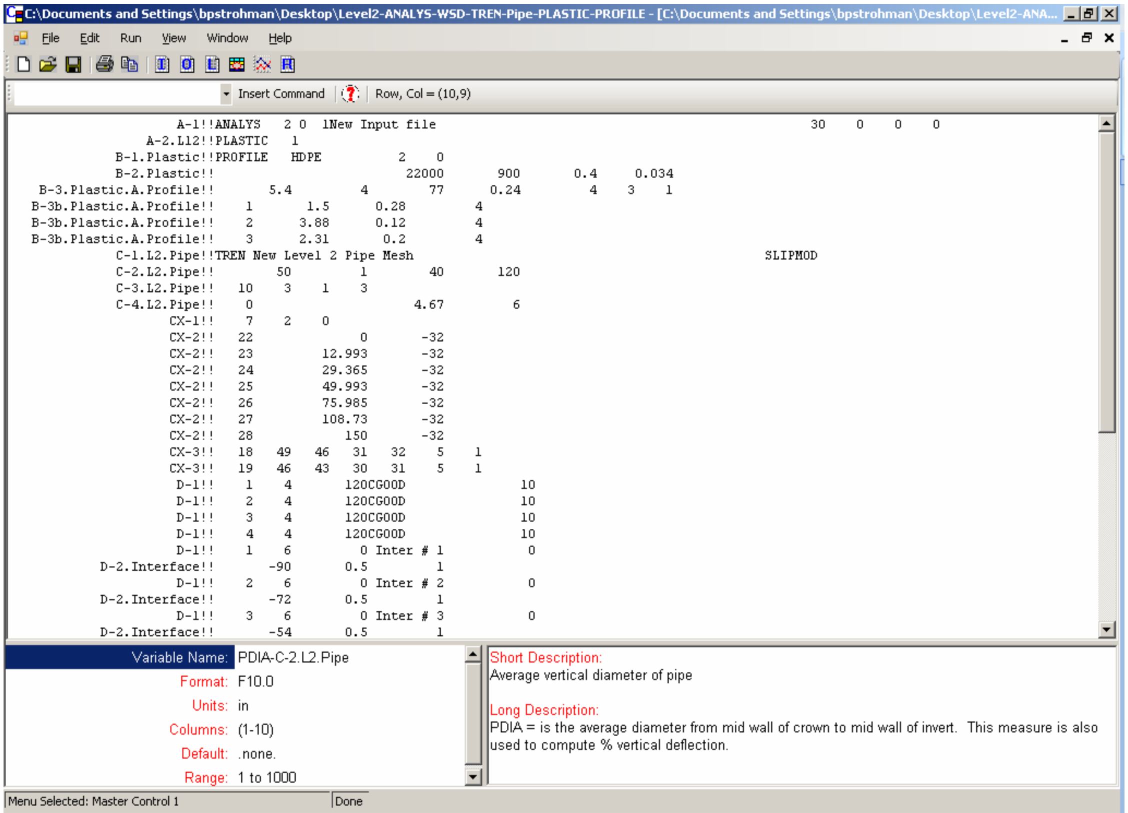


Figure 11-18 – Text Input Screen Before Being Modified

Once in this format, input a user defined soil material type using the Duncan/Selig formulation described in Chapter 5, D-3, and D-4 of the *User Manual.* Use the following properties to define the user defined soil:

Cohesion intercept - 0 psi

Initial friction angle - 23 degrees

Reduction of friction angle - 0 degrees

Magnitude of initial tangent modulus - 16

Exponent for initial tangent modulus - 0.95

Ratio of actual failure stress to model's ultimate - 0.55

Magnitude of tangent bulk modulus, Bi/Pa, Selig hyper form - 1.3

Bulk modulus parameter, eu, Selig hyper form - 0.43

Alternate form using constant Poisson's ratio - 0

The modified text input file is shown in Figure 11-19.

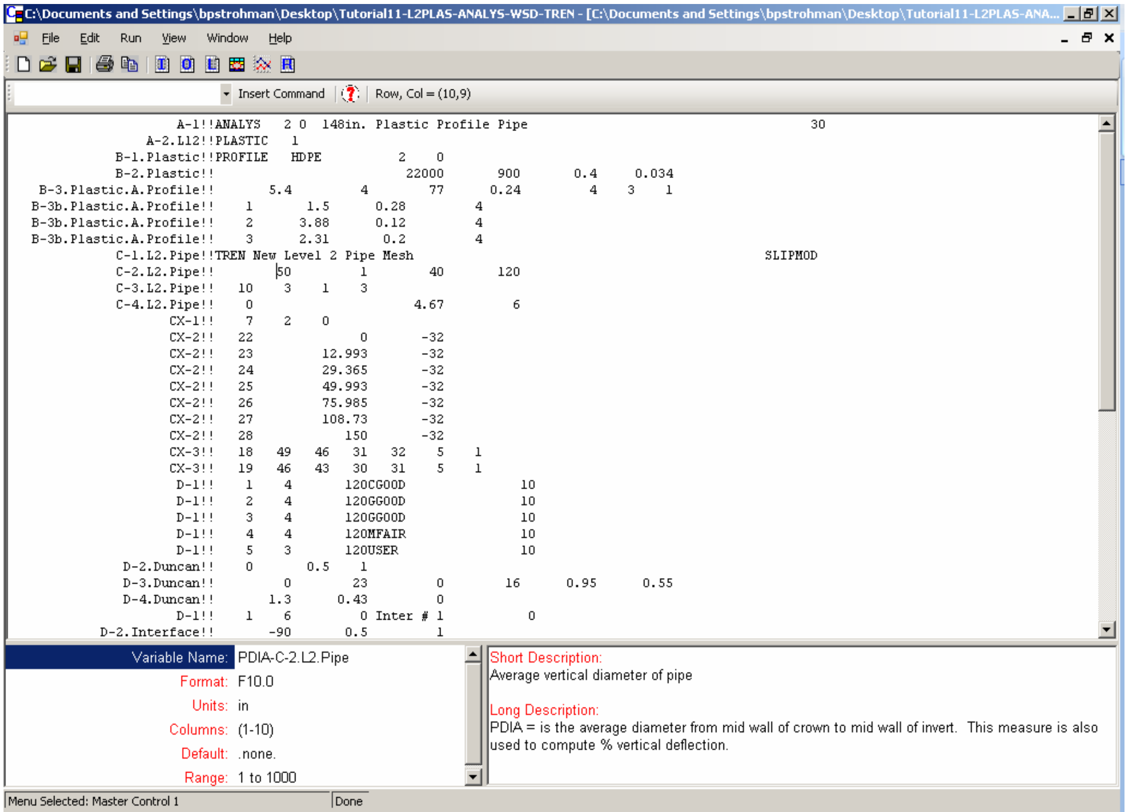


Figure 11-19 – Text Input Screen After Being Modified

After inputting the user defined material properties in the text input mode, save the file and then reopen the problem as a regular CANDE file. There should now be five material soil definitions and 16 total material definitions.

When all input is complete, click “Run” and “CANDE-2007” on the main tool bar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “Analysis Complete” window will appear. Click okay and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the text input option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

11.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the “View” menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The “Plotting Parameters” button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click “View” and then “Mesh Plot” to open the mesh plot. Open the plotting parameters menu and click the check box “Show constr. increment numbers.” Click “OK,” set Load Increment to 5 to show the entire mesh (the remaining load is placed above the mesh – see *User Manual*) and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 11-20.

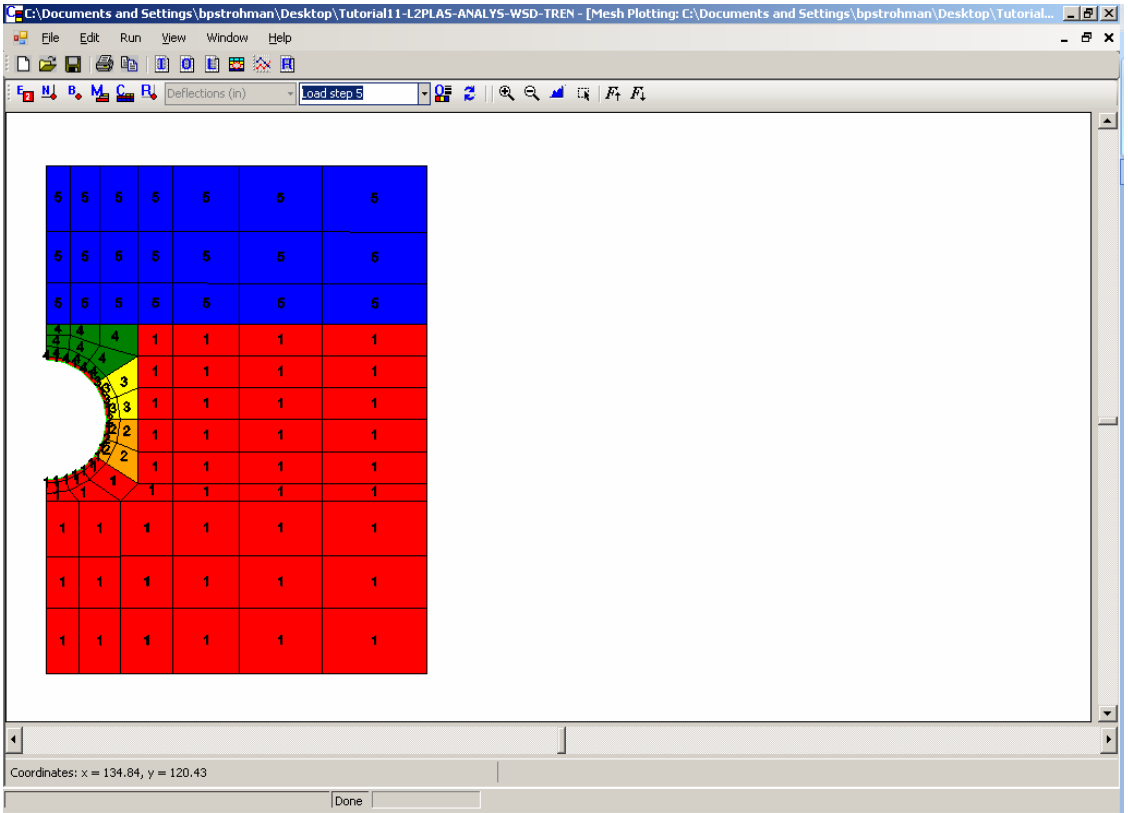


Figure 11-20 – Mesh Plot for Load Steps 1 to 5

Now open the plotting parameters menu and click the check box "Show material numbers." Then click on the toolbar icon to turn on the material information. There should be 5 distinct material zones. The mesh plot should look like Figure 11-21.

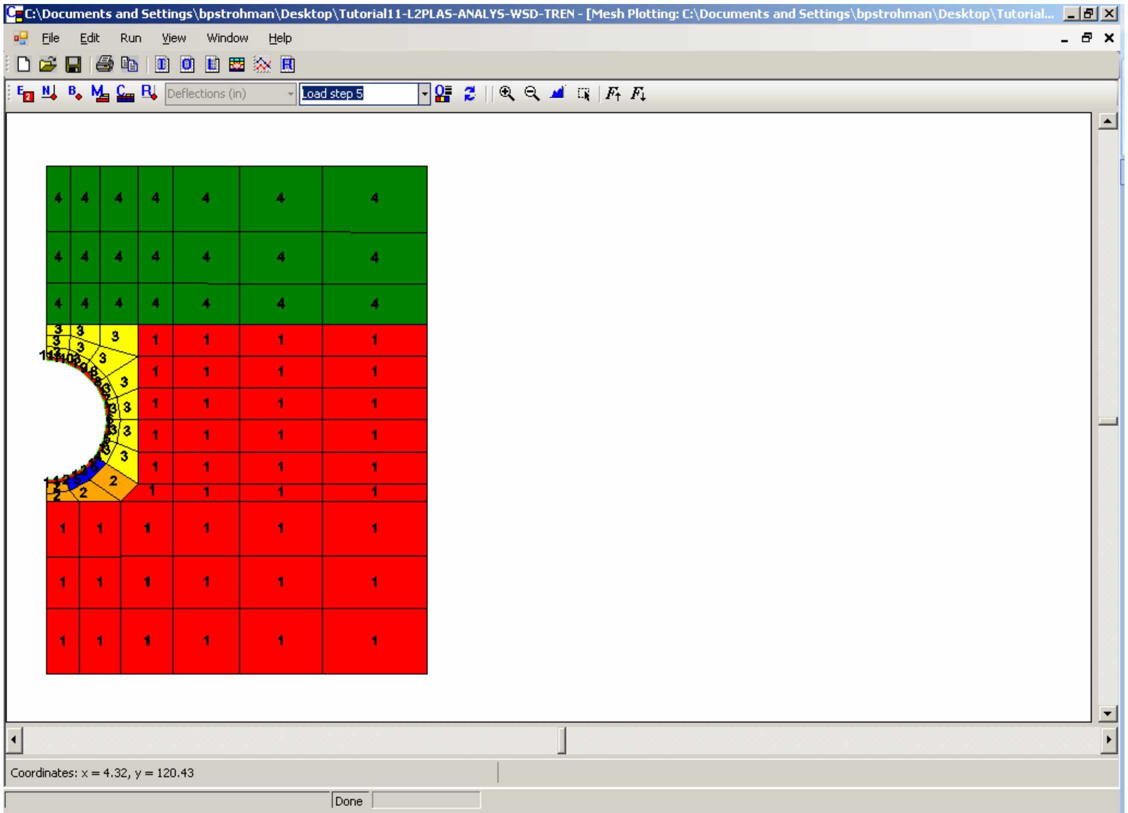


Figure 11-21 – Mesh Plot for Material Zones 1 to 5

To view deflections, open the plotting parameters window and set the deflection magnification factor to 2, click the “Ignore deflections/by load incr.” check box and set the Load Increment to 1 (see Figure 11-22).

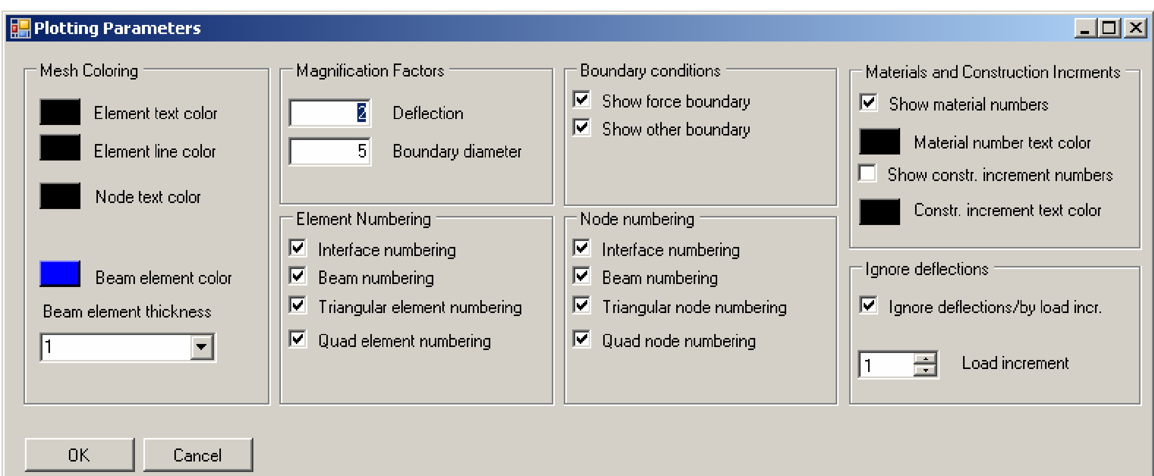


Figure 11-22 – Mesh Window Plotting Parameters

Close the window, click the icon to “Turn on/off selected output results,” and set the drop down box to deflection. Note that if the Load Increment is set to 1, the deflections due to Load Increment are shown, but when the Load Increment is set to 2, the Increment 1 deflections are ignored. Set the increment to 10 and the screen should look like Figure 11-23. Other mesh output parameters can also be selected and inspected to determine if the results are consistent with the design intent.

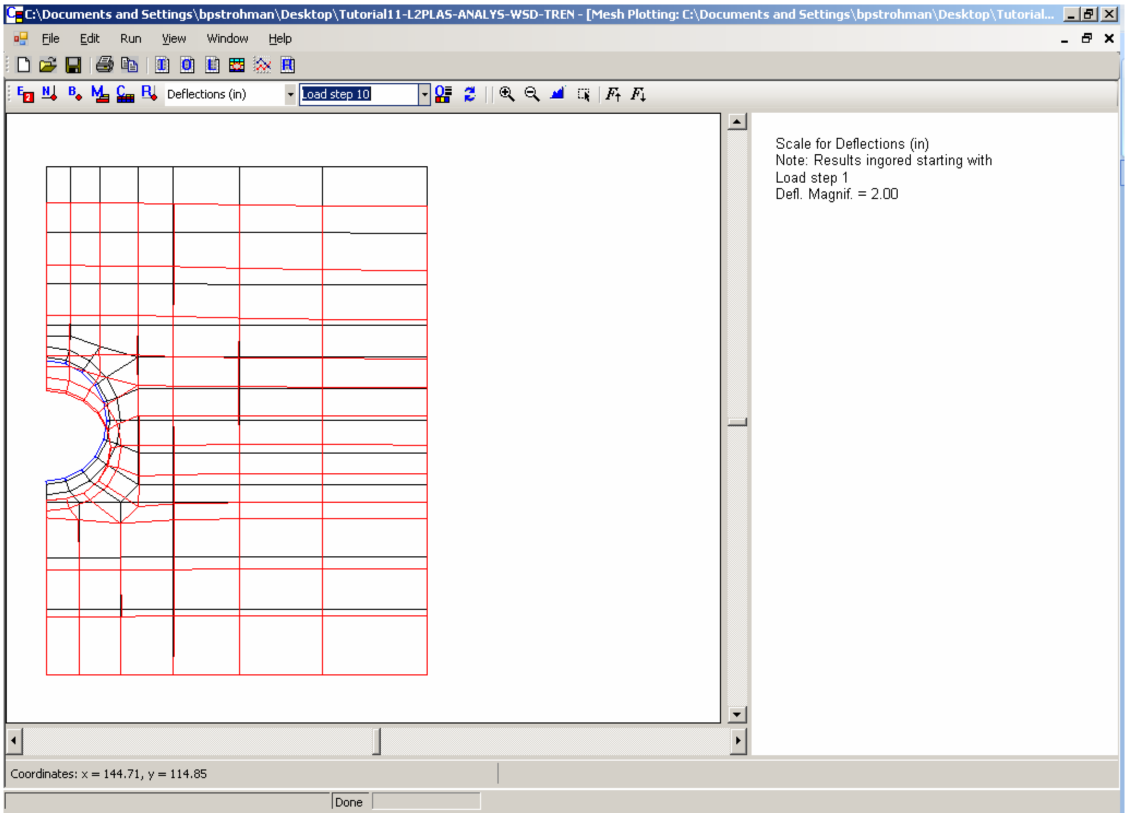


Figure 11-23 – Deflections, Excluding Those Due to Self Weight of In Situ Soil

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 11-24 compares the bending moments after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 3, 6, and 10, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Bending moment” and the screen should appear as Figure 11-24.

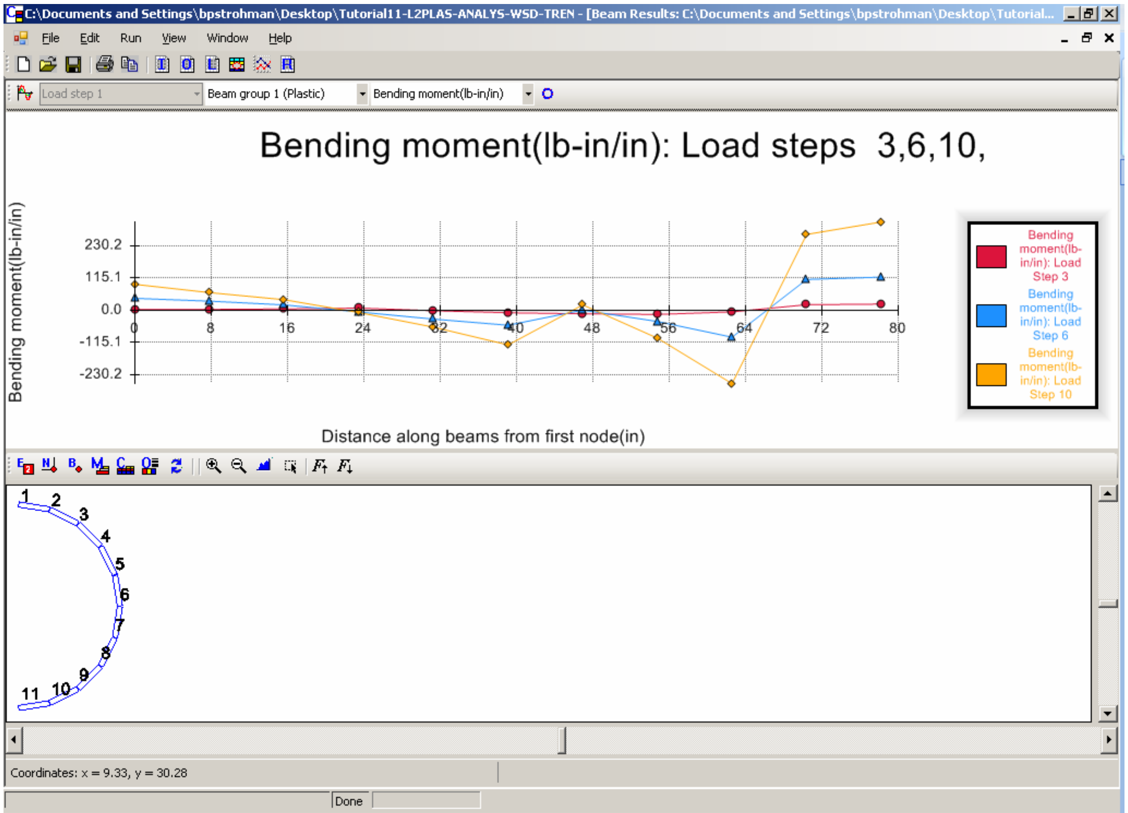


Figure 11-24 – Bending Moments for Load Steps 3, 6, and 10

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 11-25) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 11-25 shows the final assessment printed at the end of the file.

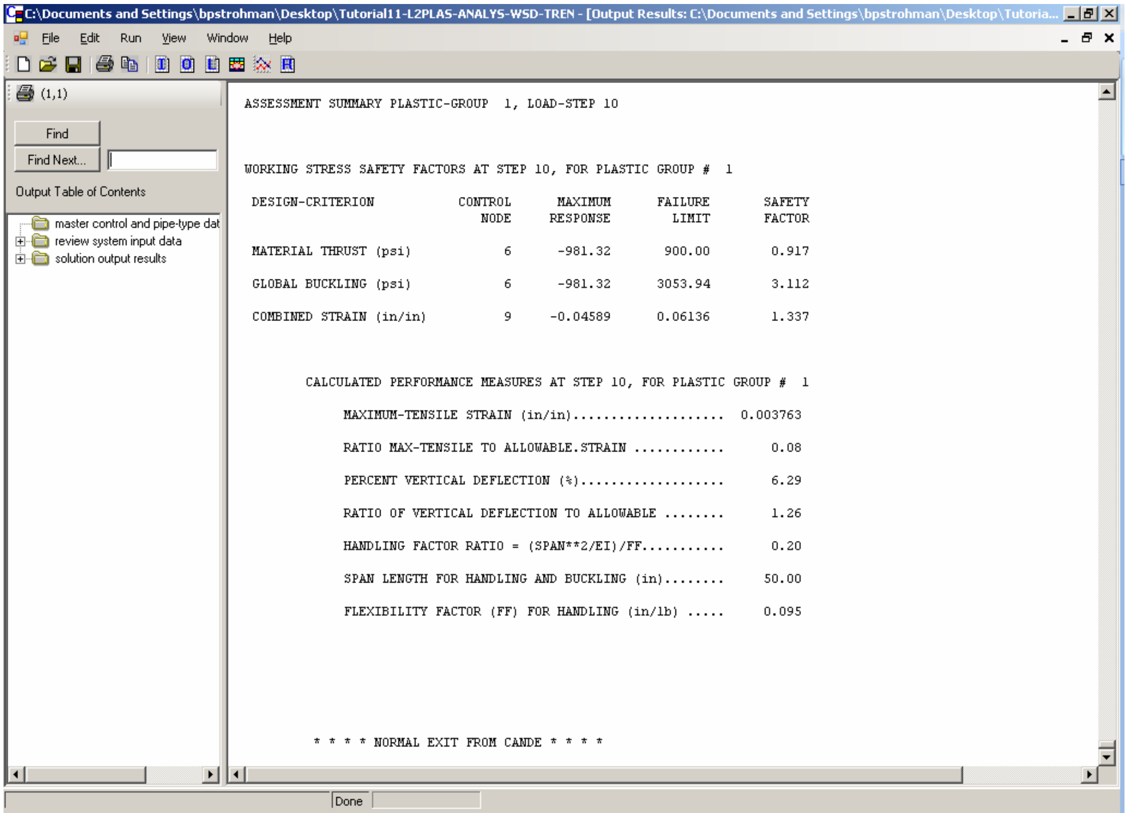


Figure 11-25 – Design Assessment Summary – Load Step 10

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 11-26 shows the Results Generator input screen set to obtain deflection, thrust, and moment data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 11-27. The three tabs shown in Figures 11-26 and 11-27 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

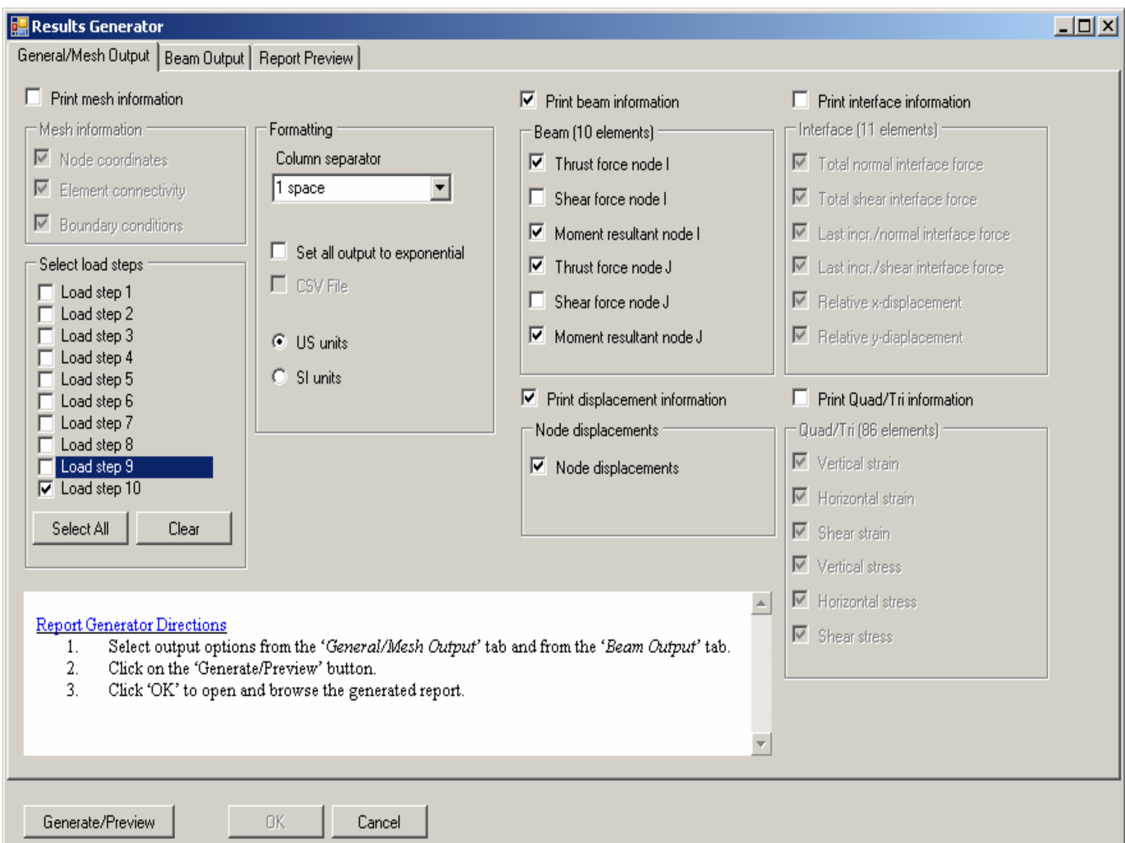
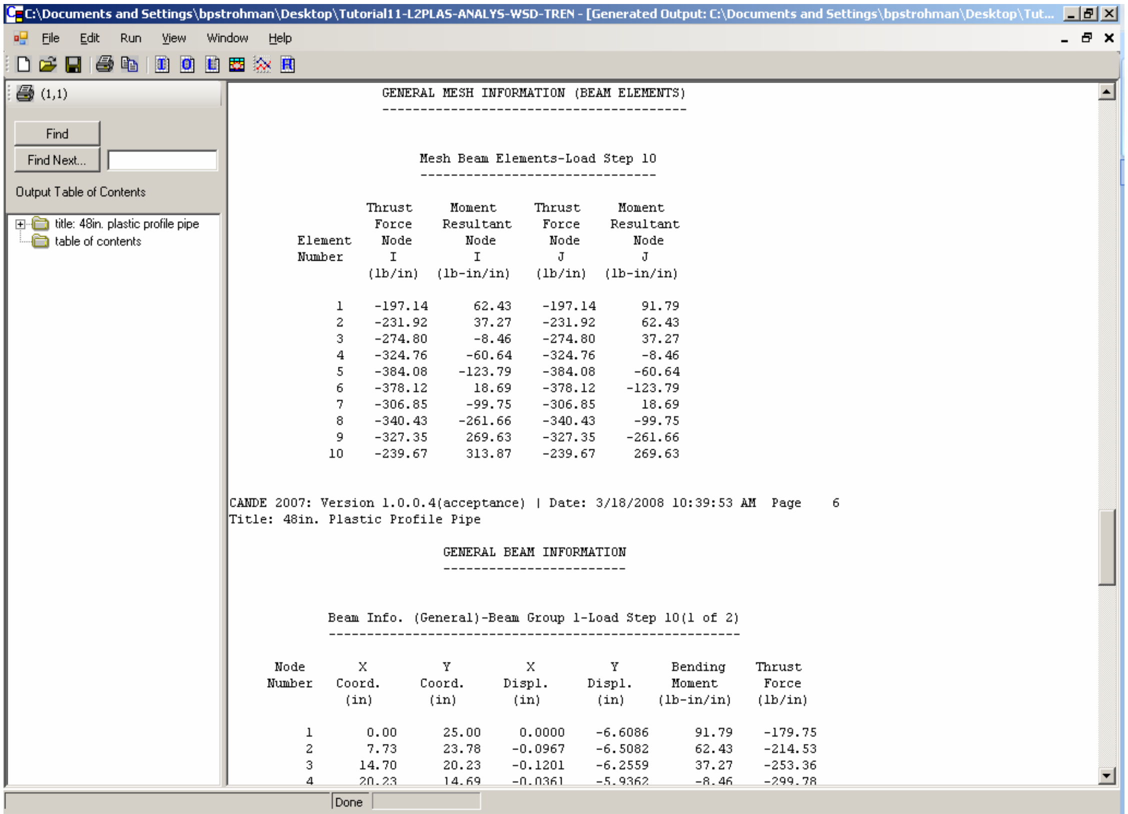


Figure 11-26 – Results Generator Input Screen – Load Step 10 Moments, Thrusts, and Deflections



GENERAL MESH INFORMATION (BEAM ELEMENTS)

Mesh Beam Elements-Load Step 10

Element Number	Thrust Force Node I (lb/in)	Moment Resultant Node I (lb-in/in)	Thrust Force Node J (lb/in)	Moment Resultant Node J (lb-in/in)
1	-197.14	62.43	-197.14	91.79
2	-231.92	37.27	-231.92	62.43
3	-274.80	-8.46	-274.80	37.27
4	-324.76	-60.64	-324.76	-8.46
5	-384.08	-123.79	-384.08	-60.64
6	-378.12	18.69	-378.12	-123.79
7	-306.85	-99.75	-306.85	18.69
8	-340.43	-261.66	-340.43	-99.75
9	-327.35	269.63	-327.35	-261.66
10	-239.67	313.87	-239.67	269.63

CANDE 2007: Version 1.0.0.4(acceptance) | Date: 3/18/2008 10:39:53 AM Page 6
Title: 48in. Plastic Profile Pipe

GENERAL BEAM INFORMATION

Beam Info. (General)-Beam Group 1-Load Step 10(1 of 2)

Node Number	X Coord. (in)	Y Coord. (in)	X Displ. (in)	Y Displ. (in)	Bending Moment (lb-in/in)	Thrust Force (lb/in)
1	0.00	25.00	0.0000	-6.6086	91.79	-179.75
2	7.73	23.78	-0.0967	-6.5082	62.43	-214.53
3	14.70	20.23	-0.1201	-6.2559	37.27	-253.36
4	20.23	14.69	-0.0361	-5.9362	-8.46	-299.78

Figure 11-27 – Results Generator – Load Step 10 Moments and Thrusts

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 12

May 2008

This page intentionally left blank

12. CANDE TEST PROBLEM 12

12.1 Problem Definition

Analyze a 120 in. x 84 in. reinforced concrete box culvert with standard ASTM steel placement with 2 ft of fill over the top of the culvert using LRFD analysis. The problem is shown schematically in Figure 12-1. The analysis will be with Level 3, using a user generated finite element mesh for an embankment installation. This problem analyzes the reinforced concrete box culvert from Tutorial Problem 7, which was performed using a Level 2 analysis.

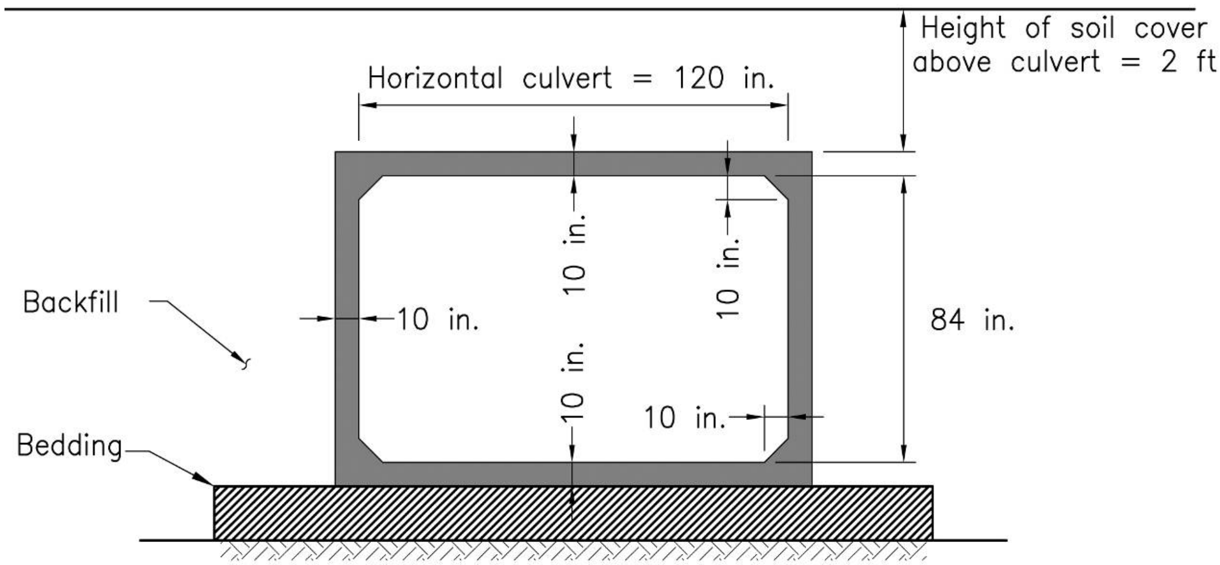


Figure 12-1 Details of Problem 12

Developing Level 3 input files for CANDE requires substantial advance planning relative to Level 2 input files. The engineer must define the installation geometry and boundary conditions in order to create a mesh. This problem uses shortcut procedures to define groups of nodes and elements. Alternatively, meshes can be defined node by node and element by element or imported into CANDE in xml format. Importing an existing mesh into CANDE is shown in Tutorial Problems 13 and 14.

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - LRFD

Solution level - FEM-user mesh (Level 3)

Number of pipe element groups (Level 3 only) - 1

Select level 3 input option - Manual input

Number of nodes - 362

Number of elements - 338

Number of boundary conditions - 100

Number of soil materials - 3

Number of interface materials - 0

Pipe material type - Concrete

Number of connected beam elements - 24

Reinforcement shape - Arbitrary (Box reinforcement can only be used in Level 2)

Soil parameters - Canned Duncan/Selig soil models except as noted:

In situ soil - Canned Isotropic-Linear Elastic, Young's modulus = 2,500 psi, Poisson's ratio = 0.25

Bedding soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Backfill soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils

Compressive strength of concrete (f'c) - 5,000 lb/in.^{2}

Shear strength equation - Boxes/3-sided < 2 ft fill (AASHTO 5.13.3.6)

Concrete strain at tension rupture - 0.0 in./in.

Note: Assuming zero tensile strength for the concrete assures the design will be for a cracked section.

Compressive strain at end of elastic range - 0.001 in./in.

Compressive strain at the initial strength limit - 0.002 in./in.

Unit weight of concrete for body weight - 150 lb/ft^3

Crack width model - Gergely-Lutz

Analysis Mode - Small deformation

Yield stress of reinforcing steel - 65,000 lb/in.^{2}

Inner surface spacing between rows of rebar - 2 in.

Outer surface spacing between rows of rebar - 2 in.

Number of inner cage layers of reinforcement - 1

Number of outer cage layers of reinforcement - 1

Type of reinforcement - Deformed wire

Nonlinear material behavior - Option 3 plus steel yielding behavior

Wall thickness and reinforcement - The following summarizes the wall thickness and reinforcement for the box culvert nodes:

Nodes	Member	Wall Thickness, in.	A_s Inner Cage 1, in.^{2}/in.	A_s Outer Cage 2, in.^{2}/in.	Cover Cage 1, in.	Cover Cage 2, in.
1	Top	10	0.0458	0.0208	1.5	1.5
2	Top	10	0.0458	0.0208	1.5	1.5
3	Top	10	0.0458	0.0208	1.5	1.5
4	Top	10	0.0458	0.0208	1.5	1.5
5	Top	10	0.0458	0.0208	1.5	1.5
6	Top	10	0.0458	0.0208	1.5	1.5
7	Top	15	0.0458	0.0208	1.5	1.5
8	Top	20	0.0458	0.0208	1.5	1.5
9	Side	15	0.02	0.0208	1.5	1.5
10	Side	10	0.02	0.0208	1.5	1.5
11	Side	10	0.02	0.0208	1.5	1.5
12	Side	10	0.02	0.0208	1.5	1.5
13	Side	10	0.02	0.0208	1.5	1.5
14	Side	10	0.02	0.0208	1.5	1.5
15	Side	10	0.02	0.0208	1.5	1.5
16	Side	10	0.02	0.0208	1.5	1.5
17	Side	15	0.02	0.0208	1.5	1.5
18	Bottom	20	0.0242	0.0208	1.5	1.5
19	Bottom	15	0.0242	0.0208	1.5	1.5
20	Bottom	10	0.0242	0.0208	1.5	1.5
21	Bottom	10	0.0242	0.0208	1.5	1.5
22	Bottom	10	0.0242	0.0208	1.5	1.5
23	Bottom	10	0.0242	0.0208	1.5	1.5

24	Bottom	10	0.0242	0.0208	1.5	1.5
25	Bottom	10	0.0242	0.0208	1.5	1.5

Number of construction steps - 6

LRFD load factor - 1.35

Load modifier - 1.05 (non-redundant for earth load)

12.2 Creating the CANDE Input Document

Figures 12-2 through 12-5 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure. After clicking “Next” on Screen 4, CANDE will display a screen indicating that an input path is initiated - click “Finish.” The user is prompted for a filename and directory.

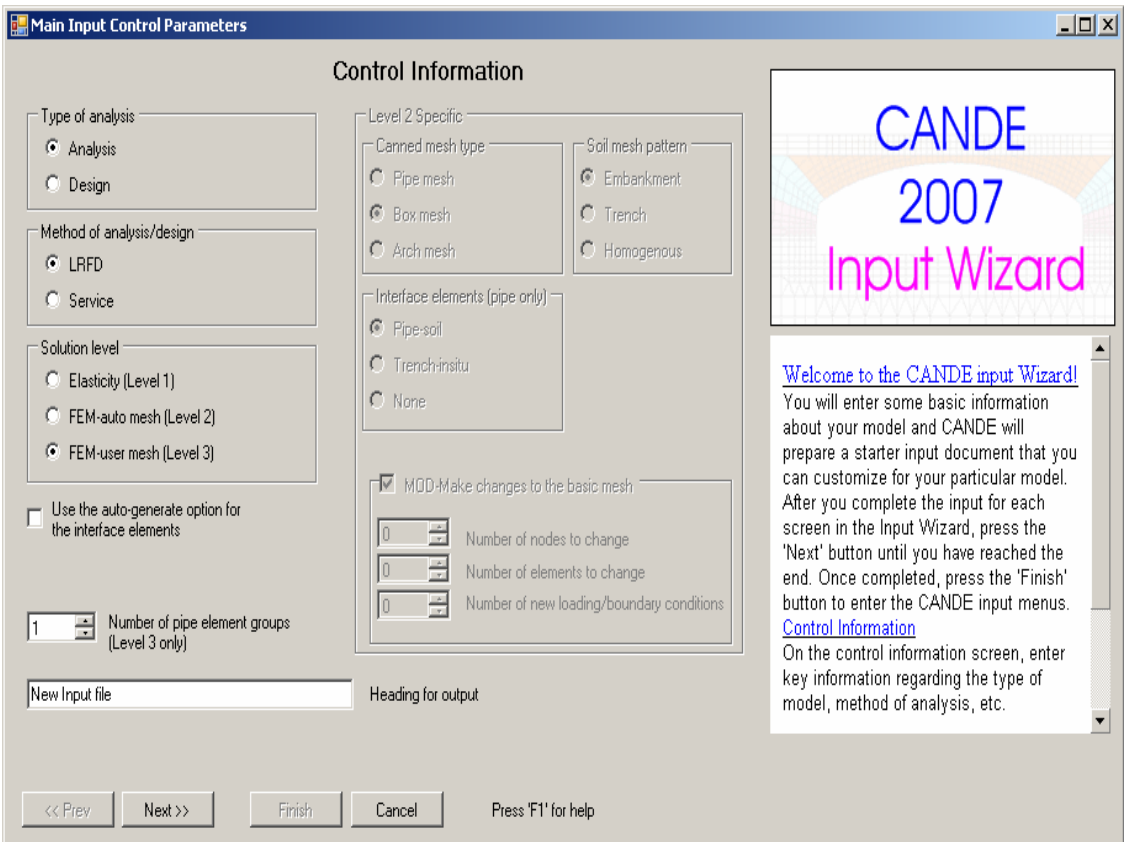


Figure 12-2 – Input Wizard, Screen 1

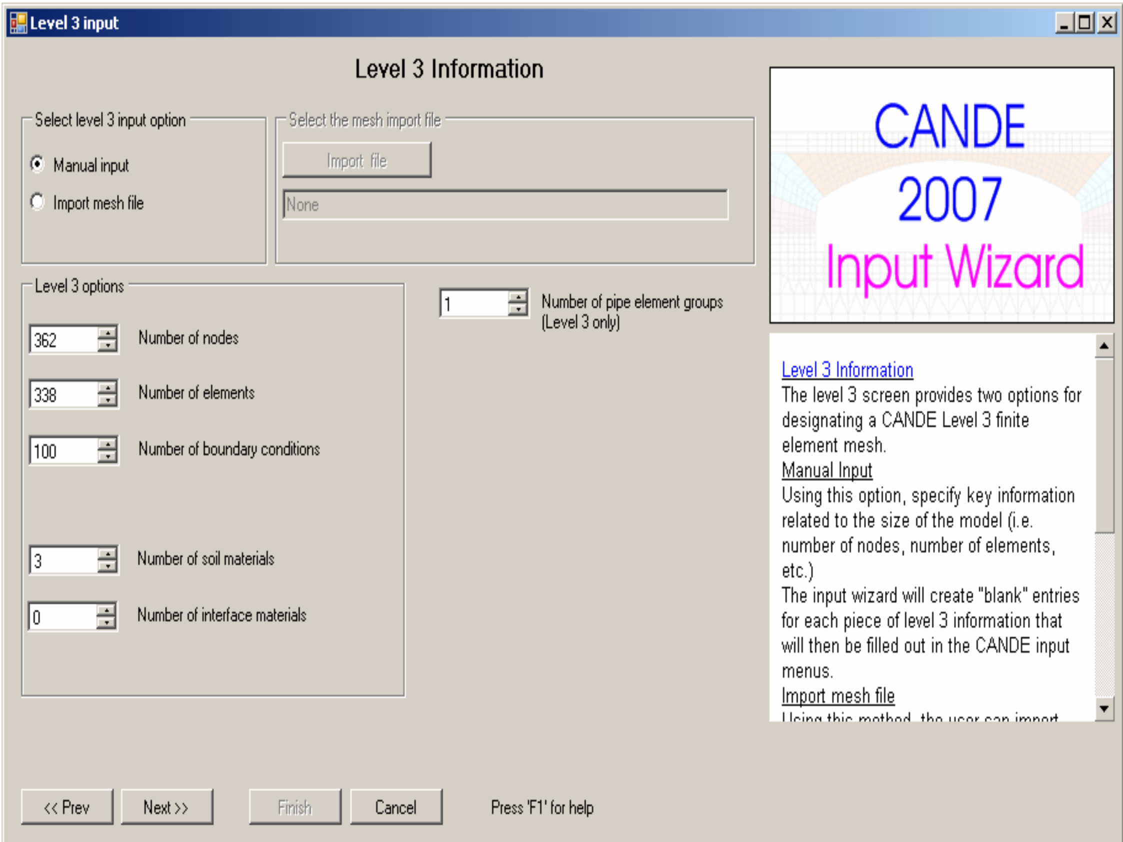


Figure 12-3 – Input Wizard, Screen 2

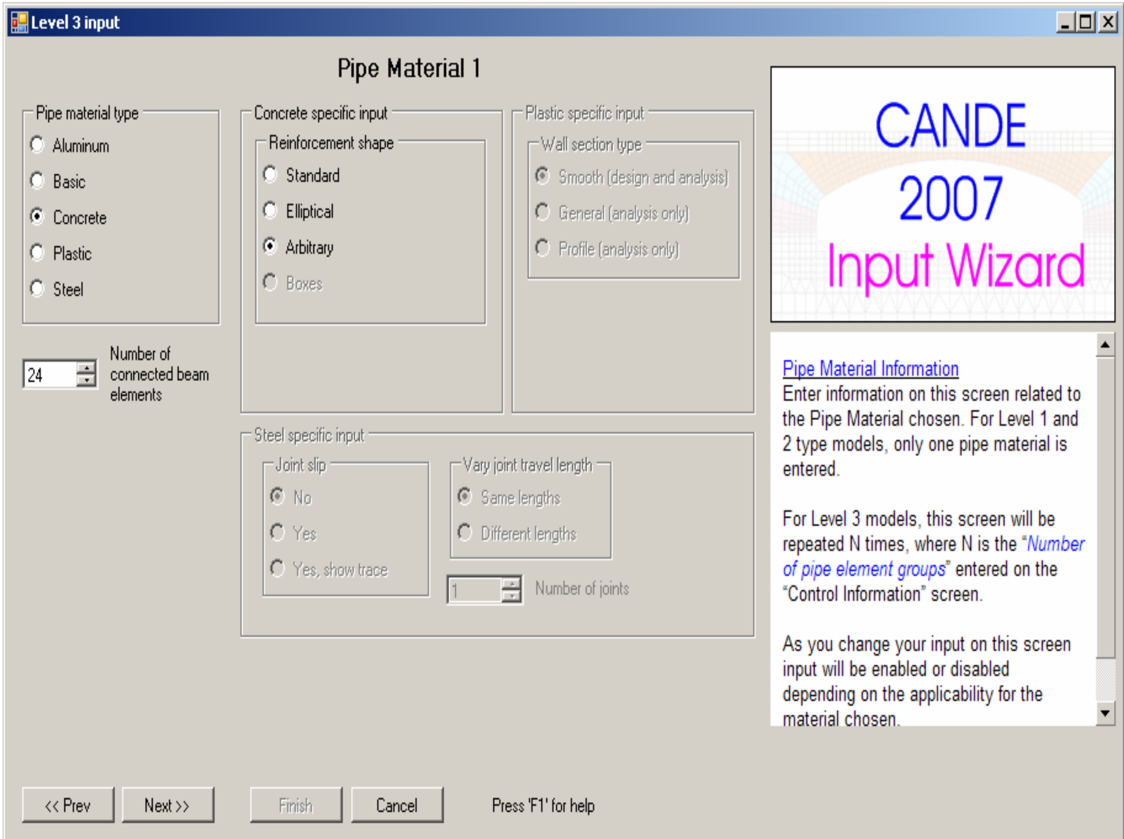


Figure 12-4 – Input Wizard, Screen 3

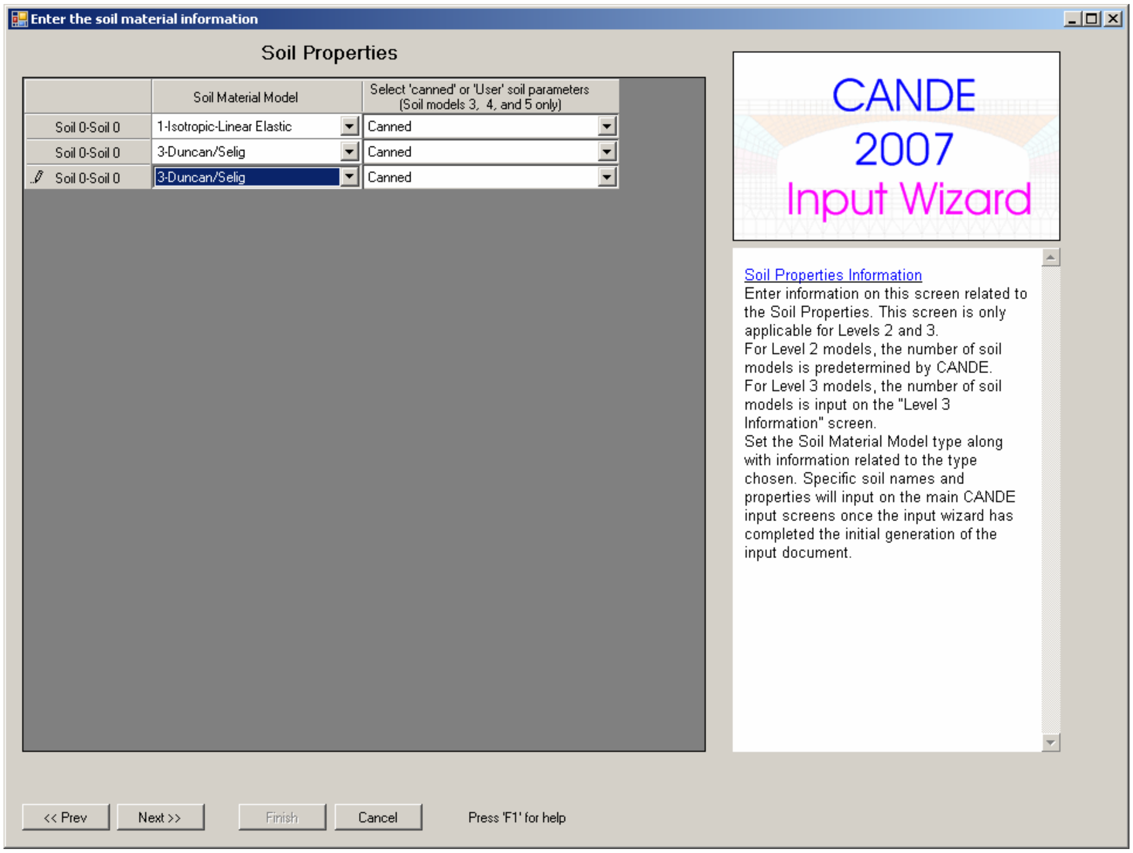


Figure 12-5 – Input Wizard, Screen 4

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 12-6. Enter an appropriate heading for output and click “Accept Input.”

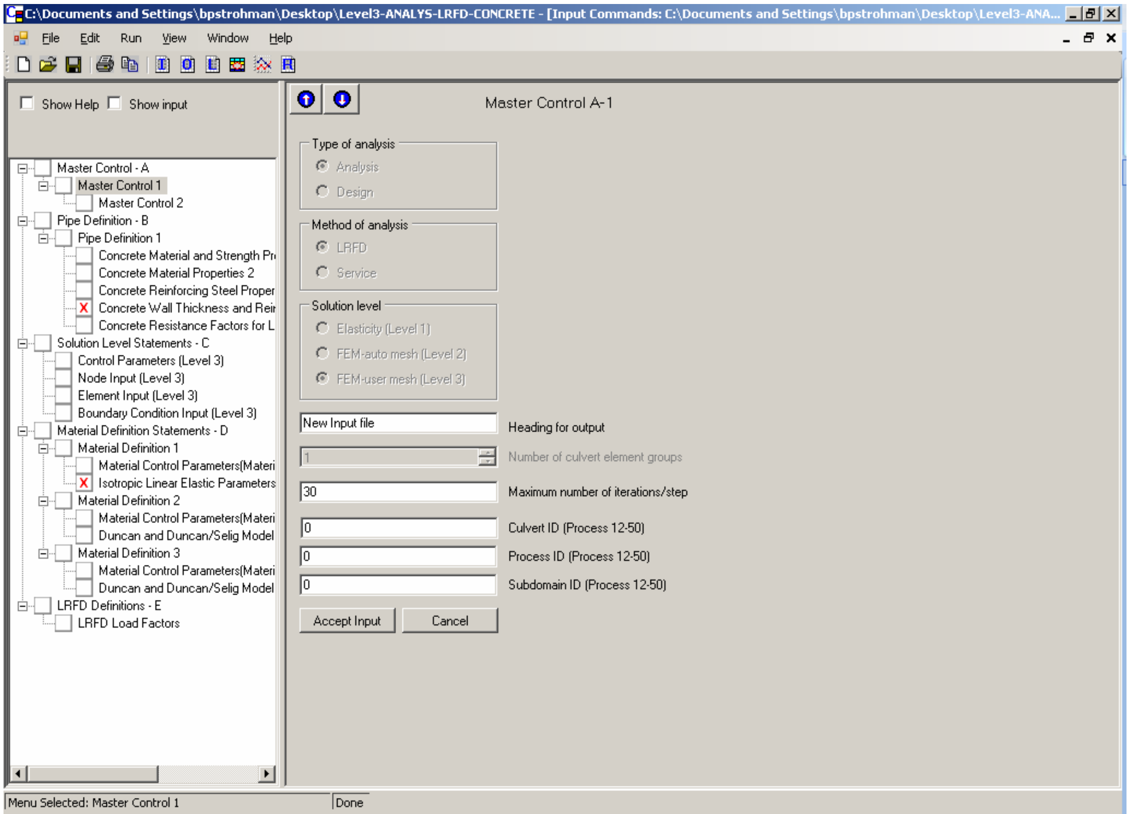


Figure 12-6 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 12-6 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 12-7 through 12-17 show the completed input for the screens requiring data for the tutorial, except that only one material definition is shown. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

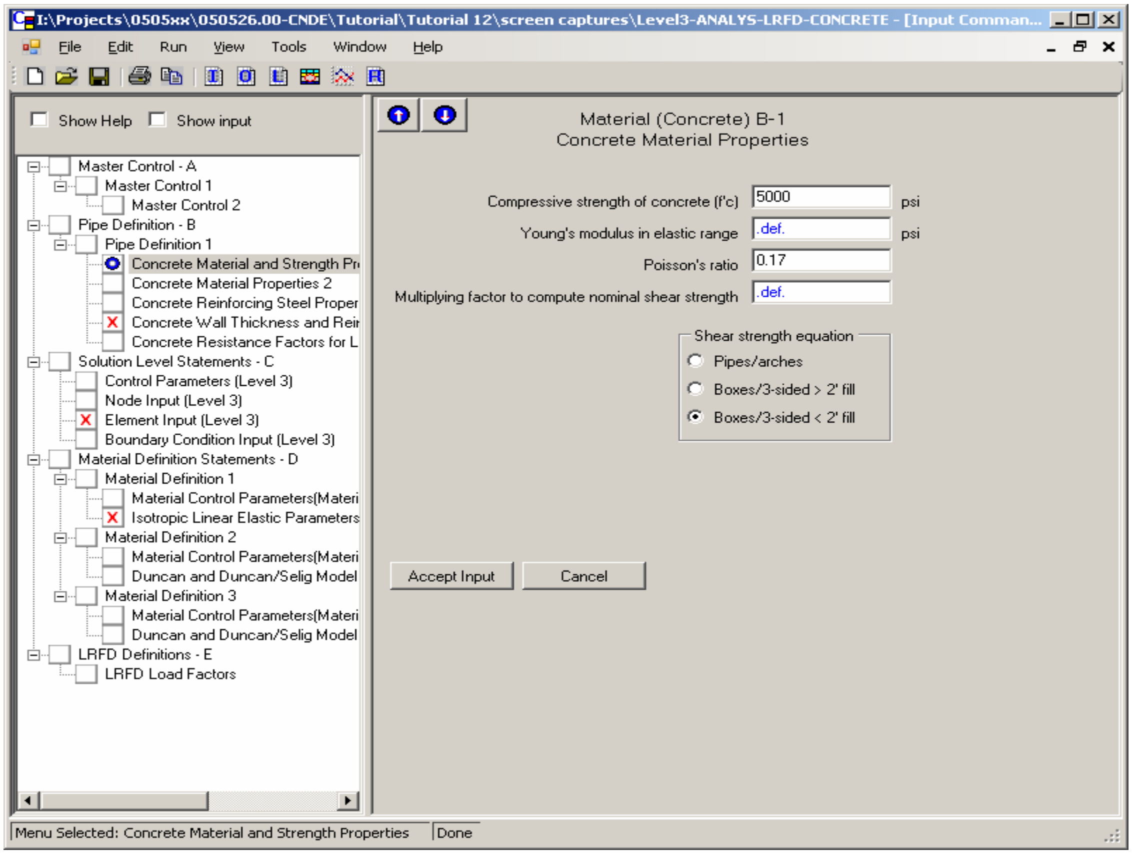


Figure 12-7 – Input Screen B-1

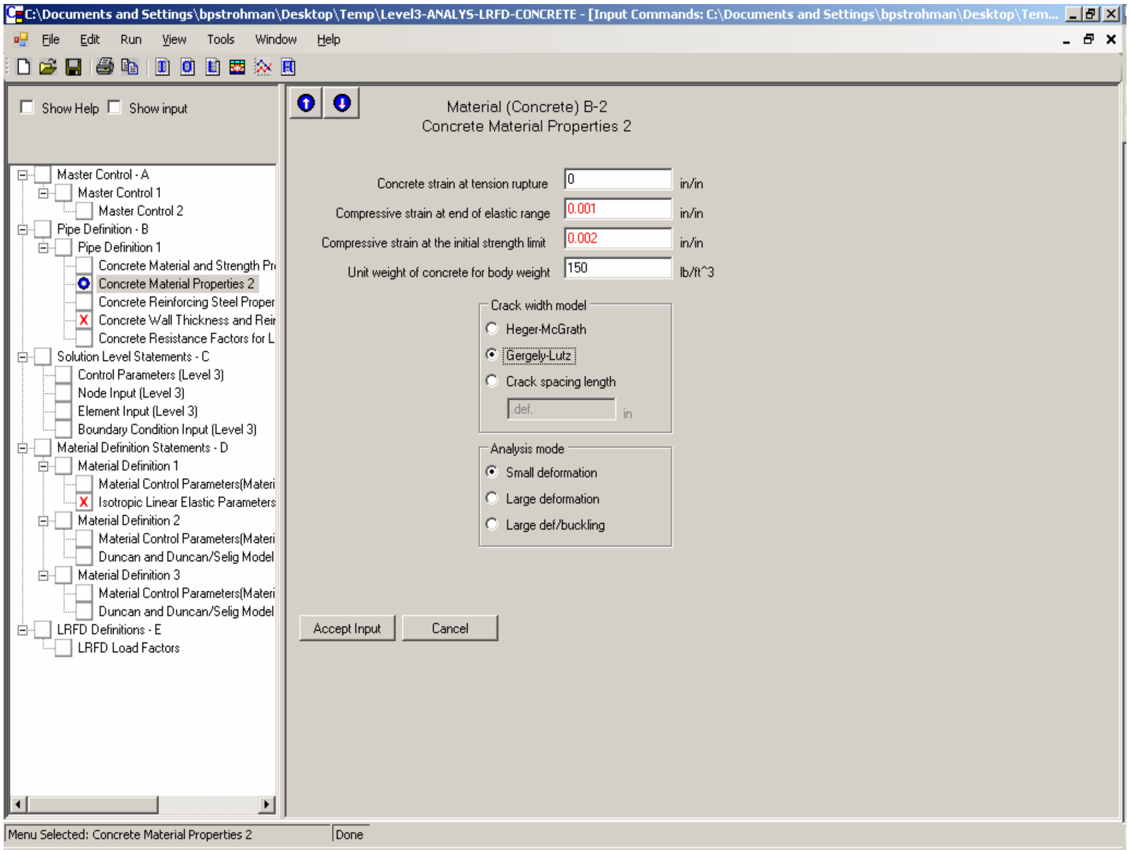


Figure 12-8 – Input Screen B-2

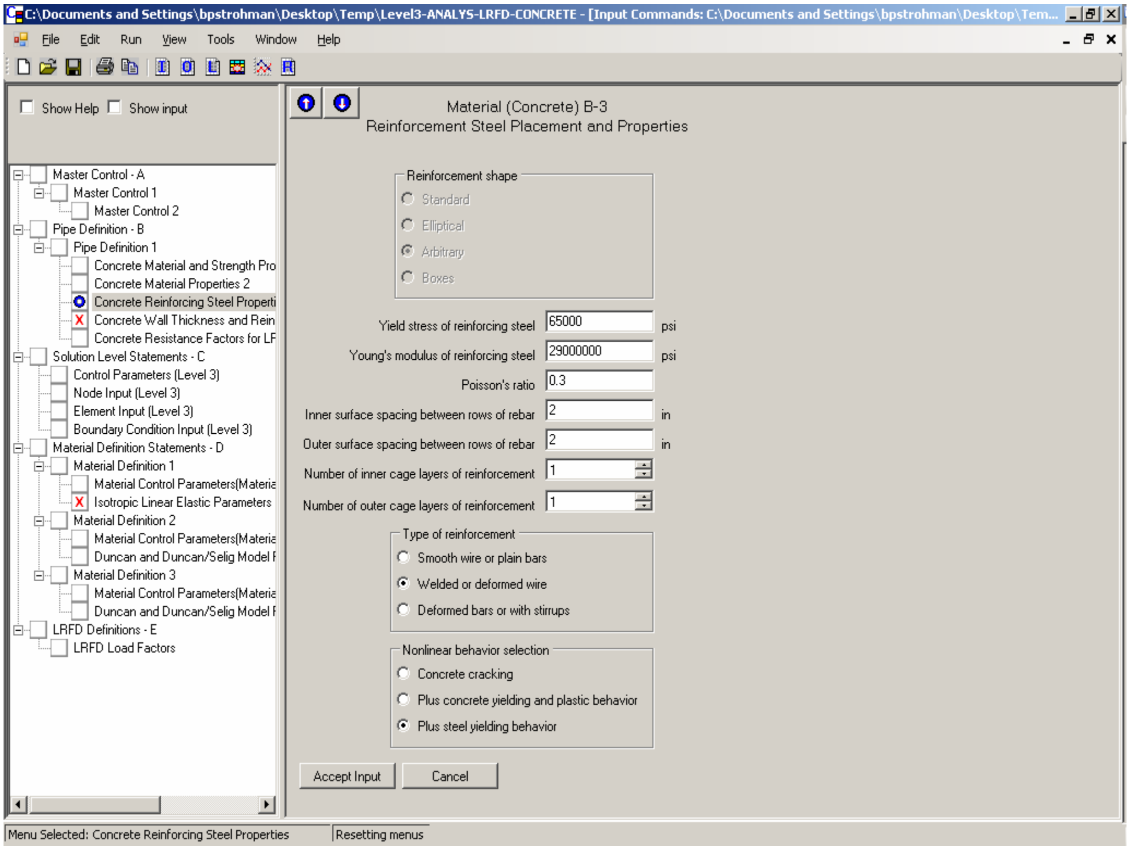


Figure 12-9 – Input Screen B-3

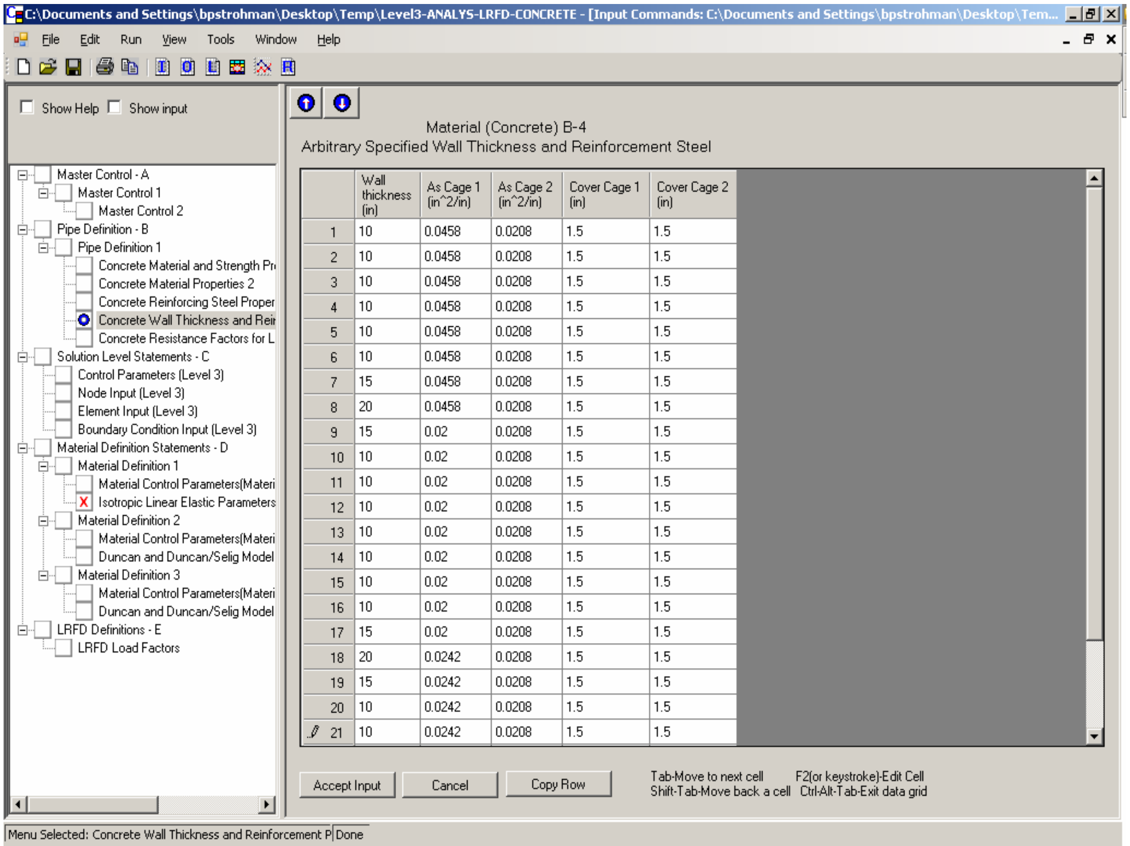


Figure 12-10 – Input Screen B-4

Enter the material information in the grid from the table provided in the 'Problem Definition'. To expedite the entry, fill in the first row, use the 'Copy Row' button to rows 2-25, and modify the wall thicknesses and inner/outer cage areas that are different.

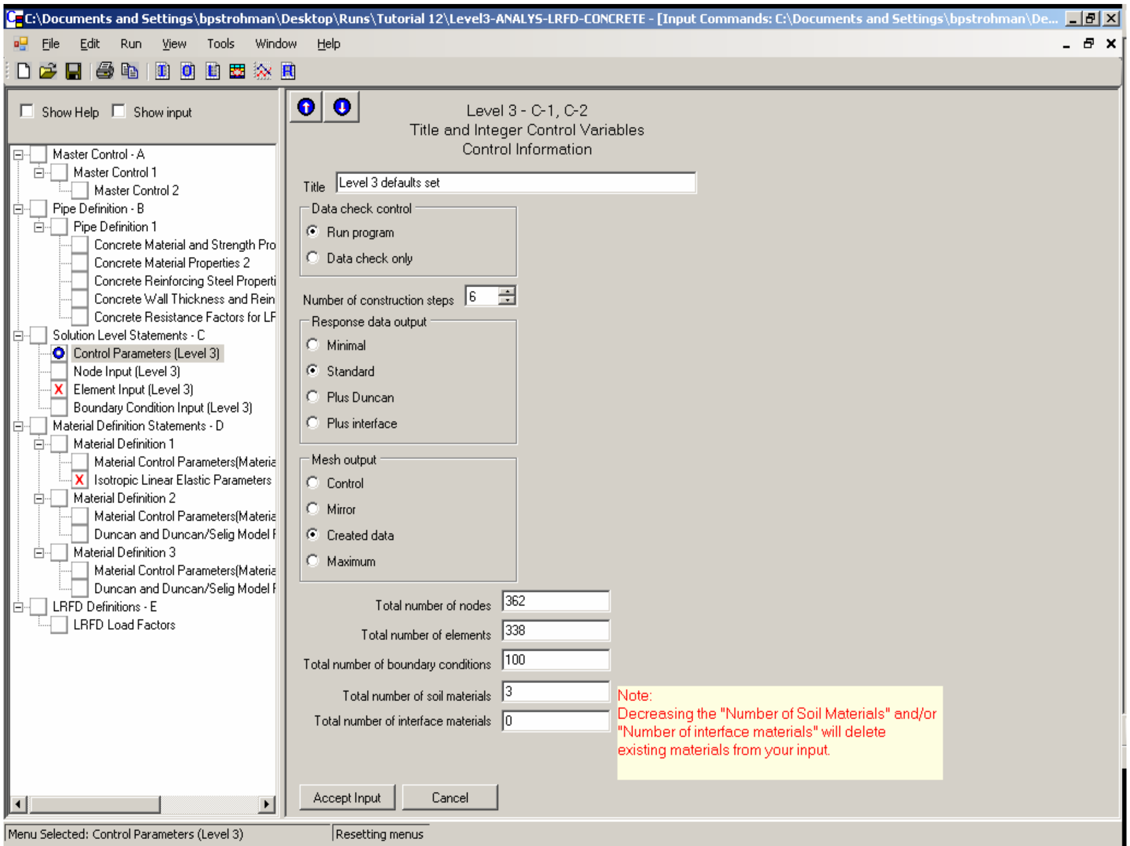


Figure 12-11 – Input Screen C-1, C-2

Nodal coordinates, elements, and boundary conditions were defined using the methods described in Chapter 5, C-3, C-4, and C-5 of the *User Manual.* Figure 12-12, 12-13, and 12-14 show the top portion of the C-3, C-4, and C-5 input screens using this method. A complete input file of the problem is displayed at the end of this section of the tutorial.

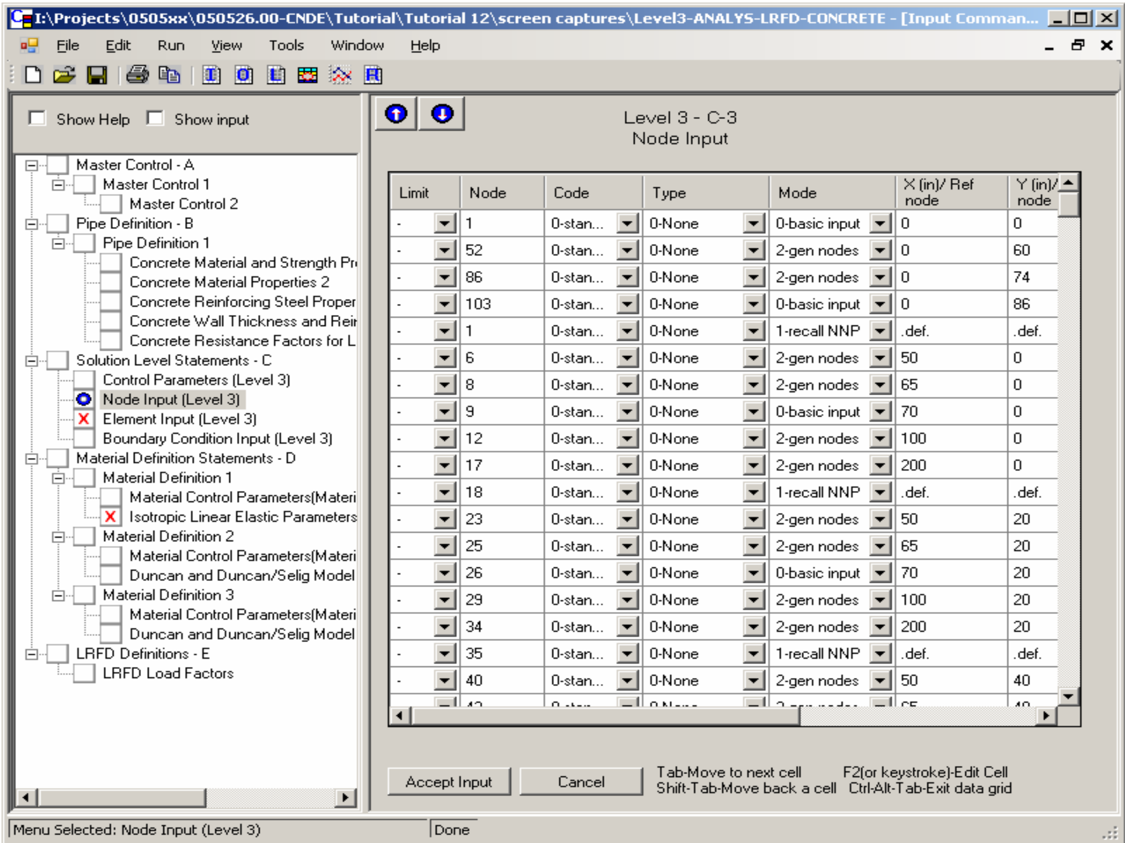


Figure 12-12 – Input Screen C-3

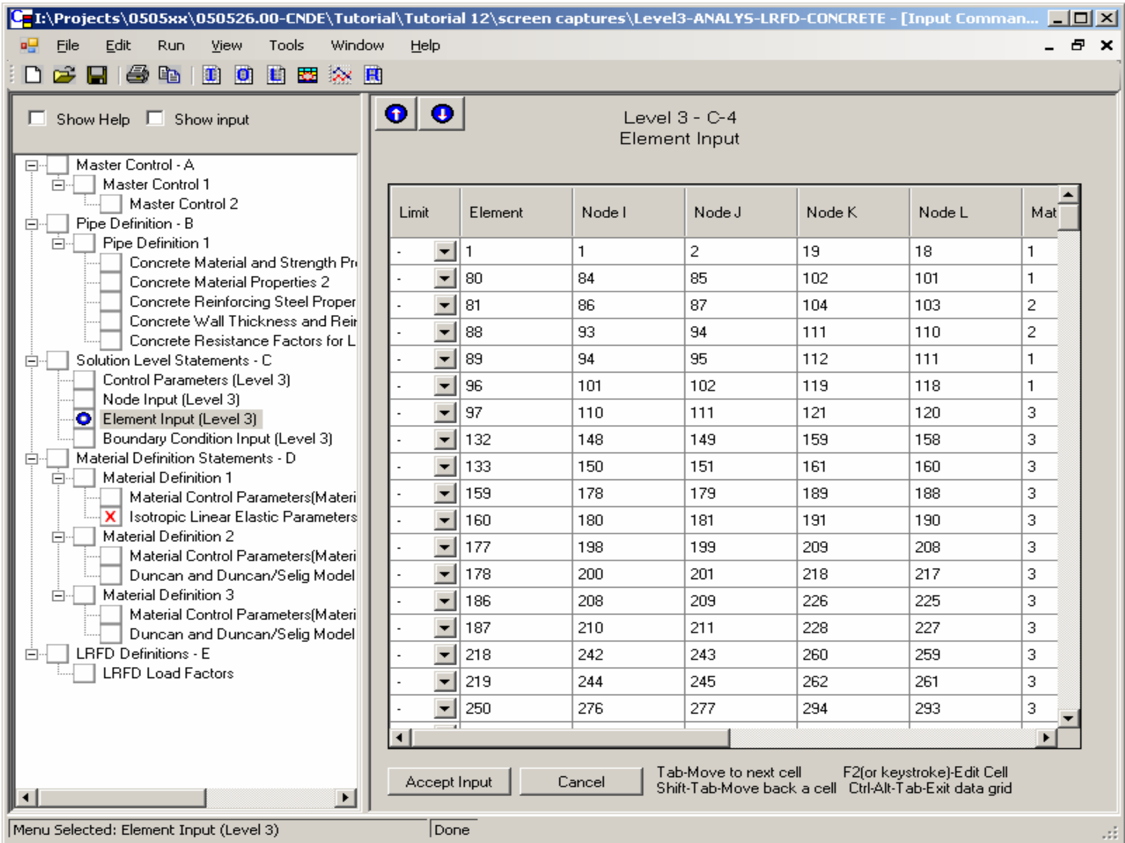


Figure 12-13 – Input Screen C-4

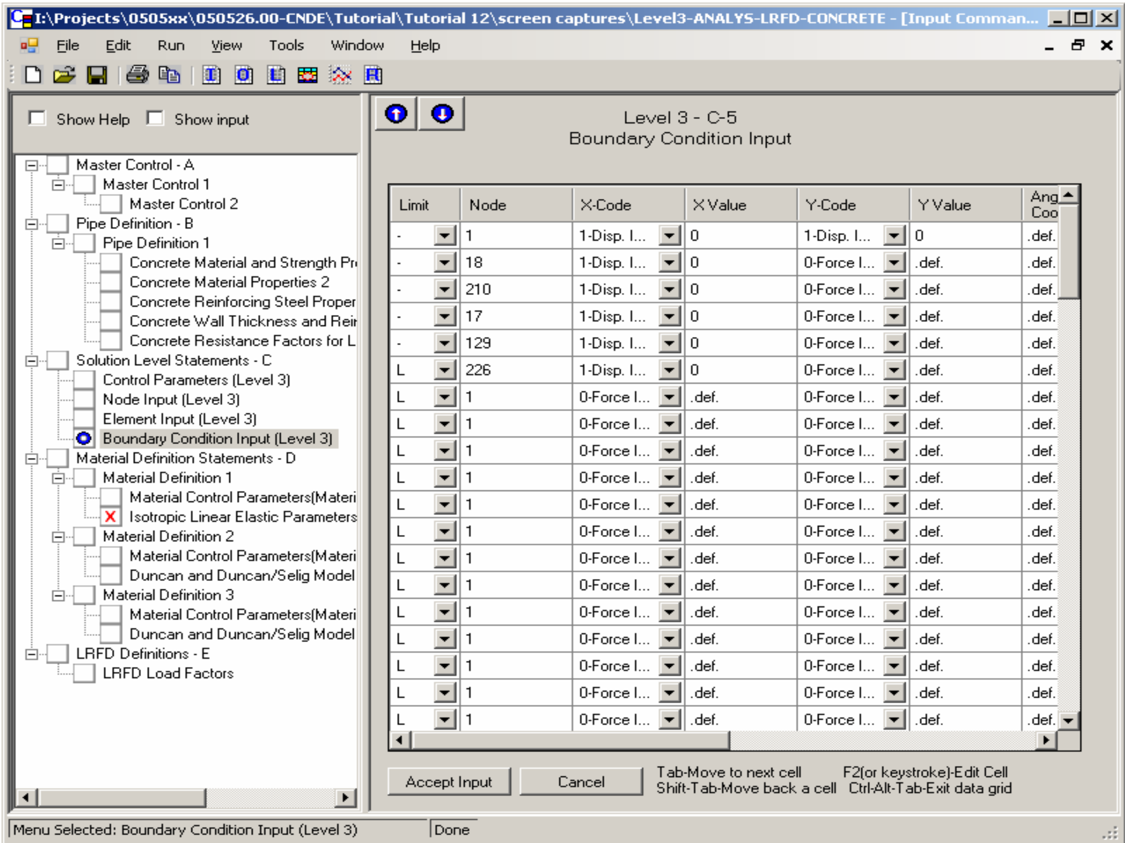


Figure 12-14 – Input Screen C-5

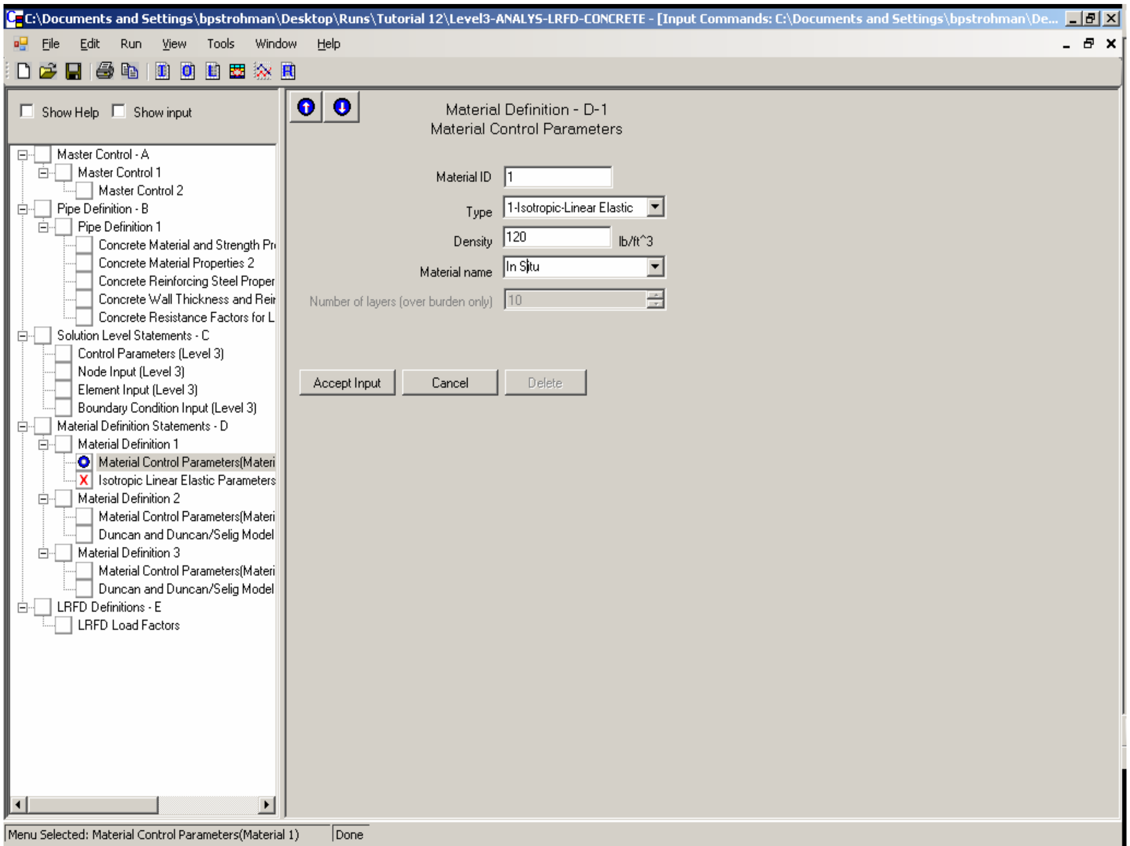


Figure 12-15 – Input Screen D-1 for Material 1
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

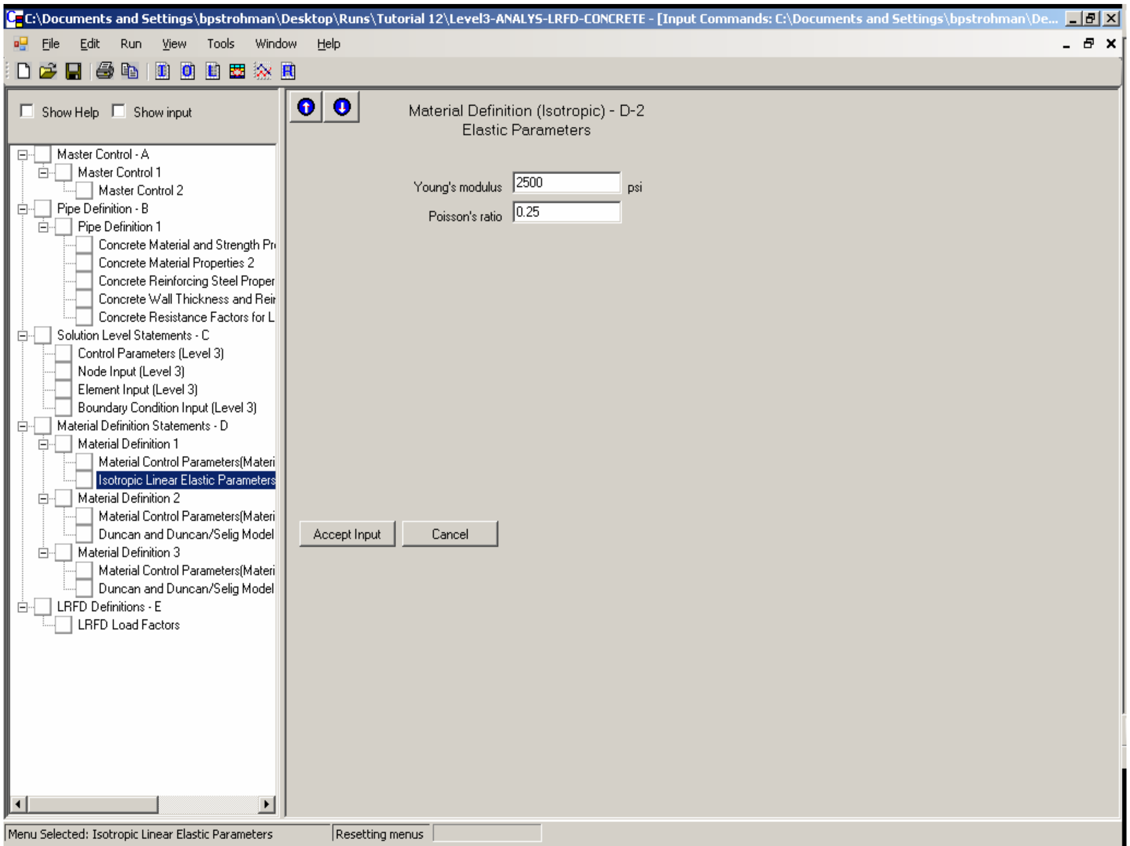


Figure 12-16 – Input Screen D-2 for Material 1
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

Under LRFD load factor in Figure 12-17 enter the combined value of the load factor and load modifier, i.e. – 1.35 * 1.05 = 1.42

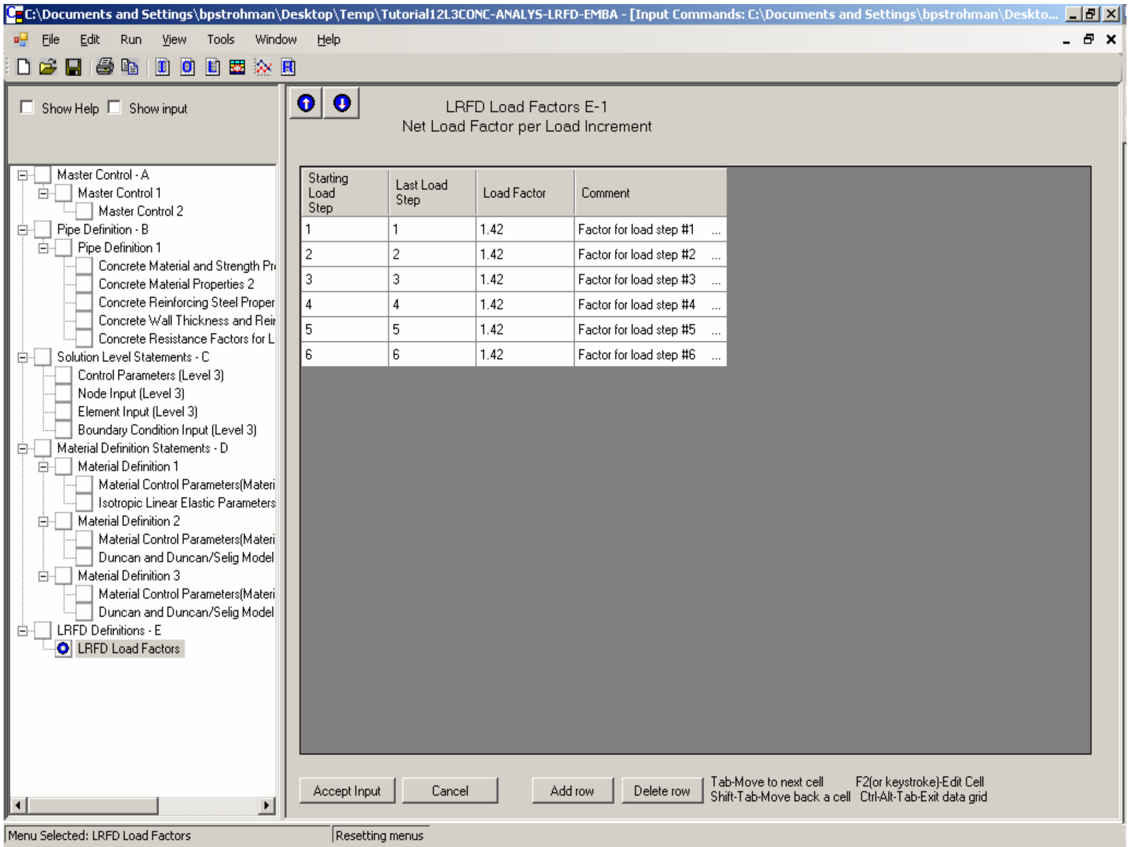


Figure 12-17 – Input Screen E-1

CANDE-2007 Input File:

```
                  A-1!!ANALYS   3 1  1120 x 84 Rein. Conc. Box                             30
               A-2.L3!!CONCRETE     24
         B-1.Concrete!!      5000              0.17             3
         B-2.Concrete!!         0     0.001     0.002       150        -1    0
         B-3.Concrete!!ARBIT          65000  29000000       0.3         2         2    1    1
2    3
     B-4.Concrete.Case1_2!!        10    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        15    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        20    0.0458    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        15      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        15      0.02    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        20    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        15    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0242    0.0208       1.5       1.5
     B-4.Concrete.Case1_2!!        10    0.0242    0.0208       1.5       1.5
         B-5.Concrete!!       0.9      0.75       0.9       0.9      0.01
               C-1.L3!!PREP Level 3 defaults set
               C-2.L3!!    6    3    0    3    1  362  338  100    3    0    0
               C-3.L3!!    1  000         0         0
               C-3.L3!!   52  002         0        60   17
               C-3.L3!!   86  002         0        74   17
               C-3.L3!!  103  000         0        86
               C-3.L3!!    1  001
               C-3.L3!!    6  002        50         0
               C-3.L3!!    8  002        65         0
               C-3.L3!!    9  000        70         0
               C-3.L3!!   12  002       100         0
               C-3.L3!!   17  002       200         0
               C-3.L3!!   18  001
               C-3.L3!!   23  002        50        20
               C-3.L3!!   25  002        65        20
               C-3.L3!!   26  000        70        20
               C-3.L3!!   29  002       100        20
               C-3.L3!!   34  002       200        20
               C-3.L3!!   35  001
               C-3.L3!!   40  002        50        40
               C-3.L3!!   42  002        65        40
               C-3.L3!!   43  000        70        40
               C-3.L3!!   46  002       100        40
               C-3.L3!!   51  002       200        40
               C-3.L3!!   52  001
               C-3.L3!!   57  002        50        60
               C-3.L3!!   59  002        65        60
               C-3.L3!!   60  000        70        60
               C-3.L3!!   63  002       100        60
               C-3.L3!!   68  002       200        60
               C-3.L3!!   69  001
               C-3.L3!!   74  002        50        67
               C-3.L3!!   76  002        65        67
               C-3.L3!!   77  000        70        67
               C-3.L3!!   80  002       100        67
               C-3.L3!!   85  002       200        67
               C-3.L3!!   86  001
```

C-3.L3!!	91	002	50	74	
C-3.L3!!	93	002	65	74	
C-3.L3!!	94	000	70	74	
C-3.L3!!	97	002	100	74	
C-3.L3!!	102	002	200	74	
C-3.L3!!	103	001			
C-3.L3!!	108	002	50	86	
C-3.L3!!	110	002	65	86	
C-3.L3!!	111	000	70	86	
C-3.L3!!	114	002	100	86	
C-3.L3!!	119	002	200	86	
C-3.L3!!	110	001			
C-3.L3!!	130	002	65	101	10
C-3.L3!!	190	002	65	165	10
C-3.L3!!	200	000	65	172.5	
C-3.L3!!	120	001			
C-3.L3!!	121	000	70	93.5	
C-3.L3!!	124	002	100	93.5	
C-3.L3!!	129	002	200	93.5	
C-3.L3!!	130	001			
C-3.L3!!	131	000	70	101	
C-3.L3!!	134	002	100	101	
C-3.L3!!	139	002	200	101	
C-3.L3!!	140	001			
C-3.L3!!	141	000	70	111.67	
C-3.L3!!	144	002	100	111.67	
C-3.L3!!	149	002	200	111.67	
C-3.L3!!	150	001			
C-3.L3!!	151	000	70	122.33	
C-3.L3!!	154	002	100	122.33	
C-3.L3!!	159	002	200	122.33	
C-3.L3!!	160	001			
C-3.L3!!	161	000	70	133	
C-3.L3!!	164	002	100	133	
C-3.L3!!	169	002	200	133	
C-3.L3!!	170	001			
C-3.L3!!	171	000	70	143.67	
C-3.L3!!	174	002	100	143.67	
C-3.L3!!	179	002	200	143.67	
C-3.L3!!	180	001			
C-3.L3!!	181	000	70	154.33	
C-3.L3!!	184	002	100	154.33	
C-3.L3!!	189	002	200	154.33	
C-3.L3!!	190	001			
C-3.L3!!	191	000	70	165	
C-3.L3!!	194	002	100	165	
C-3.L3!!	199	002	200	165	
C-3.L3!!	200	001			
C-3.L3!!	201	000	70	172.5	
C-3.L3!!	204	002	100	172.5	
C-3.L3!!	209	002	200	172.5	
C-3.L3!!	210	000	0	180	
C-3.L3!!	244	002	0	192	17
C-3.L3!!	261	000	0	198	
C-3.L3!!	278	000	0	204	
C-3.L3!!	329	002	0	288	17
C-3.L3!!	346	000	0	300	
C-3.L3!!	210	001			
C-3.L3!!	215	002	50	180	
C-3.L3!!	217	002	65	180	
C-3.L3!!	218	000	70	180	
C-3.L3!!	221	002	100	180	
C-3.L3!!	226	002	200	180	
C-3.L3!!	227	001			
C-3.L3!!	232	002	50	186	
C-3.L3!!	234	002	65	186	
C-3.L3!!	235	000	70	186	
C-3.L3!!	238	002	100	186	
C-3.L3!!	243	002	200	186	
C-3.L3!!	244	001			
C-3.L3!!	249	002	50	192	

```
       C-3.L3!!  251  002         65        192
       C-3.L3!!  252  000         70        192
       C-3.L3!!  255  002        100        192
       C-3.L3!!  260  002        200        192
       C-3.L3!!  261  001
       C-3.L3!!  266  002         50        198
       C-3.L3!!  268  002         65        198
       C-3.L3!!  269  000         70        198
       C-3.L3!!  272  002        100        198
       C-3.L3!!  277  002        200        198
       C-3.L3!!  278  001
       C-3.L3!!  283  002         50        204
       C-3.L3!!  285  002         65        204
       C-3.L3!!  286  000         70        204
       C-3.L3!!  289  002        100        204
       C-3.L3!!L 294  002        200        204
       C-4.L3!!    1    1    2   19   18    1    1
       C-4.L3!!   80   84   85  102  101    1    1         1    5   17
       C-4.L3!!   81   86   87  104  103    2    1
       C-4.L3!!   88   93   94  111  110    2    1         1    1    9
       C-4.L3!!   89   94   95  112  111    1    1
       C-4.L3!!   96  101  102  119  118    1    1         1    1    9
       C-4.L3!!   97  110  111  121  120    3    2
       C-4.L3!!  132  148  149  159  158    3    2         1    4   10
       C-4.L3!!  133  150  151  161  160    3    3
       C-4.L3!!  159  178  179  189  188    3    3         1    3   10
       C-4.L3!!  160  180  181  191  190    3    4
       C-4.L3!!  177  198  199  209  208    3    4         1    2   10
       C-4.L3!!  178  200  201  218  217    3    4
       C-4.L3!!  186  208  209  226  225    3    4         1    1   10
       C-4.L3!!  187  210  211  228  227    3    5
       C-4.L3!!  218  242  243  260  259    3    5         1    2   17
       C-4.L3!!  219  244  245  262  261    3    6
       C-4.L3!!  250  276  277  294  293    3    6         1    2   17
       C-4.L3!!  251  211  210                1    1
       C-4.L3!!  252  212  211                1    1
       C-4.L3!!  253  213  212                1    1
       C-4.L3!!  254  214  213                1    1
       C-4.L3!!  255  215  214                1    1
       C-4.L3!!  256  216  215                1    1
       C-4.L3!!  257  217  216                1    1
       C-4.L3!!  258  200  217                1    1
       C-4.L3!!  259  190  200                1    1
       C-4.L3!!  260  180  190                1    1
       C-4.L3!!  261  170  180                1    1
       C-4.L3!!  262  160  170                1    1
       C-4.L3!!  263  150  160                1    1
       C-4.L3!!  264  140  150                1    1
       C-4.L3!!  265  130  140                1    1
       C-4.L3!!  266  120  130                1    1
       C-4.L3!!  267  110  120                1    1
       C-4.L3!!  268  109  110                1    1
       C-4.L3!!  269  108  109                1    1
       C-4.L3!!  270  107  108                1    1
       C-4.L3!!  271  106  107                1    1
       C-4.L3!!  272  105  106                1    1
       C-4.L3!!  273  104  105                1    1
       C-4.L3!!L 274  103  104                1    1
       C-5.L3!!    1    1         0    1         0              1   17    1
       C-5.L3!!   18    1         0    0                        1  103   17
       C-5.L3!!  210    1         0    0                        1  278   17
       C-5.L3!!   17    1         0    0                        1  119   17
       C-5.L3!!  129    1         0    0                        1  209   10
       C-5.L3!!L 226    1         0    0                        1  294   17
          D-1!!    1    1       120In Situ             10
D-2.Isotropic!!       2500      0.25
          D-1!!    2    3       120SW95                10
   D-2.Duncan!!    0        0.5    1
          D-1!!L   3    3       120SW95                10
   D-2.Duncan!!    0        0.5    1
          E-1!!    1    1      1.42Factor for load step #1
```

```
                    E-1!!    2    2      1.42Factor for load step #2
                    E-1!!    3    3      1.42Factor for load step #3
                    E-1!!    4    4      1.42Factor for load step #4
                    E-1!!    5    5      1.42Factor for load step #5
                    E-1!!    6    6      1.42Factor for load step #6
STOP
```

When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

12.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the “View” menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The “Plotting Parameters” button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click “View” and then “Mesh Plot” to open the mesh plot. Open the plotting parameters menu and click the check box “Show constr. increment numbers.” Click “OK,” set Load Increment to 6 to show the entire mesh and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 12-18.

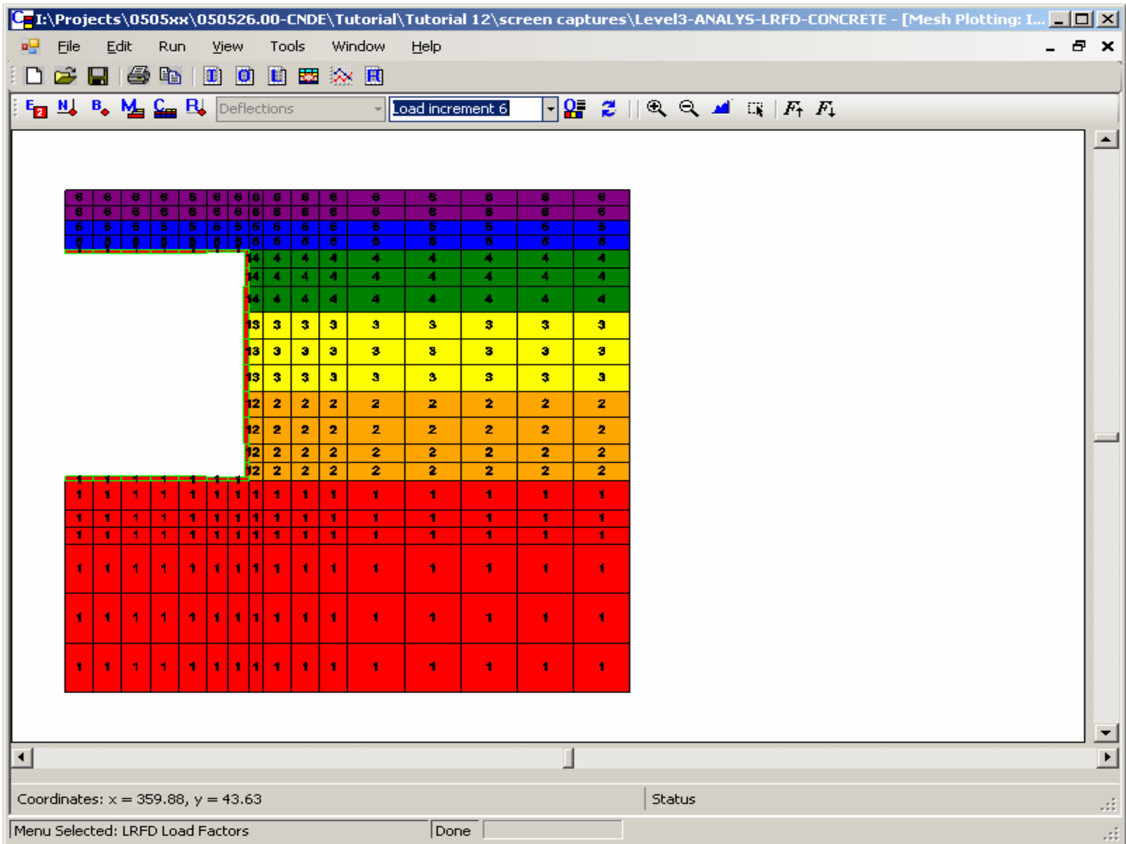


Figure 12-18 – Mesh Plot for Load Steps 1 to 6

To view deflections, open the plotting parameters window and set the deflection magnification factor to 10, click the “Ignore deflections/by load incr.” check box and set the Load Increment to 1 (see Figure 12-19).

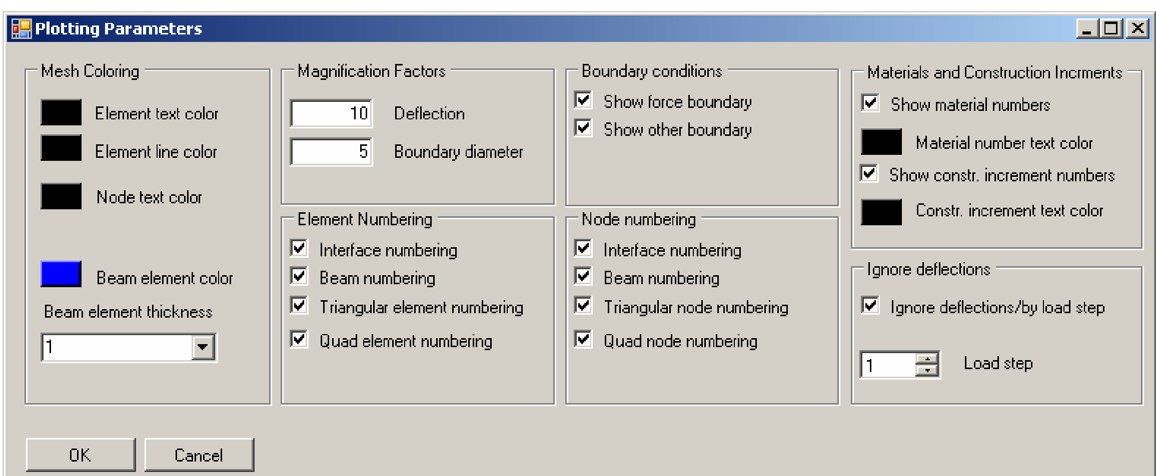


Figure 12-19 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Deflections. Note that if the Load Increment is set to 1, the deflections due to Load Increment 1 are shown, but when the Load Increment is set to 2, the Increment 1 deflections are ignored. Set the increment to 6 and the screen should look like Figure 12-20. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

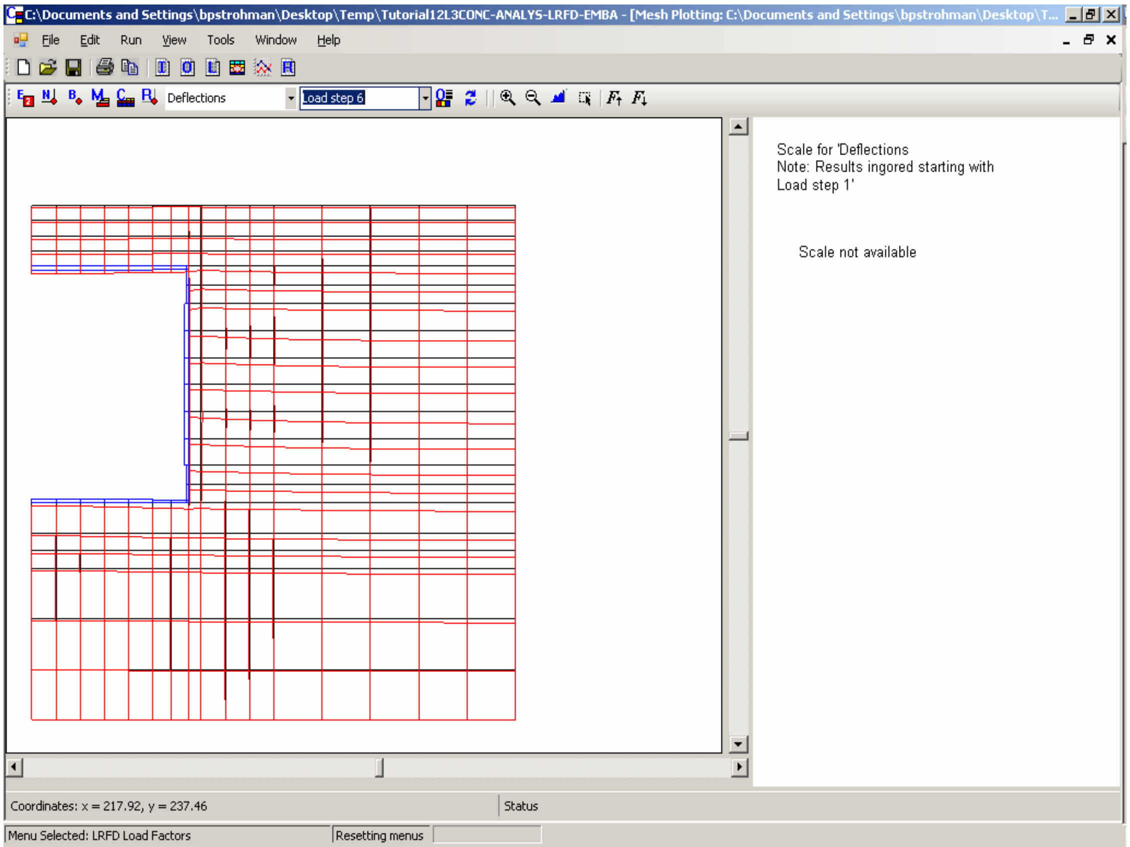


Figure 12-20 – Deflections, Excluding Those Due to Self Weight of In Situ Soil

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 12-21 compares the bending moments after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 3, 4, and 6, then “OK”. The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Bending moment” and the screen should appear as Figure 12-21.

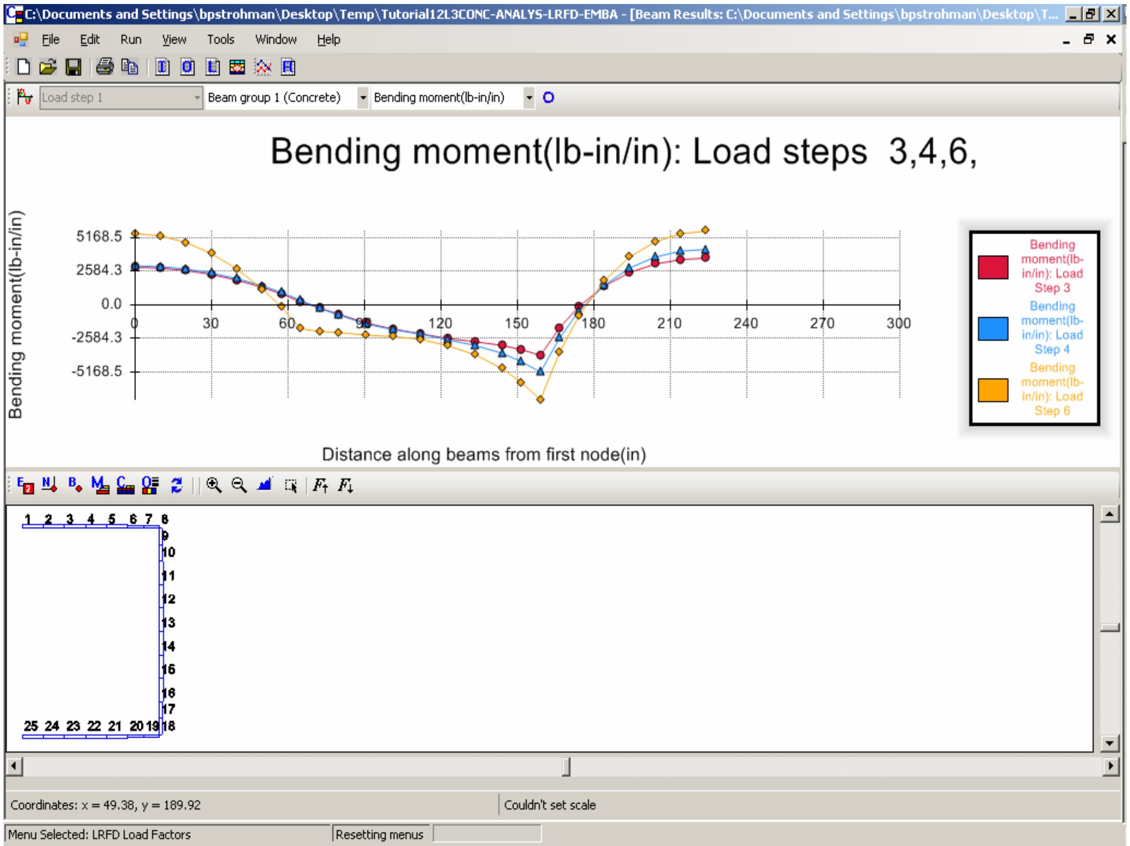


Figure 12-21 – Bending Moment for Load Steps 3, 4, and 6

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 12-22) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 12-22 shows the final assessment printed at the end of the file.

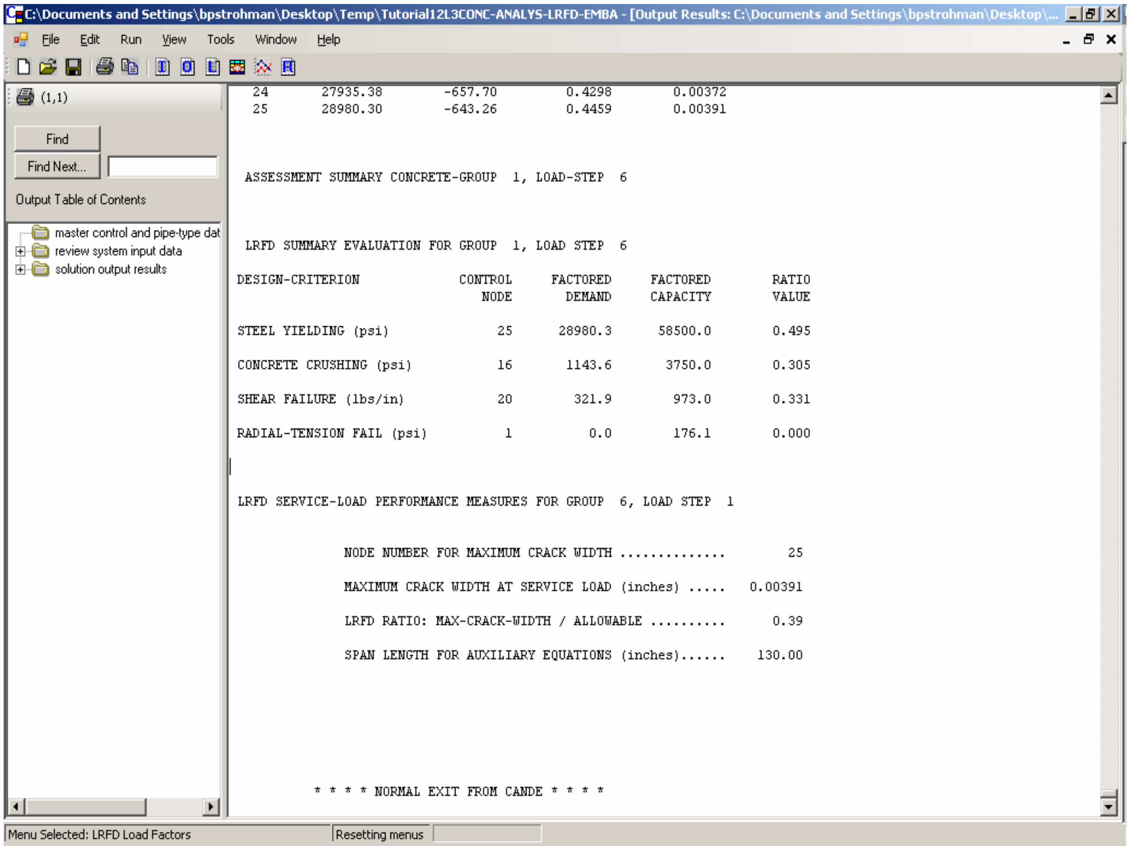


Figure 12-22 – Design Assessment Summary – Load Step 6

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 12-23 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 12-24. The three tabs shown in Figures 12-23 and 12-24 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

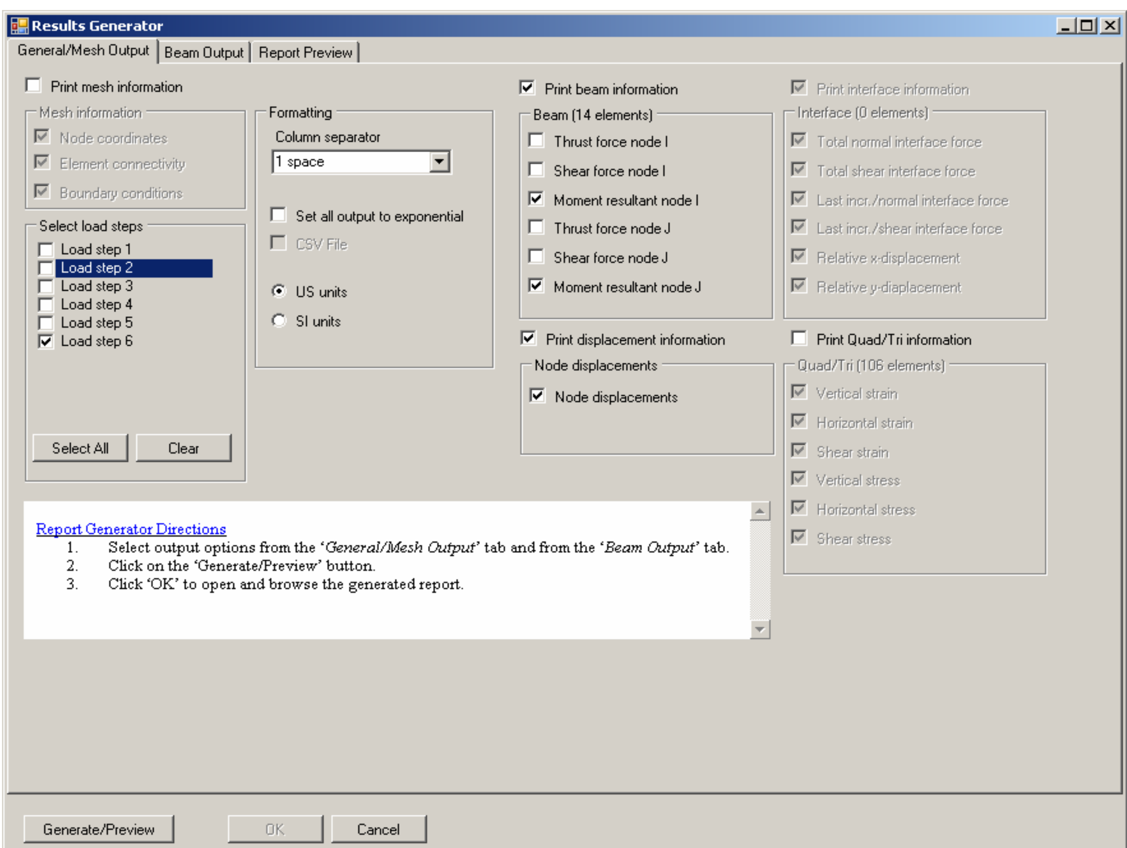
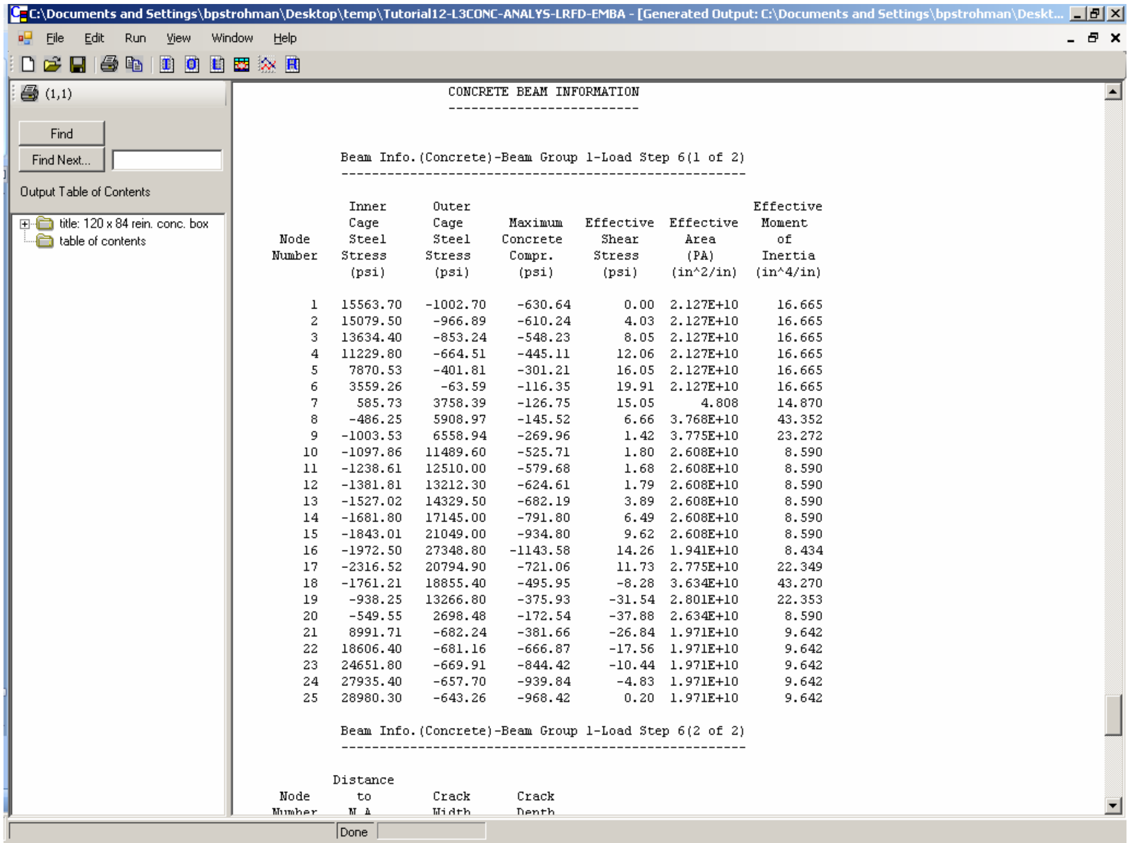


Figure 12-23 – Results Generator Input Screen – Load Step 6 Moments and Deflections



CONCRETE BEAM INFORMATION

Beam Info.(Concrete)-Beam Group 1-Load Step 6(1 of 2)

Node Number	Inner Cage Steel Stress (psi)	Outer Cage Steel Stress (psi)	Maximum Concrete Compr. (psi)	Effective Shear Stress (psi)	Effective Area (PA) (in^2/in)	Effective Moment of Inertia (in^4/in)
1	15563.70	-1002.70	-630.64	0.00	2.127E+10	16.665
2	15079.50	-966.89	-610.24	4.03	2.127E+10	16.665
3	13634.40	-853.24	-548.23	8.05	2.127E+10	16.665
4	11229.80	-664.51	-445.11	12.06	2.127E+10	16.665
5	7870.53	-401.81	-301.21	16.05	2.127E+10	16.665
6	3559.26	-63.59	-116.35	19.91	2.127E+10	16.665
7	585.73	3758.39	-126.75	15.05	4.808	14.870
8	-486.25	5908.97	-145.52	6.66	3.768E+10	43.352
9	-1003.53	6558.94	-269.96	1.42	3.775E+10	23.272
10	-1097.86	11489.60	-525.71	1.80	2.608E+10	8.590
11	-1238.61	12510.00	-579.68	1.68	2.608E+10	8.590
12	-1381.81	13212.30	-624.61	1.79	2.608E+10	8.590
13	-1527.02	14329.50	-682.19	3.89	2.608E+10	8.590
14	-1681.80	17145.00	-791.80	6.49	2.608E+10	8.590
15	-1843.01	21049.00	-934.80	9.62	2.608E+10	8.590
16	-1972.50	27348.80	-1143.58	14.26	1.941E+10	8.434
17	-2316.52	20794.90	-721.06	11.73	2.775E+10	22.349
18	-1761.21	18855.40	-495.95	-8.28	3.634E+10	43.270
19	-938.25	13266.80	-375.93	-31.54	2.801E+10	22.353
20	-549.55	2698.48	-172.54	-37.88	2.634E+10	8.590
21	8991.71	-682.24	-381.66	-26.84	1.971E+10	9.642
22	18606.40	-681.16	-666.87	-17.56	1.971E+10	9.642
23	24651.80	-669.91	-844.42	-10.44	1.971E+10	9.642
24	27935.40	-657.70	-939.84	-4.83	1.971E+10	9.642
25	28980.30	-643.26	-968.42	0.20	1.971E+10	9.642

Beam Info.(Concrete)-Beam Group 1-Load Step 6(2 of 2)

Node Number	Distance to N.A.	Crack Width	Crack Depth

Figure 12-24 – Results Generator – Load Step 6 Beam Stresses

This page intentionally left blank

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 13

May 2008

This page intentionally left blank

13. CANDE TEST PROBLEM 13

13.1 Problem Definition

Analyze a 217-inch span (82-inch rise) 3-segment type corrugated steel long span arch setting on concrete footings with 3 ft of fill over the top of the arch using Working Stress (service) analysis. The problem is shown schematically in Figure 13-1. The analysis will be with Level 3, using an imported finite element arch mesh in xml format from Tutorial Problem 9 for a trench installation having interface elements. This problem analyzes the corrugated steel long span arch from Tutorial Problem 9, which was performed using a Level 2 analysis.

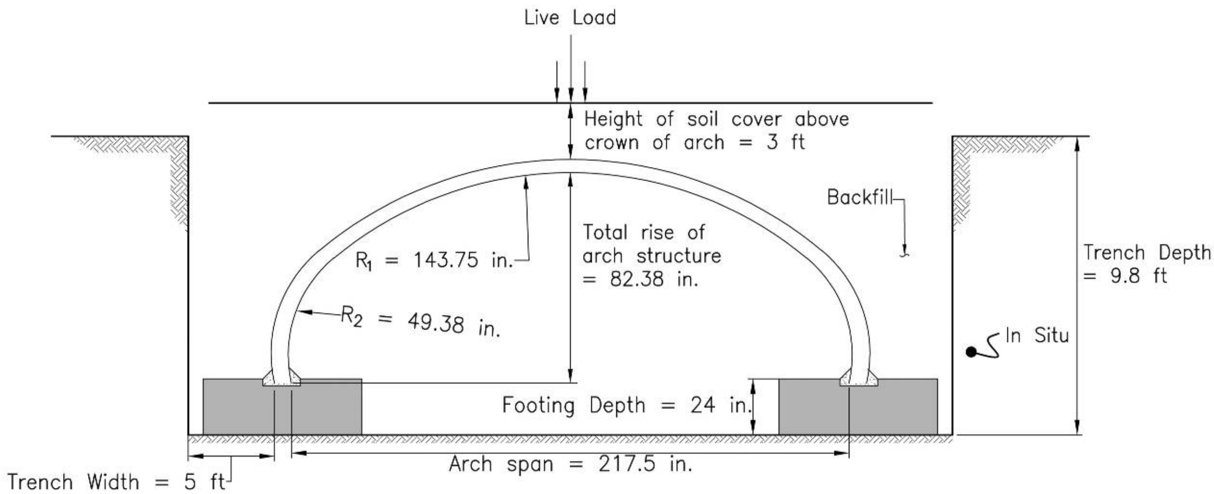


Figure 13-1 Details of Problem 13

Developing Level 3 input files for CANDE requires substantial advance planning relative to Level 2 input files. The engineer must define the installation geometry and boundary conditions in order to create a mesh. This problem imports Level 2 Tutorial Problem 9 into CANDE using xml format.

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - Service

Solution level - FEM-user mesh (Level 3)

Number of pipe element groups (Level 3 only) - 1

Pipe material type - Steel

Number of connected beam elements - 19

Joint slip - No joint slip

Soil parameters - Canned Duncan/Selig soil models except as noted:

In situ soil - ML 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Footing - Canned Linear Elastic - Young's modulus = 3,600,000 psi, Poisson's ratio = 0.17

Backfill soil - SW 95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils and 150 lb/ft^3 for the footing

Density of steel - 0.284 lb/in.^{3}

Culvert material behavior - Bilinear stress/strain

Analysis mode - Large deformation/buckling

Area of pipe wall unit length - 0.267 in.^{2}/in. (Assumes 0.218 in. thick with 6 in. x 2 in. corrugations)

Moment of inertia of pipe wall/unit length - 0.127 in.^{4}/in. (Assumes 0.218 in. thick with 6 in. x 2 in. corrugations)

Section modulus of pipe wall/unit length - 0.115 in.^{3}/in. (Assumes 0.218 in. thick with 6 in. x 2 in. corrugations)

Number of construction steps - 13

Interface material properties - The following summarizes material property definitions for the interface elements:

Interface number	Angle from x-axis to normal interface (degrees) *	Coefficient of friction between nodes i and j	Tensile breaking force of contact nodes
1	90	0.3	10
2	86.67	0.3	10
3	83.33	0.3	10
4	80.00	0.3	10
5	76.67	0.3	10
6	73.33	0.3	10
7	70.00	0.3	10
8	66.67	0.3	10
9	63.33	0.3	10
10	60.00	0.3	10
11	56.67	0.3	10
12	53.33	0.3	10
13	50.00	0.3	10
14	41.02	0.3	10
15	32.04	0.3	10
16	23.06	0.3	10
17	14.07	0.3	10
18	5.09	0.3	10
19	-3.89	0.3	10

* *For an example calculation depicting how these interface angles were computed and/or an explanation of the shortcut method for defining interface material properties, see CANDE Tutorial Problem 6.*

13.2 Creating the CANDE Input Document

Figures 13-2 through 13-5 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure.

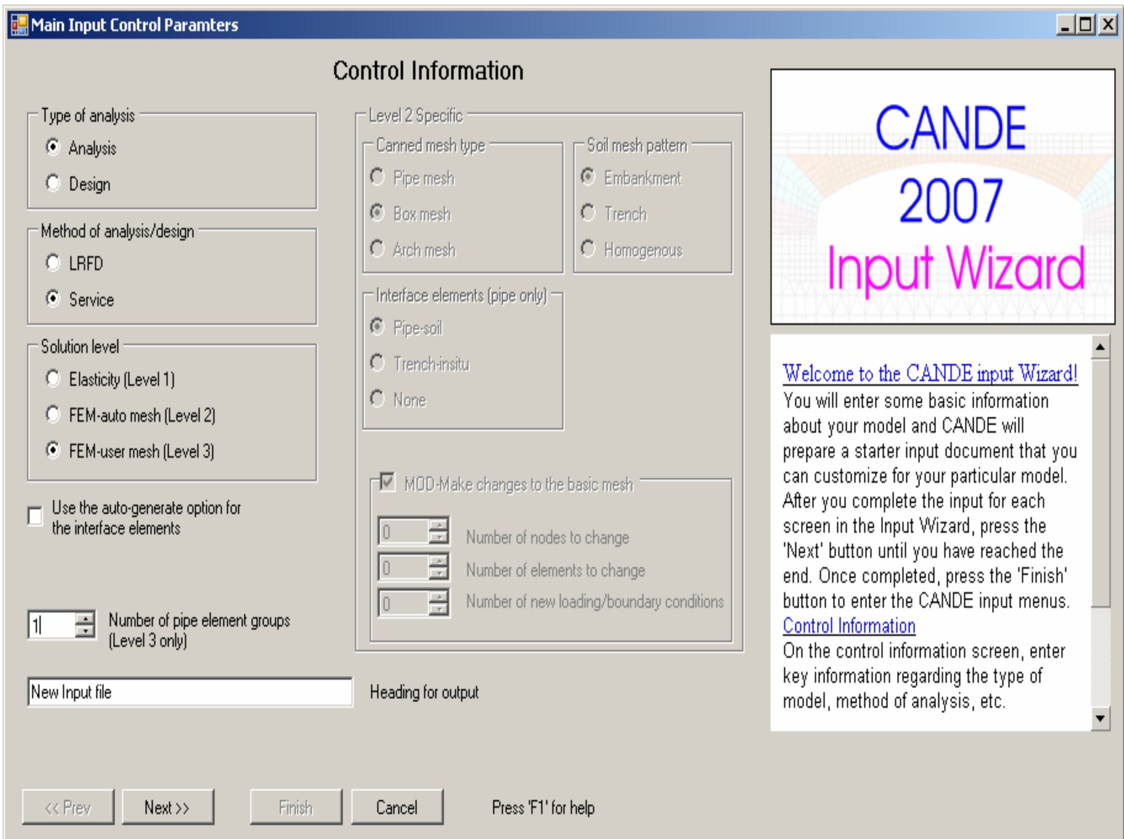


Figure 13-2 – Input Wizard, Screen 1

To import the mesh from Tutorial Problem 9 using xml format, select "Import Mesh file" (see Figure 13-3). After selecting "Import Mesh file," perform the following steps: click on the symbol Import file , select "CANDE Mesh XML, click "import," select the file named 'Tutorial9L2METL-ANALYS-WSD-TREN_MeshGeom.xml', click "Open," and then click "OK." Figure 13-3 displays the input screen after all of these actions have been performed. Note: the xml filename is based on the filename selected by the user with _MeshGeom.xml added onto the end. This xml file is automatically generated by CANDE anytime a valid Level 2 or Level 3 problem is successfully run. After selecting this file, the "Level 3 options" fields (Figure 13-3) should fill in. Now click "Next" and CANDE will prompt the user to input the pipe material information. Enter the data provided above and the screen should appear as Figure 13-4. Now click "Next" and input the soil properties. After clicking "Next" on Screen 4, CANDE will display a screen indicating that an input path is initiated - click "Finish." The user is prompted for a filename and directory.

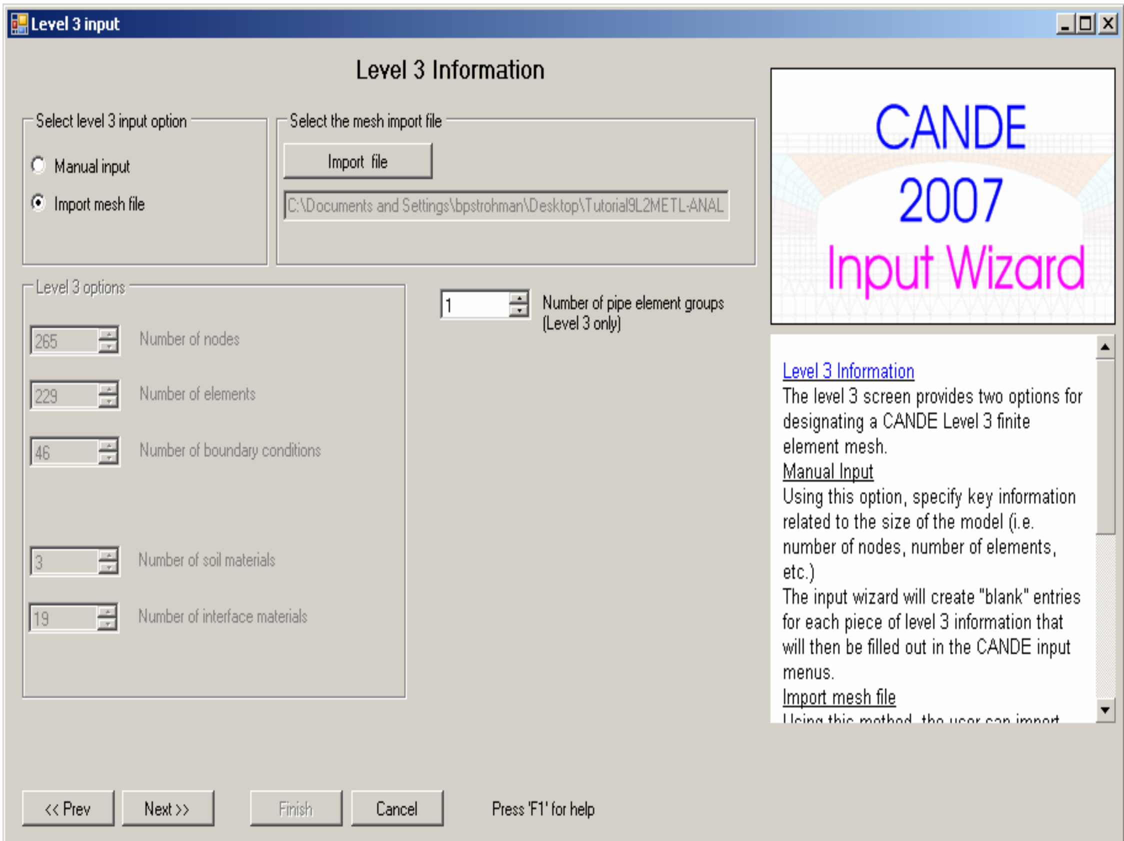


Figure 13-3 – Input Wizard, Screen 2

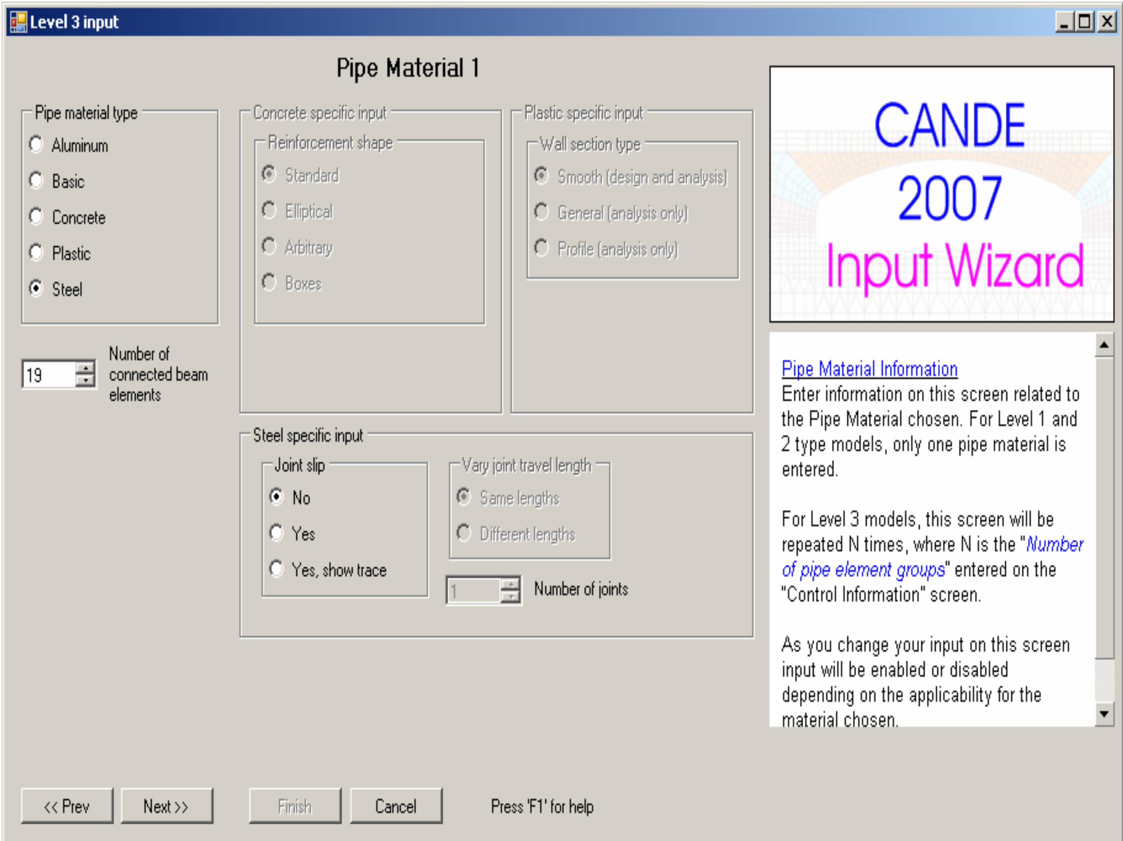


Figure 13-4 – Input Wizard, Screen 3

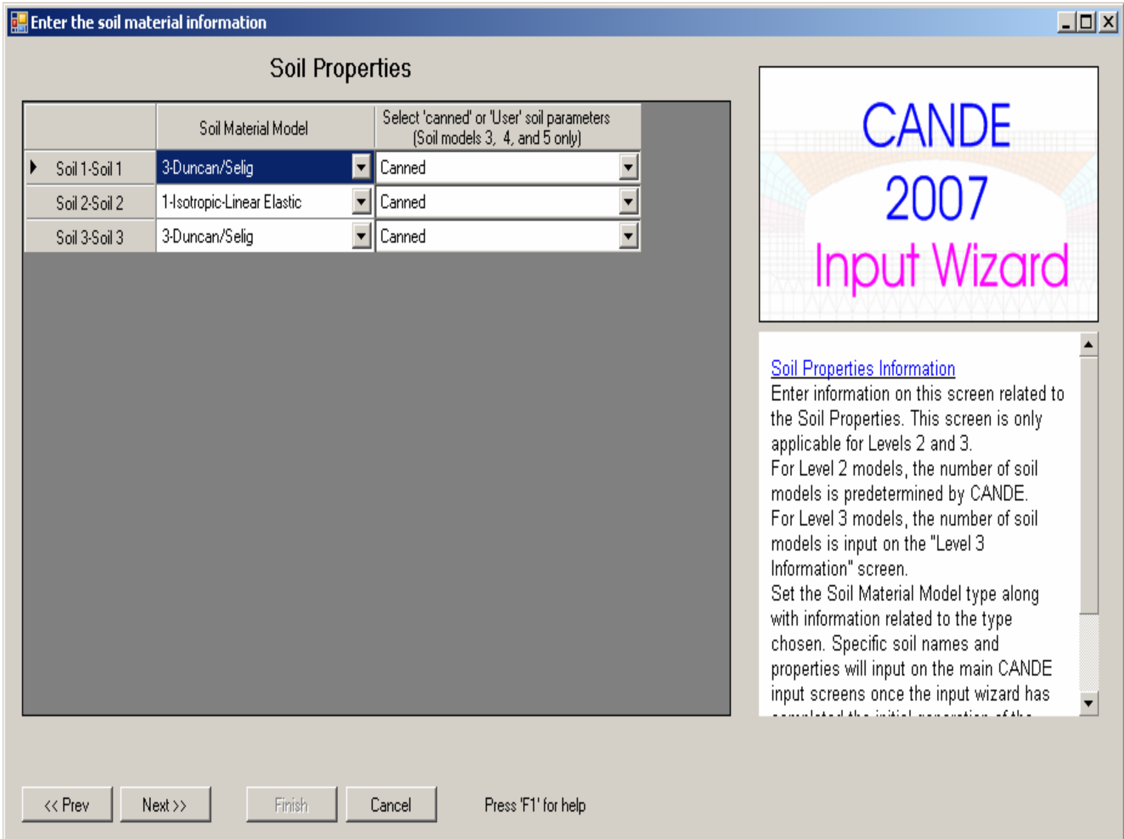


Figure 13-5 – Input Wizard, Screen 4

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 13-6. Enter an appropriate heading for output and click “Accept Input.”

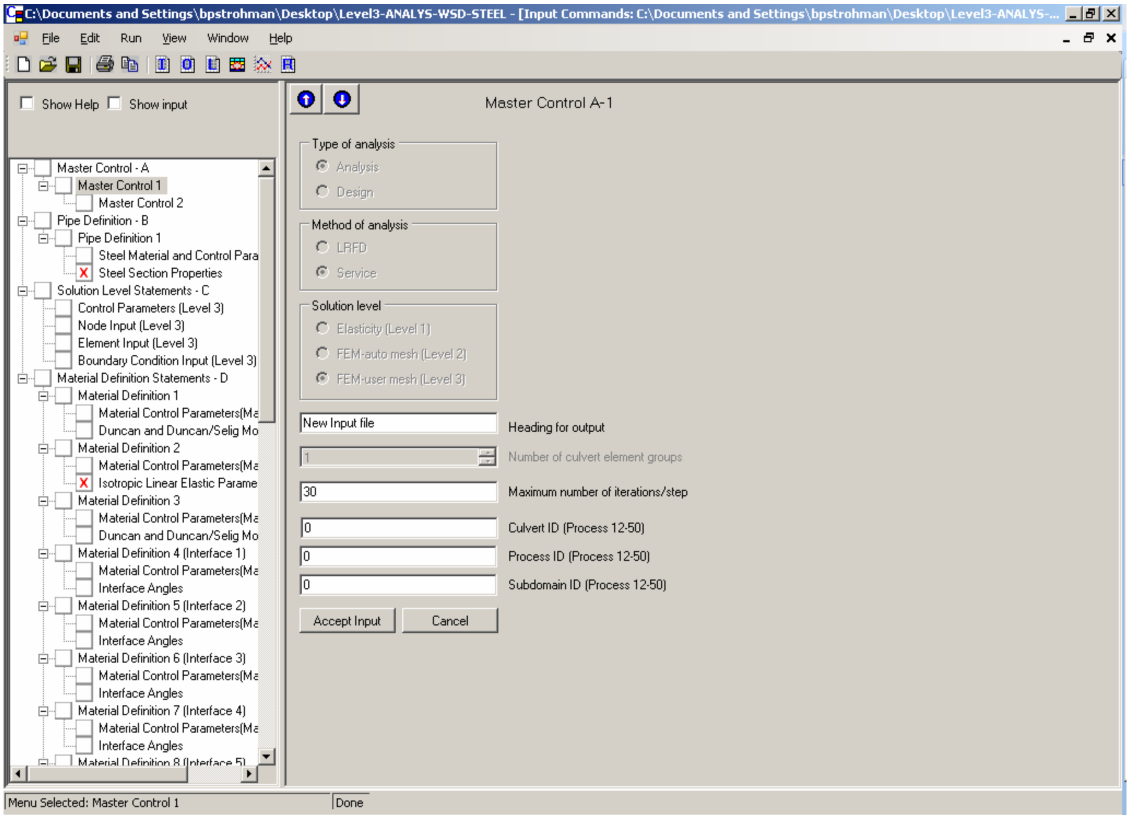


Figure 13-6 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 13-6 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 13-7 through 13-11 show the completed input for the screens requiring data for the tutorial, except that only one material definition is shown. Note: Interface material properties are stored in the xml file and are therefore imported directly into the program from Tutorial 9. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

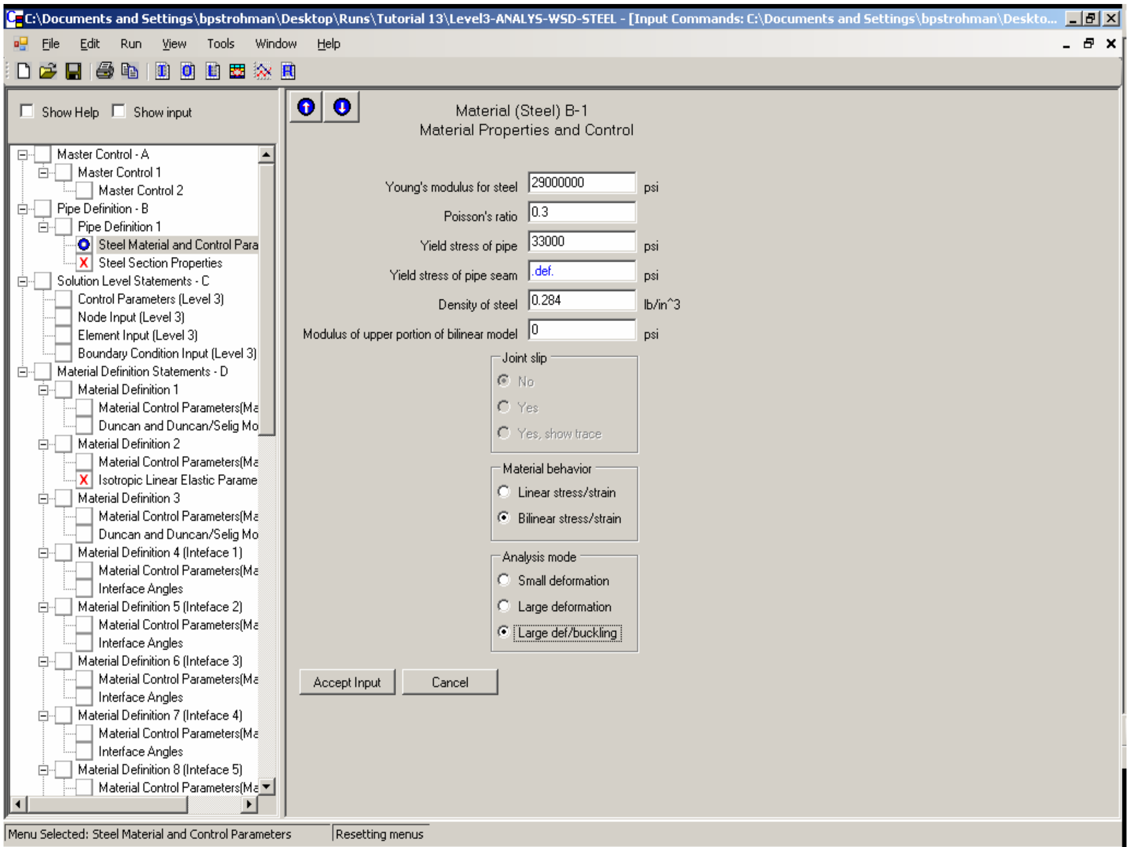


Figure 13-7 – Input Screen B-1

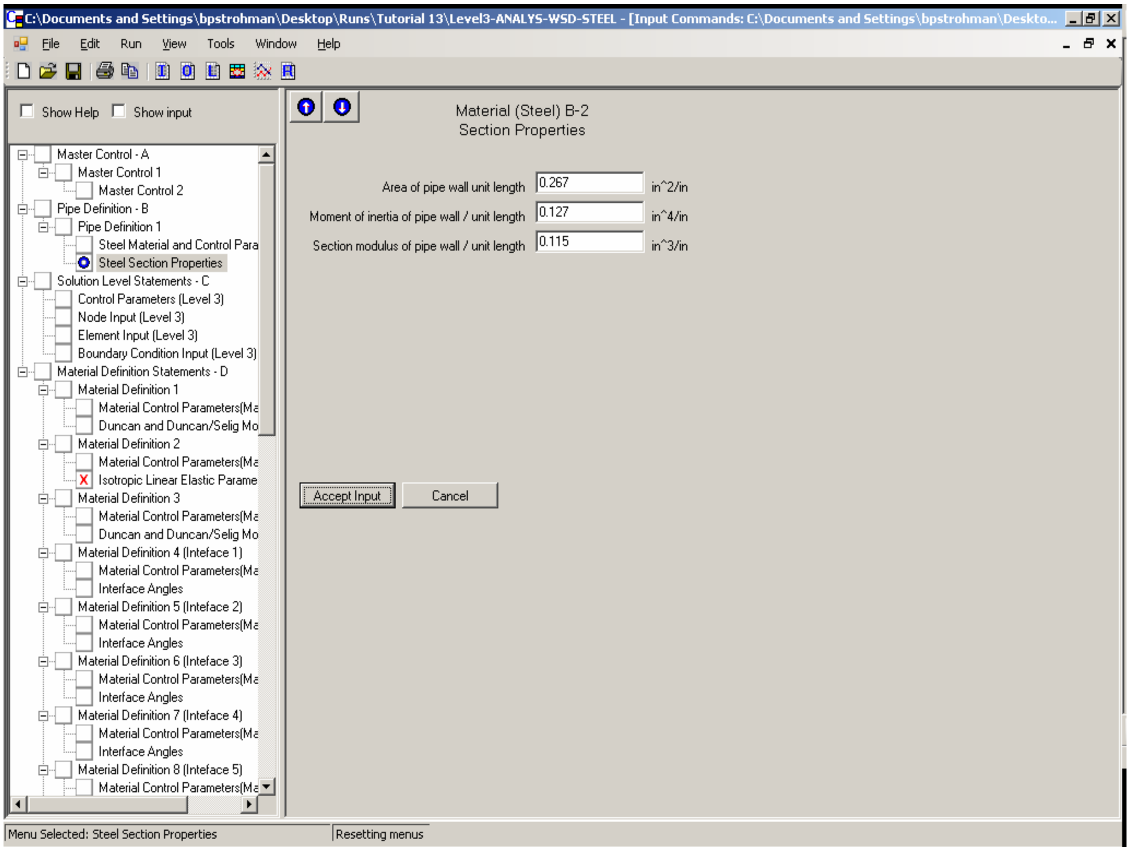


Figure 13-8 – Input Screen B-2

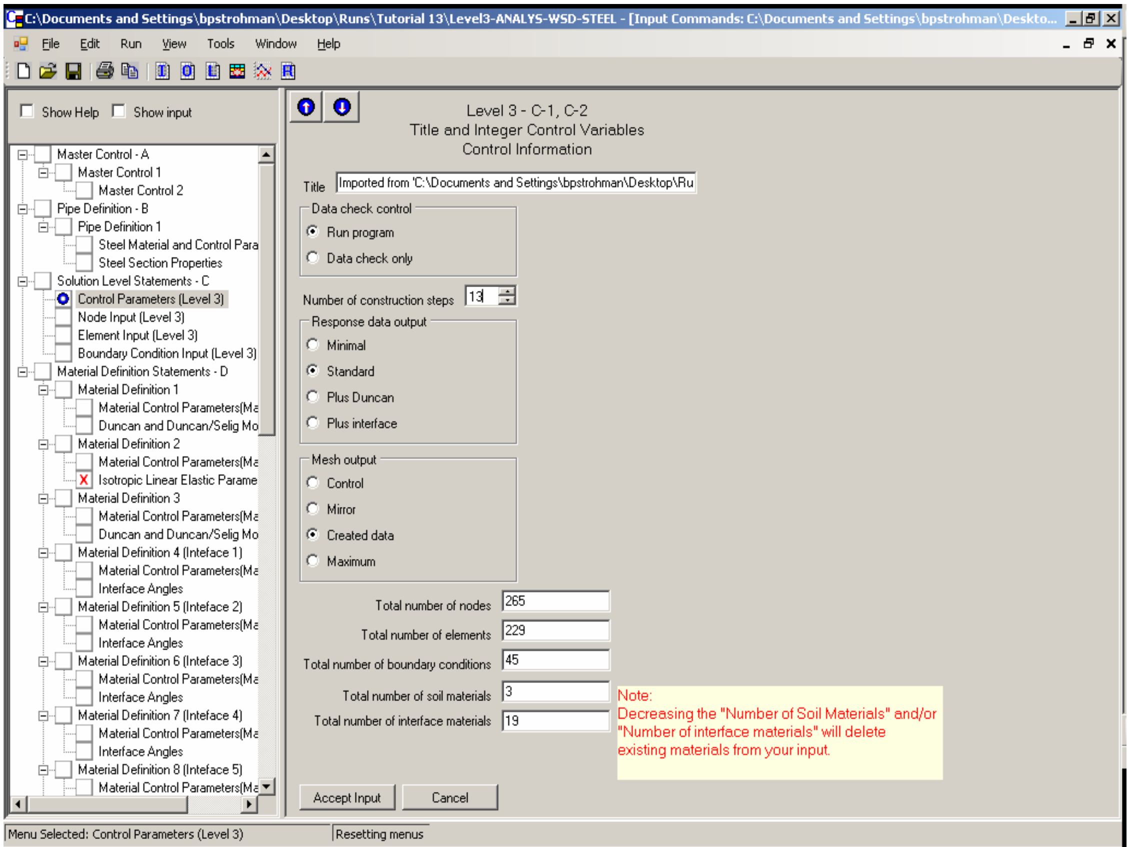


Figure 13-9 – Input Screen C-1, C-2

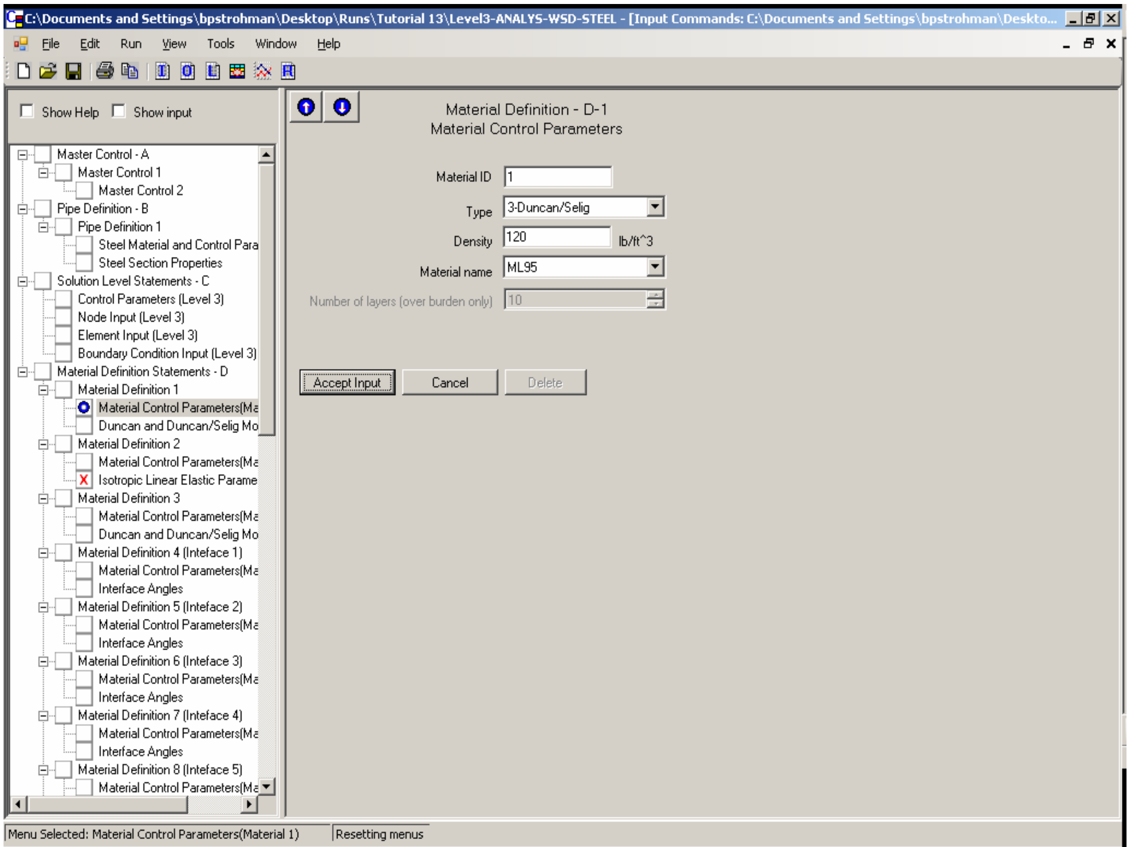


Figure 13-10 – Input Screen D-1 for Material 1

(Note: Repeat for Materials 2 and 3 with values provided in the ‘Problem Definition’)

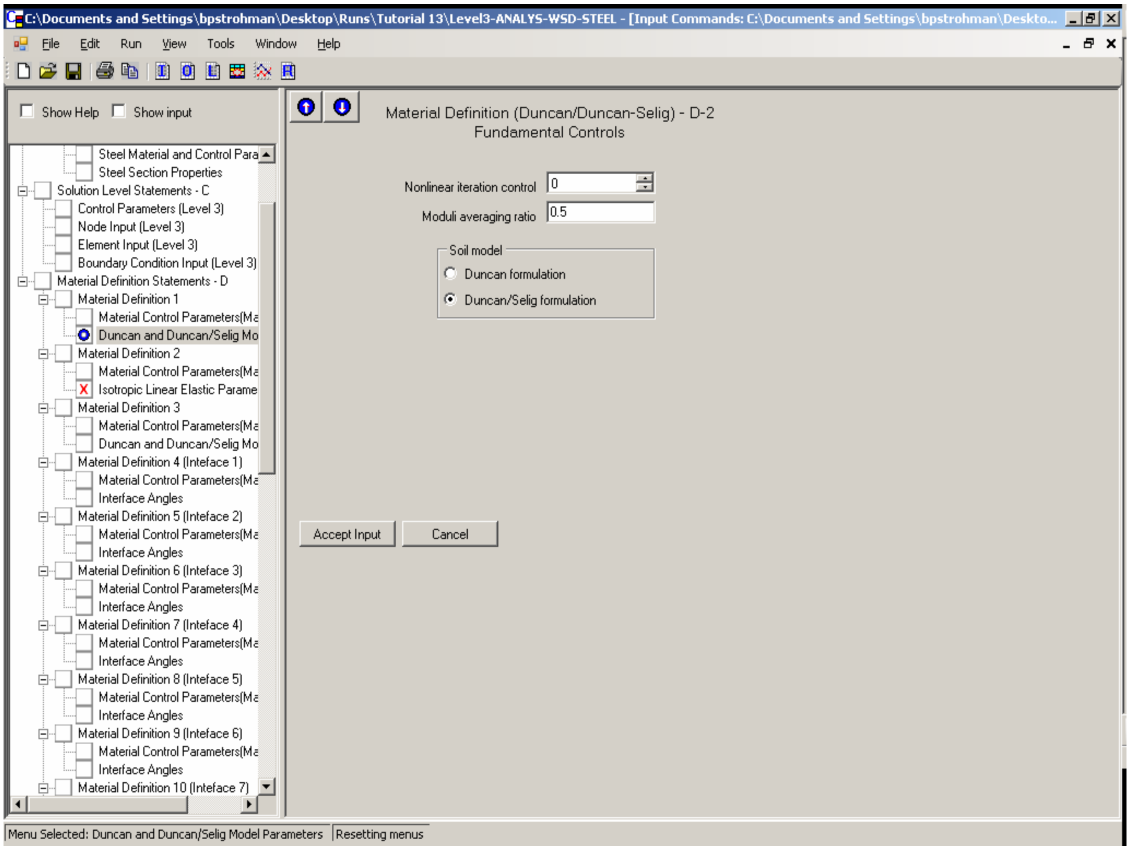


Figure 13-11 – Input Screen D-2 for Material
(Note: Repeat for Materials 2 and 3 with values provided in the 'Problem Definition')

When all input is complete and saved, click "Run" and "CANDE-2007" on the main toolbar to execute the program. This will open the "Running CANDE" window which will allow you to monitor the run as it progresses. When the program is complete, a check box "CANDE Analysis Complete" window will appear. Click "OK" and check if the "Running CANDE" window states "Normal Exit From CANDE." If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the "Close" button in the "Running CANDE" window to return to the user interface.

13.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the “View” menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The “Plotting Parameters” button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click “View” and then “Mesh Plot” to open the mesh plot. Open the plotting parameters menu and click the check box “Show constr. increment numbers.” Click “OK,” set Load Increment to 9 to show the entire mesh and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 13-12.

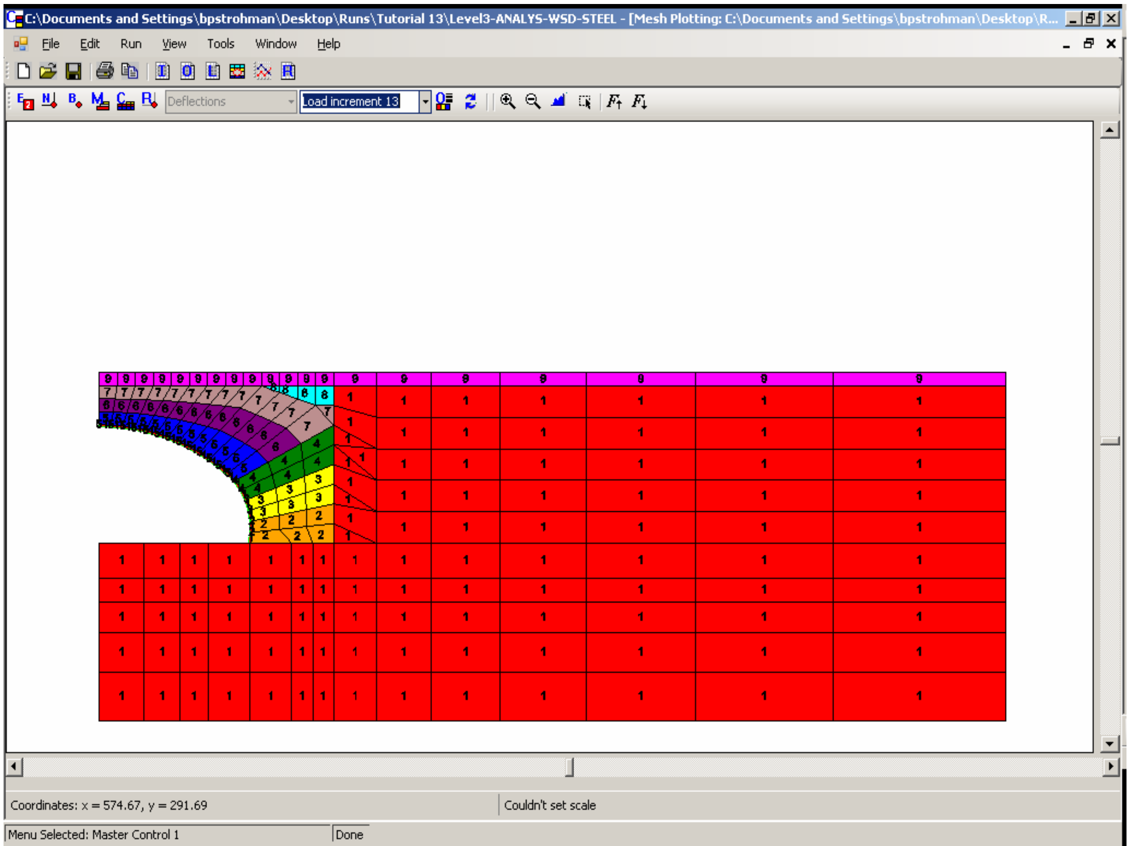


Figure 13-12 – Mesh Plot for Load Steps 1 to 9

To view shear strain, open the plotting parameters window and set the deflection magnification factor to 10 (see Figure 13-13).

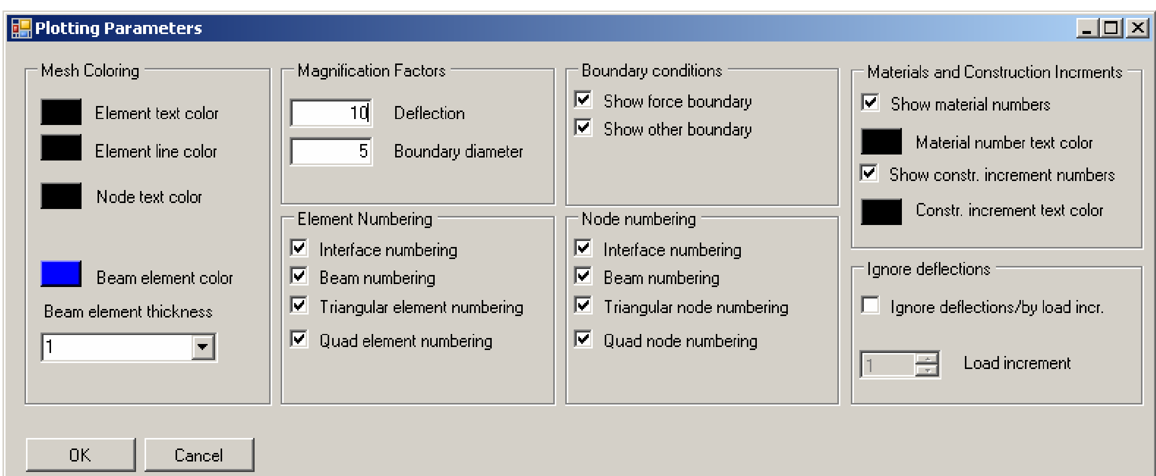


Figure 13-13 – Mesh Window Plotting Parameters

Select “OK” to close the window. Click the icon to “Turn on/off selected output results” and set the drop down box to Shear Strain. Set the increment to 13 and the screen should look like Figure 13-14, which also shows the deflected mesh geometry. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

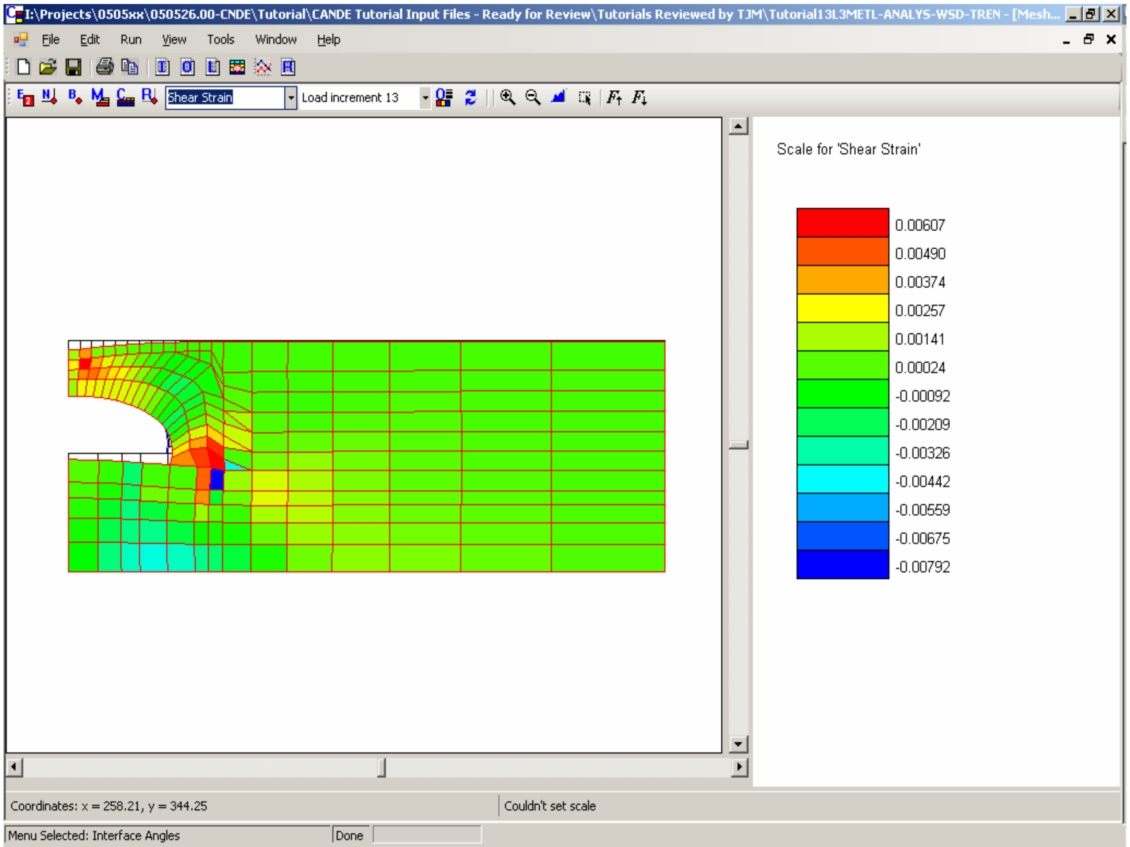


Figure 13-14 – Shear Strain Load Step 13

Similarly, using the “View CANDE Graphs” button on the toolbar or selecting “View/Graphs” the user can explore the forces in the culvert itself. Figure 13-15 compares the thrusts after three increments. This is accomplished using the “Plot multiple load steps” icon at the far left of the toolbar to uncheck “Show single load step” and check Load Steps 7, 11, and 13, then “OK.” The “Turn on/off view of pipe” icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to “Thrust Force” and the screen should appear as Figure 13-15.

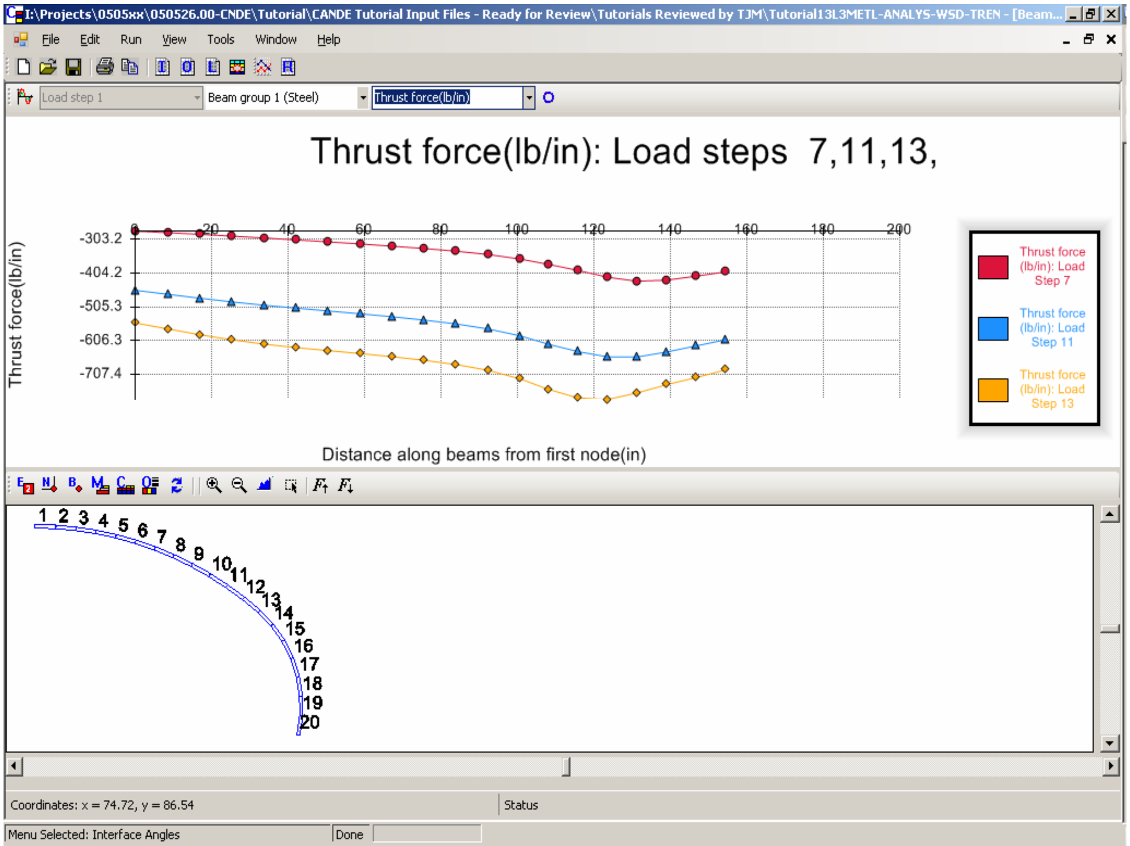


Figure 13-15 – Thrust Force for Load Steps 7, 11, and 13

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 13-16) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 13-16 shows the final assessment printed at the end of the file.

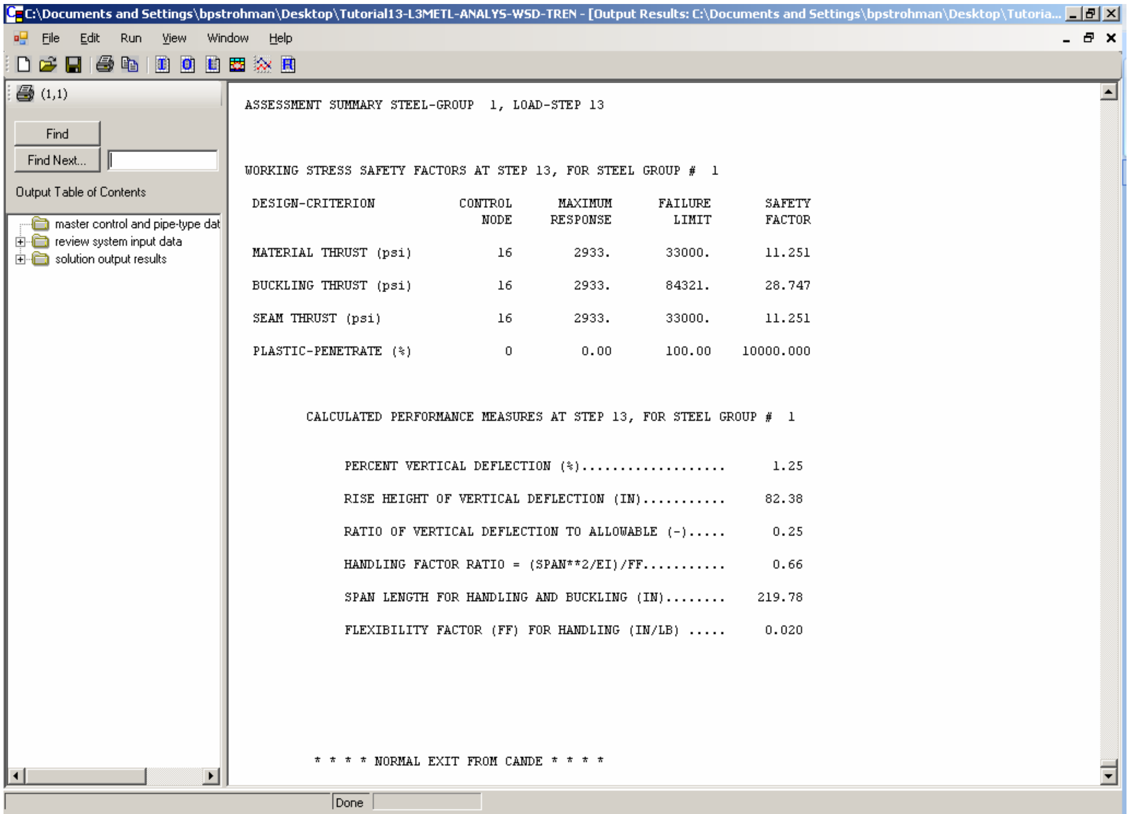


Figure 13-16 – Design Assessment Summary – Load Step 13

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 13-17 shows the Results Generator input screen set to obtain deflection and shear data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 13-18. The three tabs shown in Figures 13-17 and 13-18 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

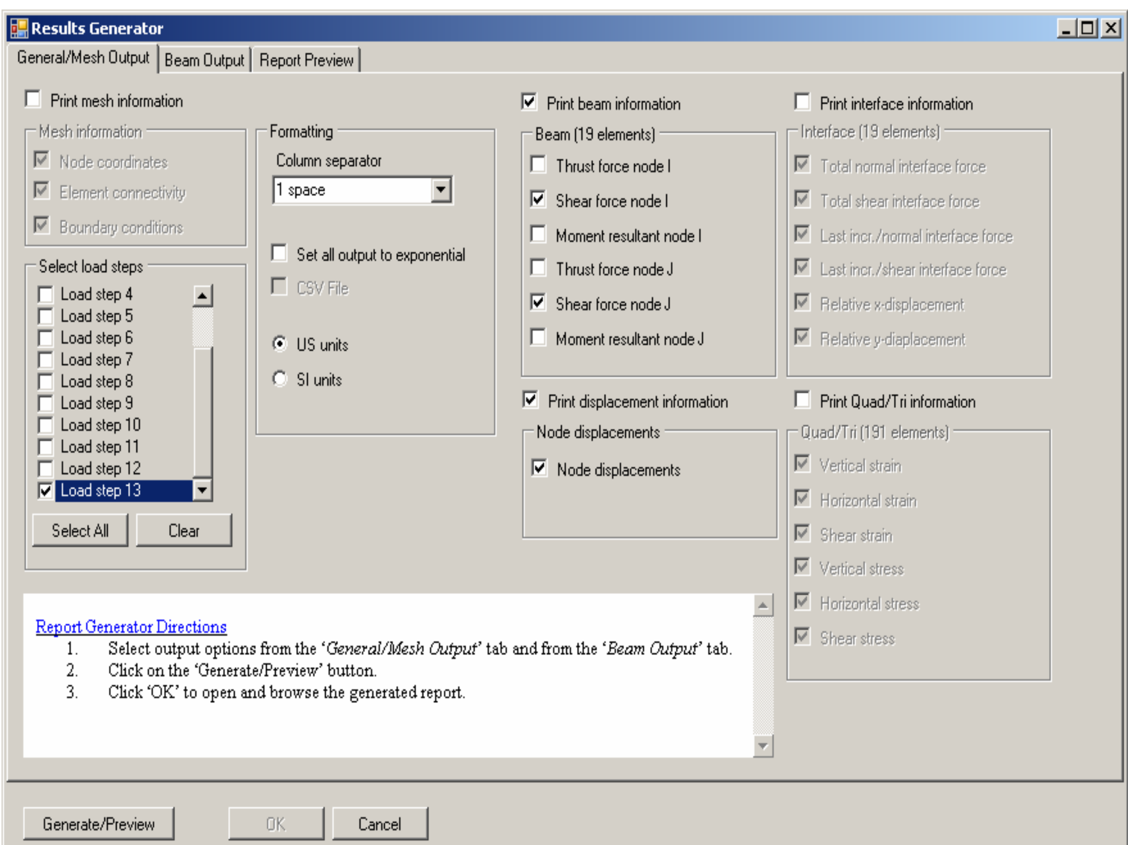
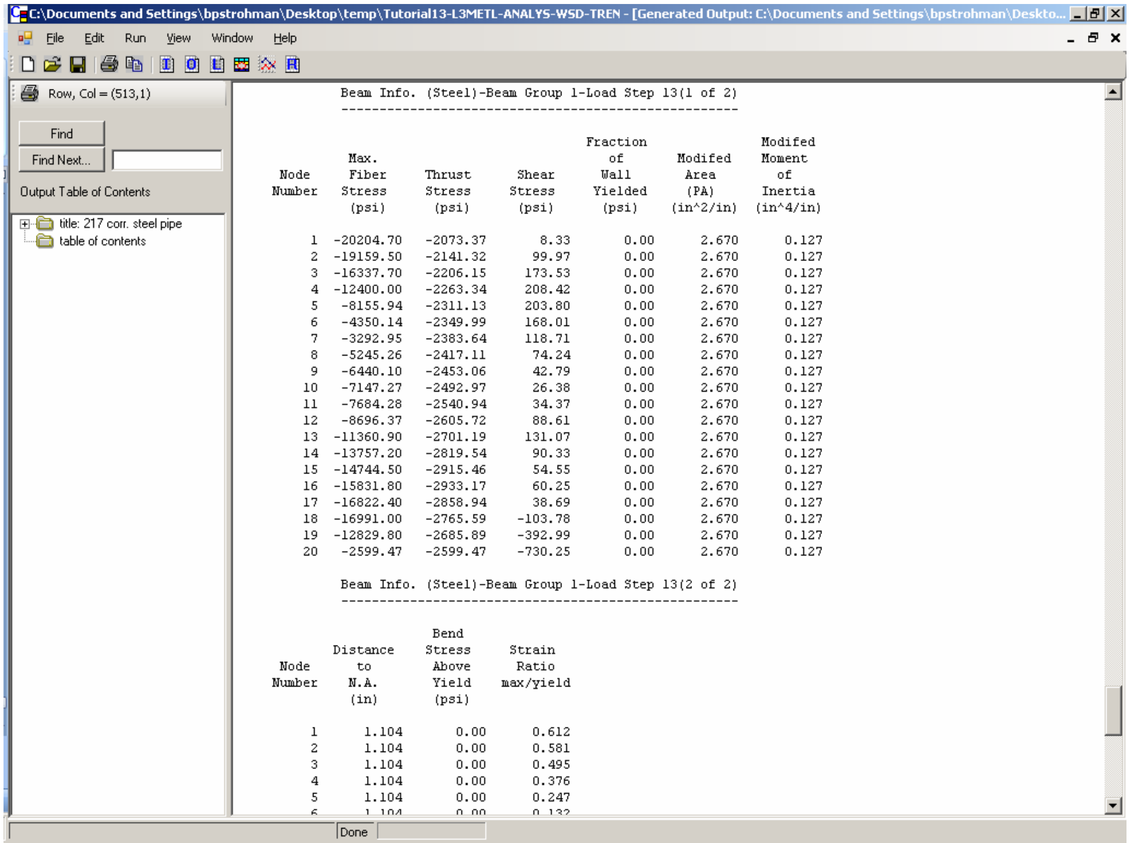


Figure 13-17 – Results Generator Input Screen – Load Step 13 Shear and Deflections



Beam Info. (Steel)-Beam Group 1-Load Step 13(1 of 2)

Node Number	Max. Fiber Stress (psi)	Thrust Stress (psi)	Shear Stress (psi)	Fraction of Wall Yielded (psi)	Modifed Area (PA) (in^2/in)	Modifed Moment of Inertia (in^4/in)
1	-20204.70	-2073.37	8.33	0.00	2.670	0.127
2	-19159.50	-2141.32	99.97	0.00	2.670	0.127
3	-16337.70	-2206.15	173.53	0.00	2.670	0.127
4	-12400.00	-2263.34	208.42	0.00	2.670	0.127
5	-8155.94	-2311.13	203.80	0.00	2.670	0.127
6	-4350.14	-2349.99	168.01	0.00	2.670	0.127
7	-3292.95	-2383.64	118.71	0.00	2.670	0.127
8	-5245.26	-2417.11	74.24	0.00	2.670	0.127
9	-6440.10	-2453.06	42.79	0.00	2.670	0.127
10	-7147.27	-2492.97	26.38	0.00	2.670	0.127
11	-7684.28	-2540.94	34.37	0.00	2.670	0.127
12	-8696.37	-2605.72	88.61	0.00	2.670	0.127
13	-11360.90	-2701.19	131.07	0.00	2.670	0.127
14	-13757.20	-2819.54	90.33	0.00	2.670	0.127
15	-14744.50	-2915.46	54.55	0.00	2.670	0.127
16	-15831.80	-2933.17	60.25	0.00	2.670	0.127
17	-16822.40	-2858.94	38.69	0.00	2.670	0.127
18	-16991.00	-2765.59	-103.78	0.00	2.670	0.127
19	-12829.80	-2685.89	-392.99	0.00	2.670	0.127
20	-2599.47	-2599.47	-730.25	0.00	2.670	0.127

Beam Info. (Steel)-Beam Group 1-Load Step 13(2 of 2)

Node Number	Distance to N.A. (in)	Bend Stress Above Yield (psi)	Strain Ratio max/yield
1	1.104	0.00	0.612
2	1.104	0.00	0.581
3	1.104	0.00	0.495
4	1.104	0.00	0.376
5	1.104	0.00	0.247

Figure 13-18 – Results Generator – Load Step 13 Beam Stresses

NCHRP Project 15-28

Modernize and Upgrade CANDE for Analysis and Design of Buried Structures

User Tutorial – Problem 14

May 2008

This page intentionally left blank

14. CANDE TEST PROBLEM 14

14.1 Problem Definition

Analyze a two material structure composed of a reinforced concrete U-shaped base with 15-foot span and 5-foot rise supporting a pin connected, corrugated aluminum arch-shaped roof with 13 ft of fill over the top of the arch. The problem is shown schematically in Figure 14-1. The analysis will be with Level 3, using an imported finite element mesh in xml format.

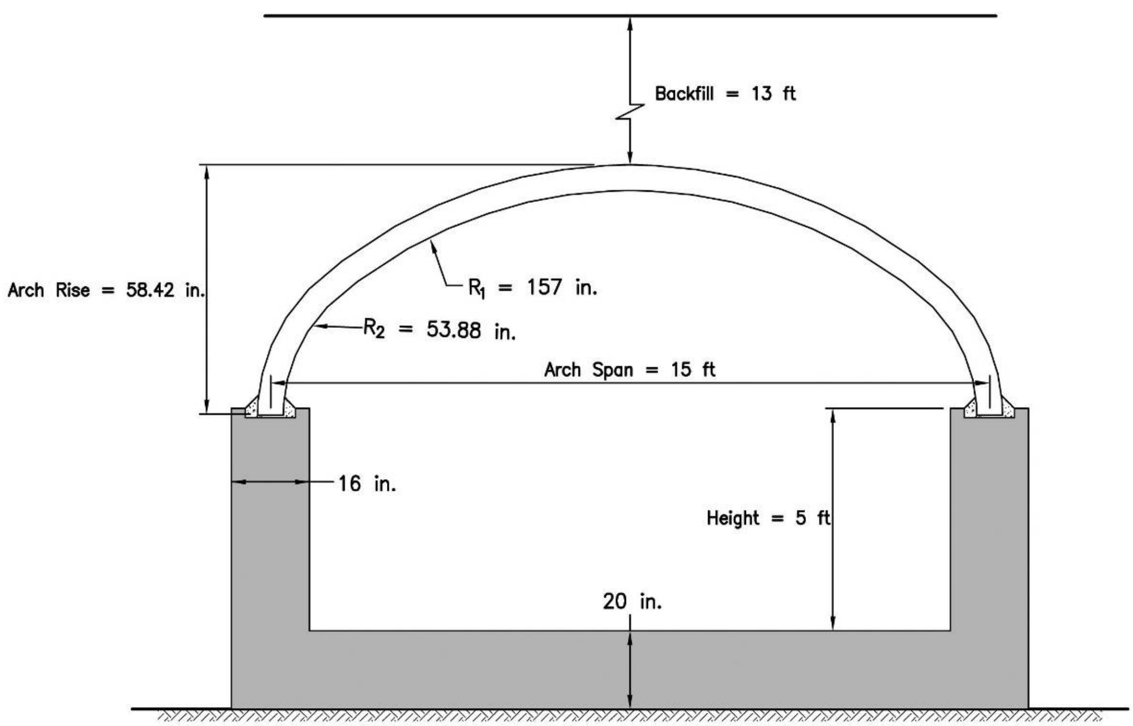


Figure 14-1 Details of Problem 14

Developing Level 3 input files for CANDE requires substantial advance planning relative to Level 2 input files. The engineer must define the installation geometry and boundary conditions in order to create a mesh. This problem imports an existing finite element mesh in xml format.

Some of the most important parameters assumed for the test problem are listed below. Most of these parameters can have a significant impact on the design. Actual values should be used whenever possible. Lacking actual data, designers should vary the uncertain parameters to investigate the sensitivity of the design. Note that CANDE has numerous additional input parameters that typically default to common values but which should be reviewed for each design.

Type of analysis - Analysis

Method of analysis/design - Service

Solution level - FEM-user mesh (Level 3)

Number of pipe element groups (Level 3 only) - 2

Pipe material type (Pipe Material 1) - Concrete

Reinforcement shape (Pipe Material 1) - Arbitrary

Number of connected beam elements (Pipe Material 1) - 40

Pipe material type (Pipe Material 2) - Aluminum

Number of connected beam elements (Pipe Material 2) - 26

Soil parameters -

In situ soil - Canned Linear Elastic - Young's modulus = 3,500 psi, Poisson's ratio = 0.25

Backfill soil - Canned Duncan/Selig - SW-95, LRFD stiffness control = 0, Moduli averaging ratio = 0.5, soil model = Duncan/Selig formulation

Soil density - 120 lb/ft^3 for all soils

The following are input parameters for Pipe Material 1:

Compressive strength of concrete (f'c) - 5,000 lb/in.^{2}

Shear strength equation - Boxes/3-sided > 2 ft fill (AASHTO 5.14.5.3)

Concrete strain at tension rupture - 0.00001 in./in.

Compressive strain at end of elastic range - 0.001 in./in.

Compressive strain at the initial strength limit - 0.002 in./in.

Unit weight of concrete for body weight - 150 lb/ft^3

Crack width model - Gergely-Lutz

Analysis mode - Small deformation

Yield stress of reinforcing steel - 65,000 lb/in.^{2}

Inner surface spacing between rows of rebar - 2 in.

Outer surface spacing between rows of rebar - 2 in.

Type of reinforcement - Welded or deformed wire

Nonlinear behavior selection - Option 3 Plus steel yielding behavior

Concrete wall thickness and reinforcement steel - The following summarizes the wall thickness and reinforcing steel for the reinforced concrete U-shaped base (Pipe Material 1):

Node	Wall Thickness (in.)	As Cage 1 (in.^{2}/in.)	As Cage 2 (in.^{2}/in.)	Cover Cage 1 (in.)	Cover Cage 2 (in.)
1	16	0.02	0.02	1.5	1.5
2	16	0.02	0.02	1.5	1.5
3	16	0.02	0.02	1.5	1.5
4	16	0.02	0.02	1.5	1.5
5	16	0.02	0.02	1.5	1.5
6	16	0.02	0.02	1.5	1.5
7	16	0.02	0.02	1.5	1.5
8	16	0.02	0.02	1.5	1.5
9	16	0.02	0.02	1.5	1.5
10	16	0.02	0.02	1.5	1.5
11	16	0.02	0.02	1.5	1.5
12	20	0.02	0.02	1.5	1.5
13	20	0.02	0.02	1.5	1.5
14	20	0.02	0.02	1.5	1.5
15	20	0.02	0.02	1.5	1.5
16	20	0.02	0.02	1.5	1.5
17	20	0.02	0.02	1.5	1.5
18	20	0.02	0.02	1.5	1.5
19	20	0.02	0.02	1.5	1.5
20	20	0.02	0.02	1.5	1.5
21	20	0.02	0.02	1.5	1.5
22	20	0.02	0.02	1.5	1.5
23	20	0.02	0.02	1.5	1.5
24	20	0.02	0.02	1.5	1.5
25	20	0.02	0.02	1.5	1.5
26	20	0.02	0.02	1.5	1.5
27	20	0.02	0.02	1.5	1.5
28	20	0.02	0.02	1.5	1.5
29	20	0.02	0.02	1.5	1.5
30	20	0.02	0.02	1.5	1.5
31	16	0.02	0.02	1.5	1.5
32	16	0.02	0.02	1.5	1.5
33	16	0.02	0.02	1.5	1.5
34	16	0.02	0.02	1.5	1.5
35	16	0.02	0.02	1.5	1.5
36	16	0.02	0.02	1.5	1.5
37	16	0.02	0.02	1.5	1.5

38	16	0.02	0.02	1.5	1.5
39	16	0.02	0.02	1.5	1.5
40	16	0.02	0.02	1.5	1.5
41	16	0.02	0.02	1.5	1.5

In the above table, Cage 1 refers to the inside face reinforcing steel and Cage 2 refers to the outside face reinforcing steel.

The following are input parameters for Pipe Material 2:

Density - 0.0975 lb/in.^{3}

Material behavior - Bilinear stress/strain

Area of pipe wall / unit length - 0.146 in.^{2}/in. (Assumes 0.125 in. thick with 9 in. x 2.5 in. corrugations)

Moment of inertia of pipe wall / unit length - 0.104 in.^{4}/in. (Assumes 0.125 in. thick with 9 in. x 2.5 in. corrugations)

Section modulus of pipe wall / unit length - 0.185 in.^{3}/in. (Assumes 0.125 in. thick with 9 in. x 2.5 in. corrugations)

The following are the remaining general input parameters:

Number of construction steps - 14

Total number of soil materials - 2

Total number of interface materials - 2 (to model the pin connection between the reinforced concrete U-shaped base and the corrugated aluminum arch-shaped roof, interface elements with high values of friction and tensile breaking force between contact nodes must be used at each of the points where the aluminum arch connects with the concrete base. Interface properties are list below).

Angle from x-axis to normal interface - Varies for each interface element. Since the angle of the aluminum arch with respect to the x-axis at the point where it meets the concrete base is 90 degrees, the interface angles for the pin connection will be ±90 degrees. Whether the angle is ±90 degrees for a particular node depends on how the interface element is defined. See Figure 5.5-18 of the *User Manual* for details regarding the interface elements.

Coefficient of friction between nodes I and J - 1,000,000

Tensile breaking force of contact nodes - 100,000 lb/in.

14.2 Creating the CANDE Input Document

Figures 14-2 through 14-4 show the CANDE Input Wizard screens created to initiate the input document. Enter the data to define the structure.

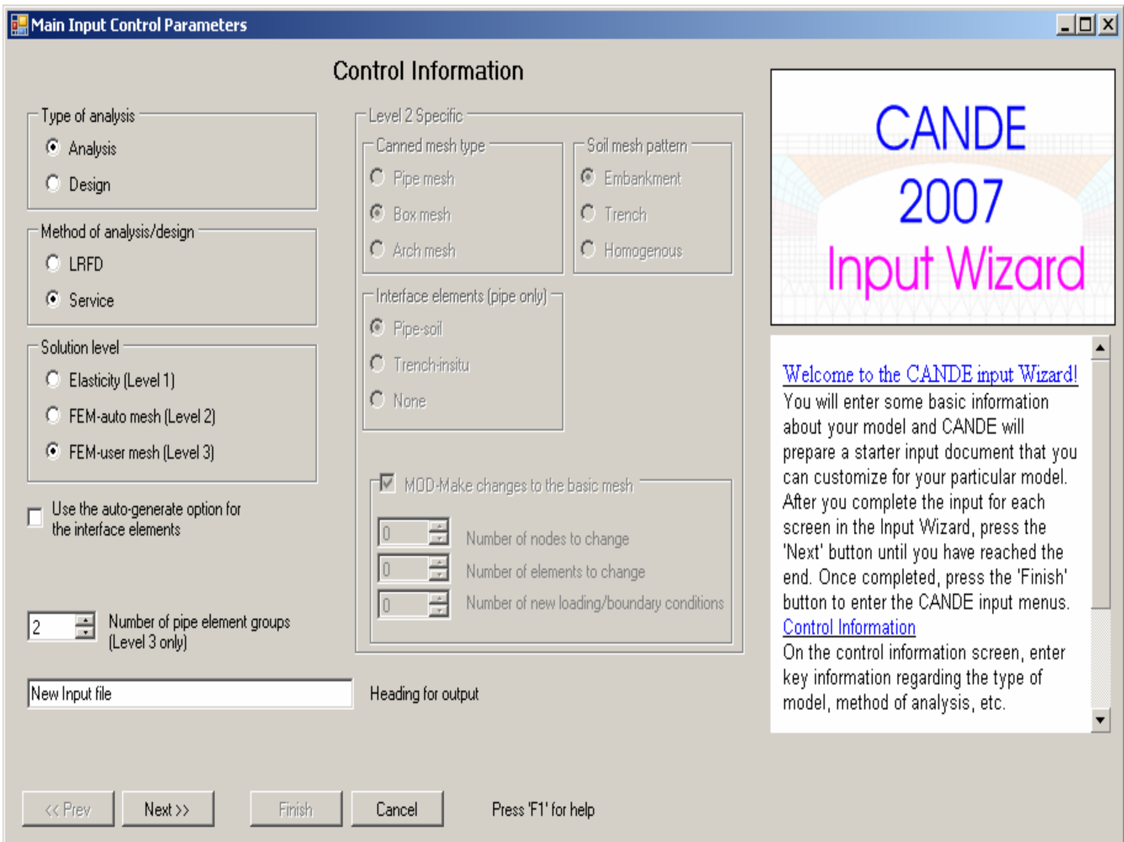


Figure 14-2 – Input Wizard, Screen 1

To import the mesh using xml format, select "Import Mesh file" (see Figure 14-3). After selecting "Import Mesh file," perform the following steps: click on the symbol Import file, select "CANDE Mesh XML," click "import," select the file named 'Tutorial14Import_MeshGeom.xml', click "Open," and then click "OK." Figure 14-3 displays the input screen after all of these actions have been performed. *Note: the xml filename is based on the filename selected by the user with _MeshGeom.xml added onto the end. This xml file is automatically generated by CANDE anytime a valid Level 2 or Level 3 problem is successfully run.* After selecting this file, the "Level 3 options" fields (Figure 14-3) should fill in. Now click "Next" and CANDE will prompt the user to input the pipe material information for each of the pipe groups. Enter the data provided above and the screens should appear as Figures 14-4 and 14-5. Now click “Next” and input the soil properties. After clicking “Next” on Screen 5, CANDE will display a screen indicating that an input path is initiated - click “Finish.” The user is prompted for a filename and directory.

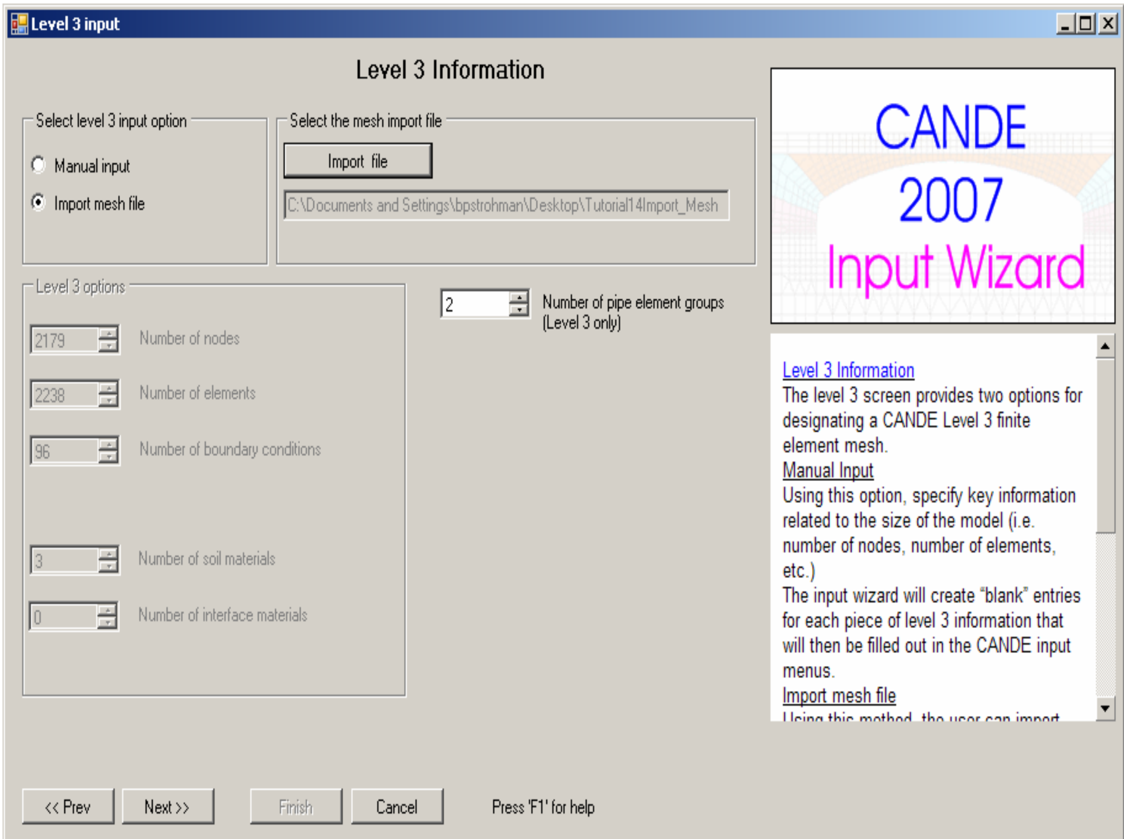


Figure 14-3 – Input Wizard, Screen 2

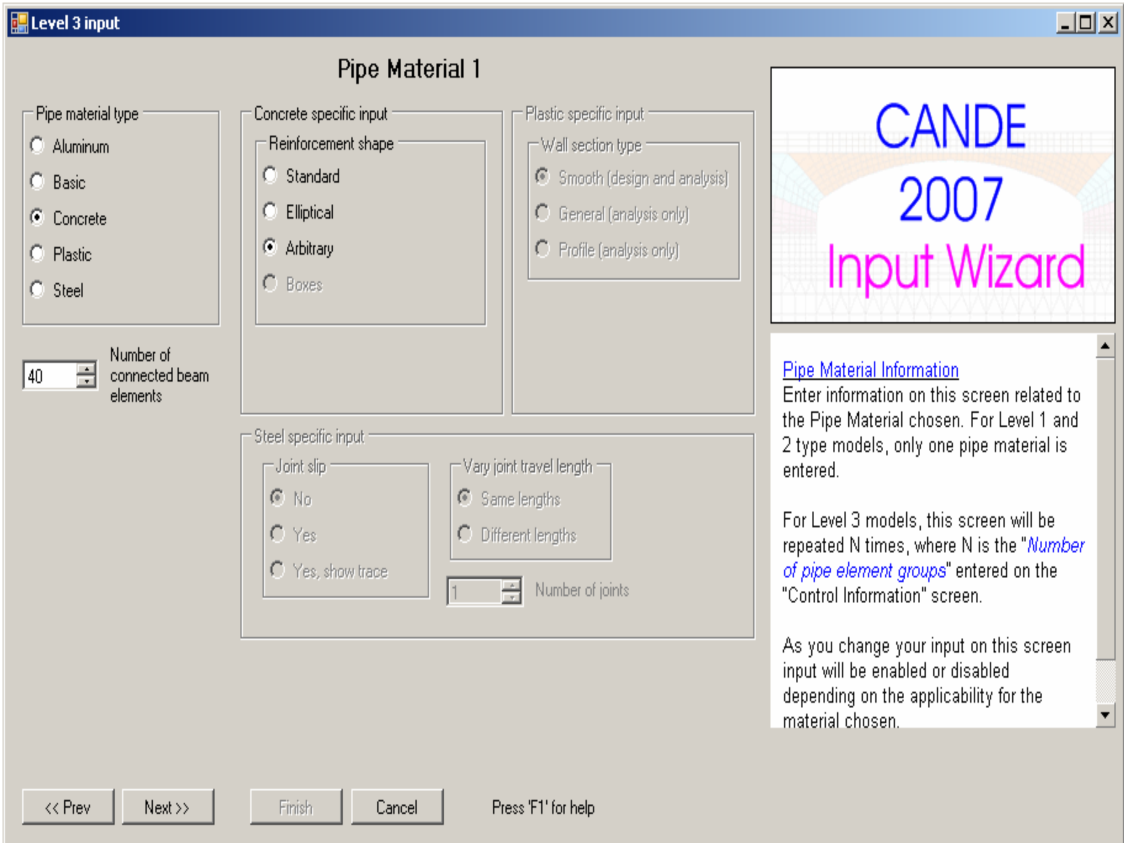


Figure 14-4 – Input Wizard, Screen 3

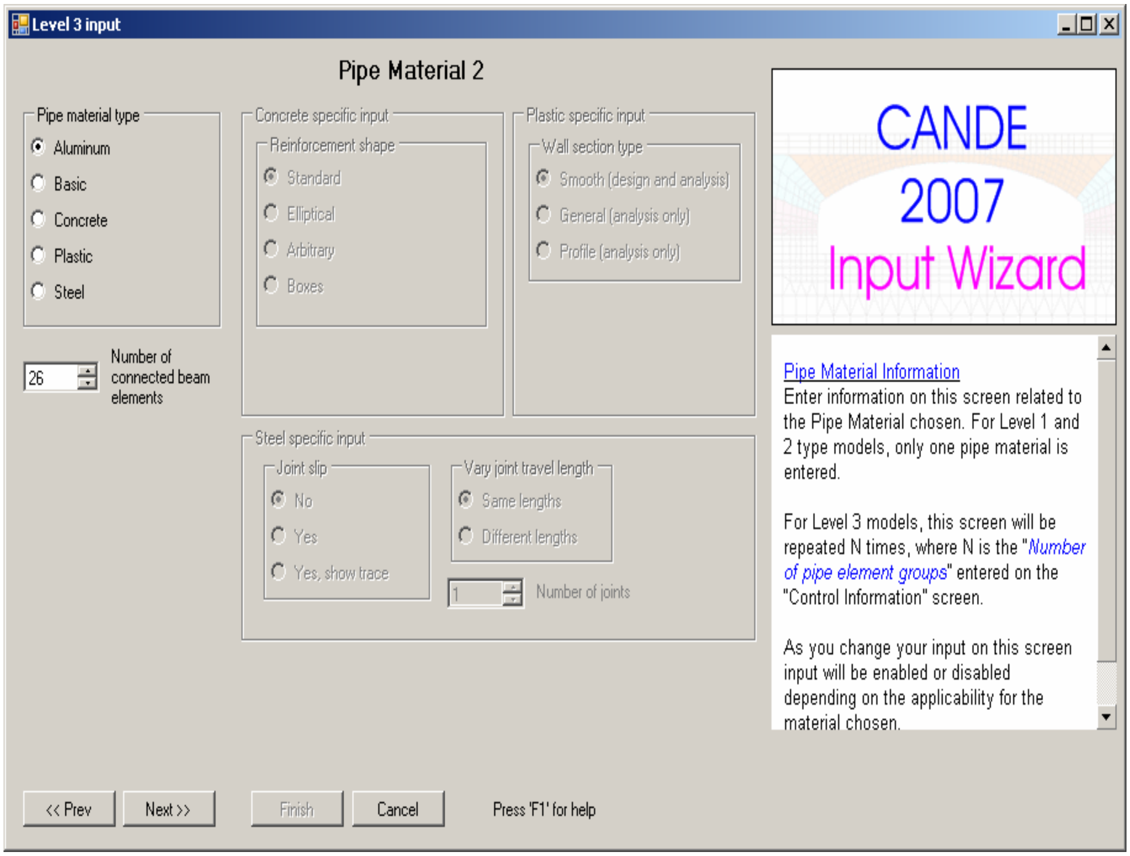


Figure 14-5 – Input Wizard, Screen 4

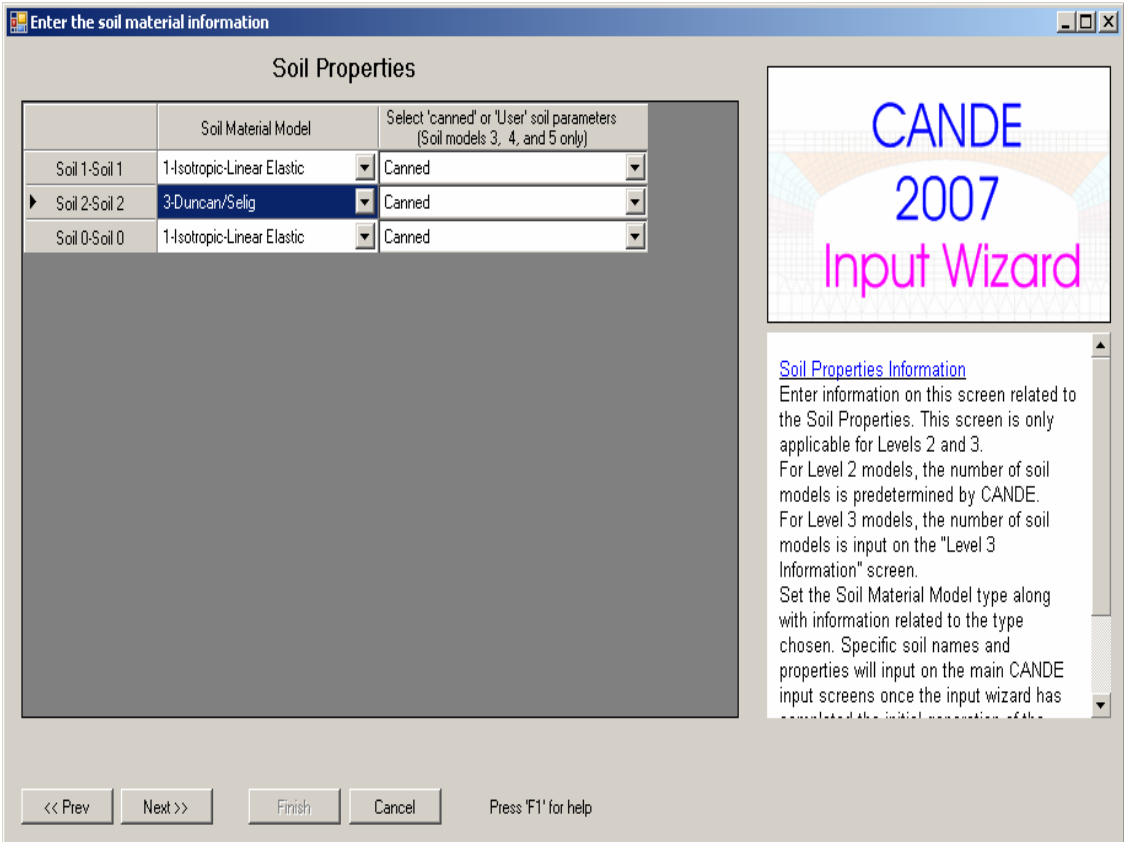


Figure 14-6 – Input Wizard, Screen 5

The Input Wizard then creates an input file and opens CANDE Input screen A-1, Figure 14-7. Enter an appropriate heading for output and click "Accept Input."

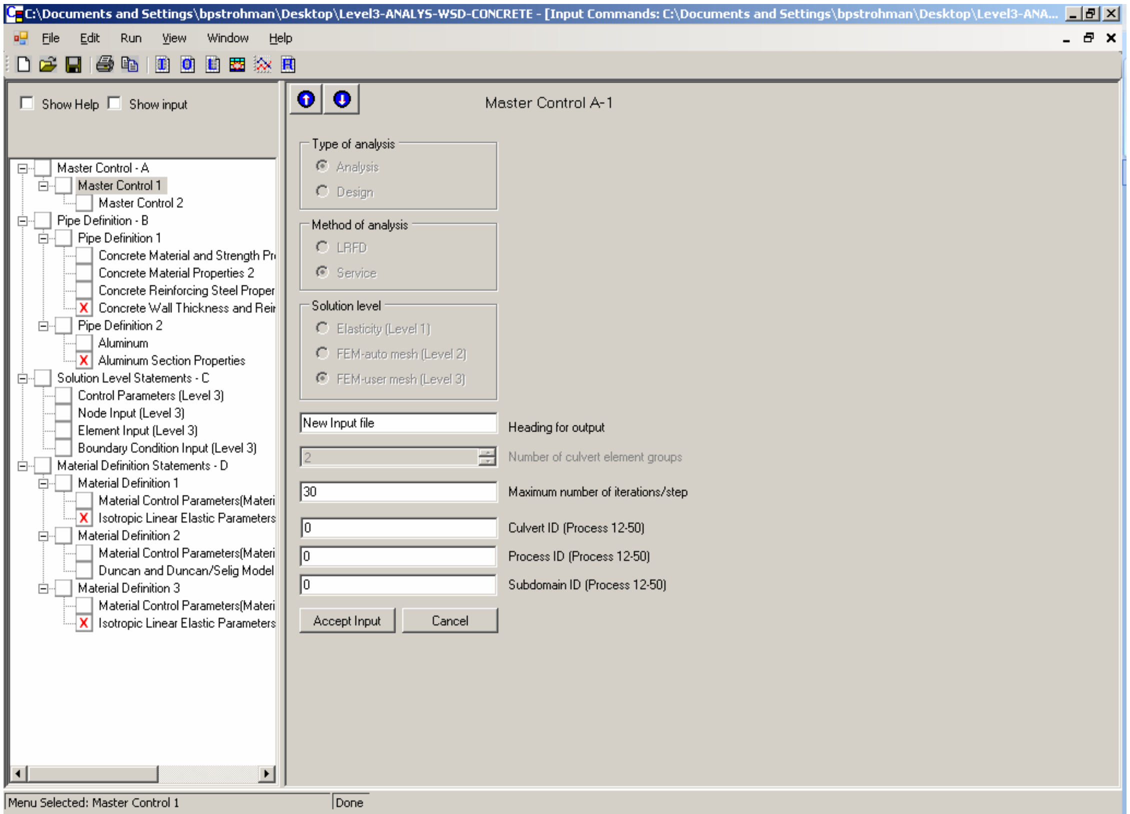


Figure 14-7 – Master Control Screen as Set Up by Input Wizard

The control panel on the left of Figure 14-7 shows "X" for all the screens that require additional input for which no default is provided. In actual design situations engineers should review each screen to determine that the default values are appropriate as many of the defaults have a significant impact on the final design. Figures 14-8 through 14-28 show the completed input for the screens requiring data for the tutorial. Input the remaining definition screens. As input is modified, a 'blue circle' icon will appear in the menu input tree. This is a visual indicator that some input on that menu has been modified. After completing each screen, click "Accept Input" and the blue circle in the control panel will disappear indicating all fields have data.

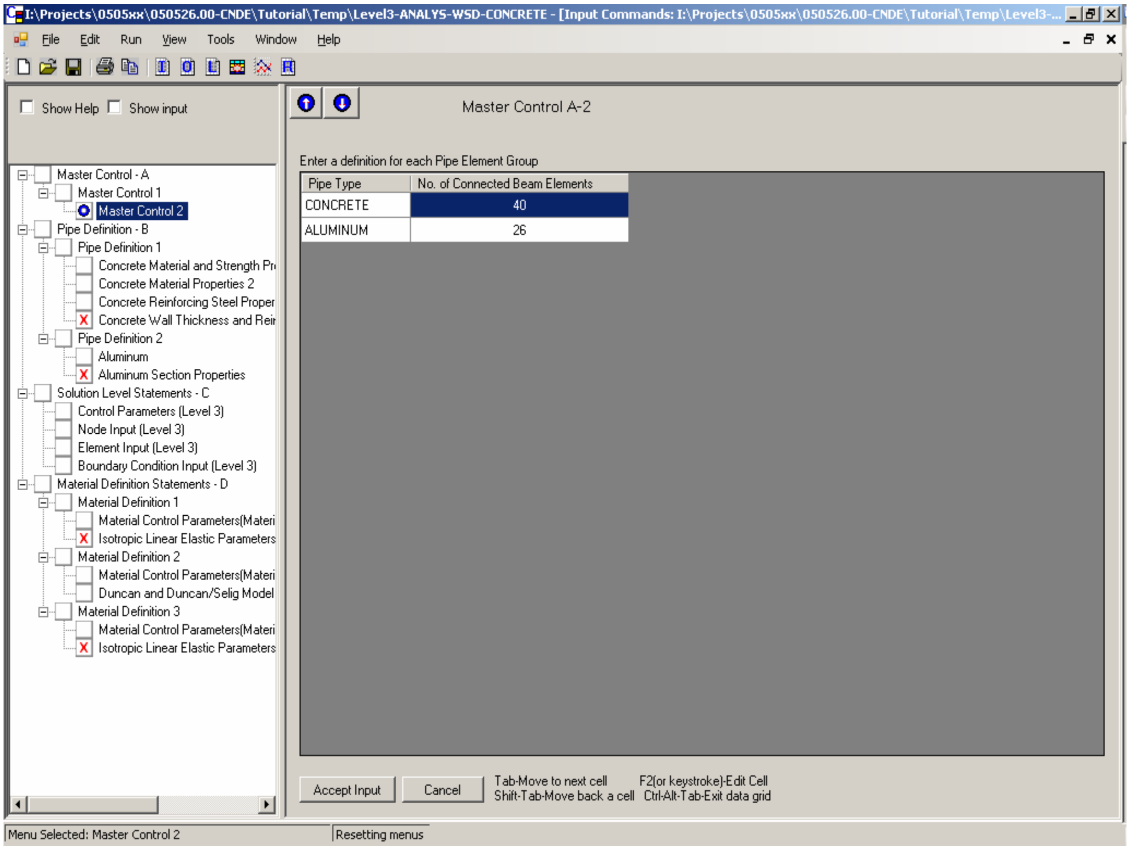


Figure 14-8 – Input Screen A-2

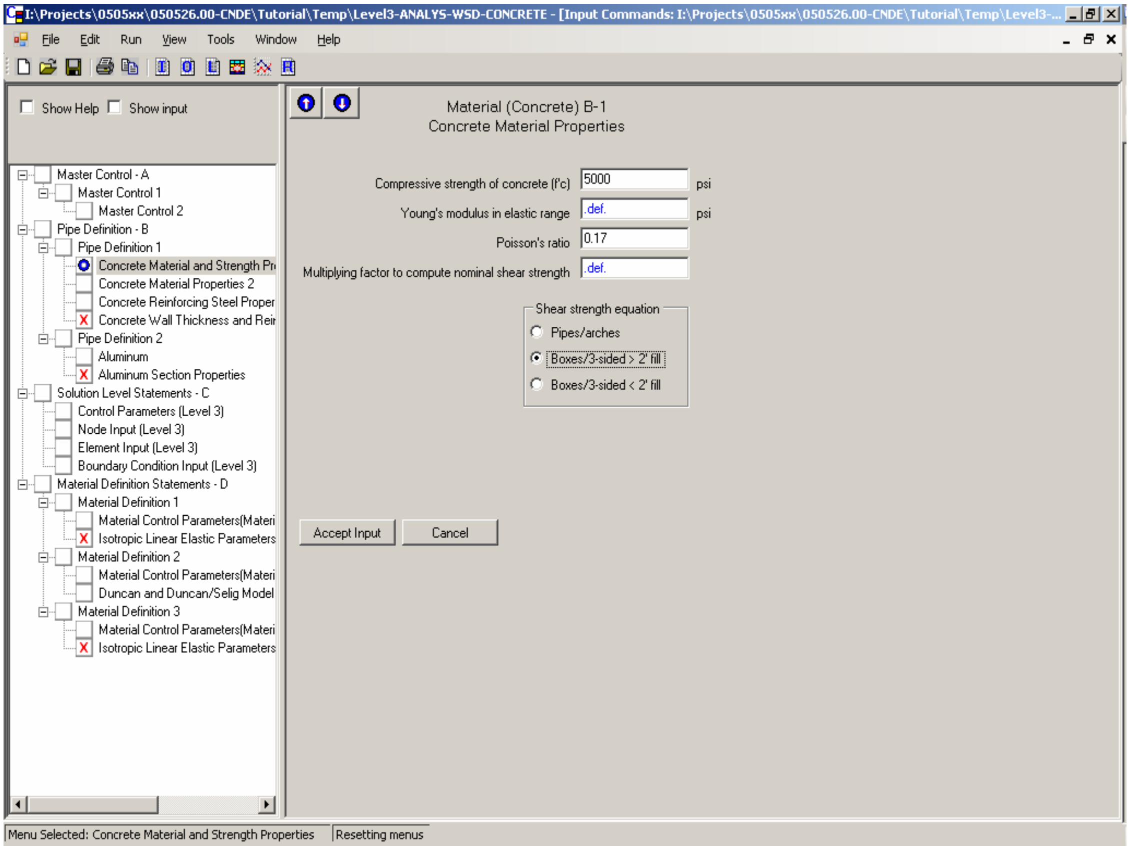


Figure 14-9 – Input Screen B-1: Pipe Material 1 - Concrete

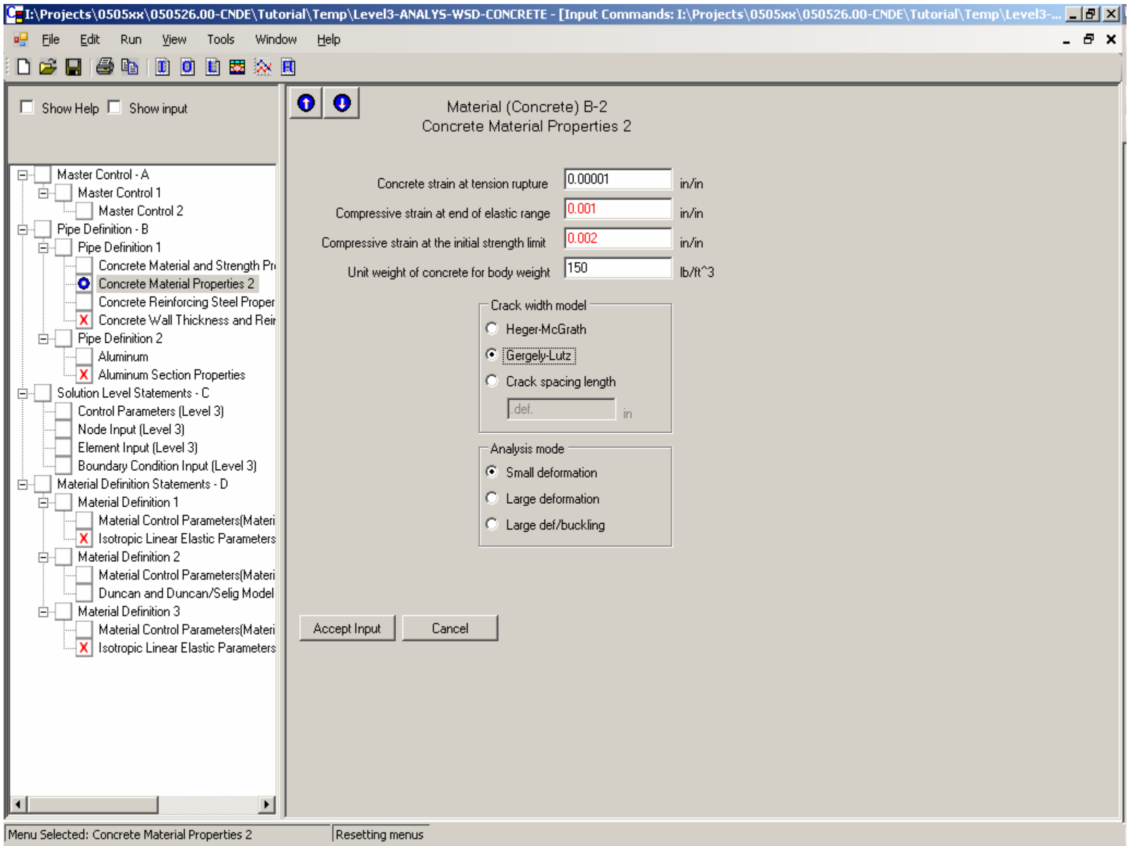


Figure 14-10 – Input Screen B-2: Pipe Material 1 - Concrete

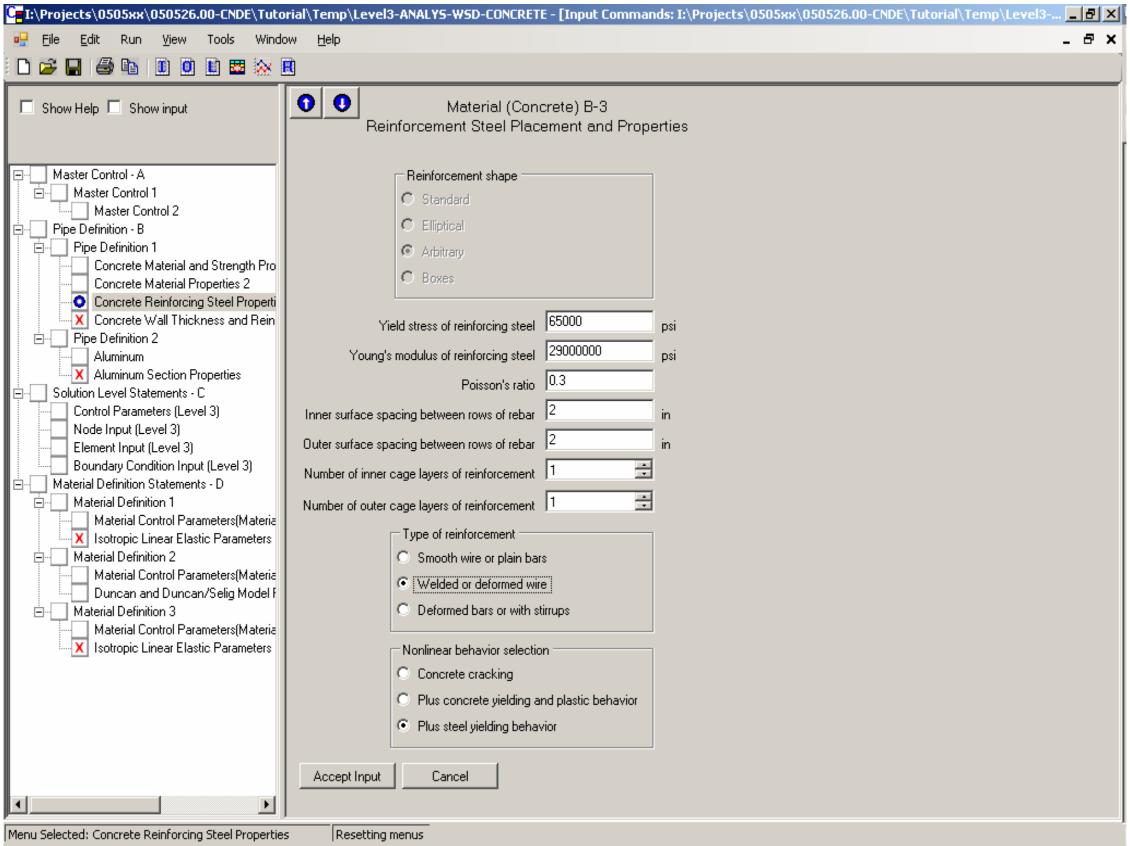


Figure 14-11 – Input Screen B-3: Pipe Material 1 - Concrete

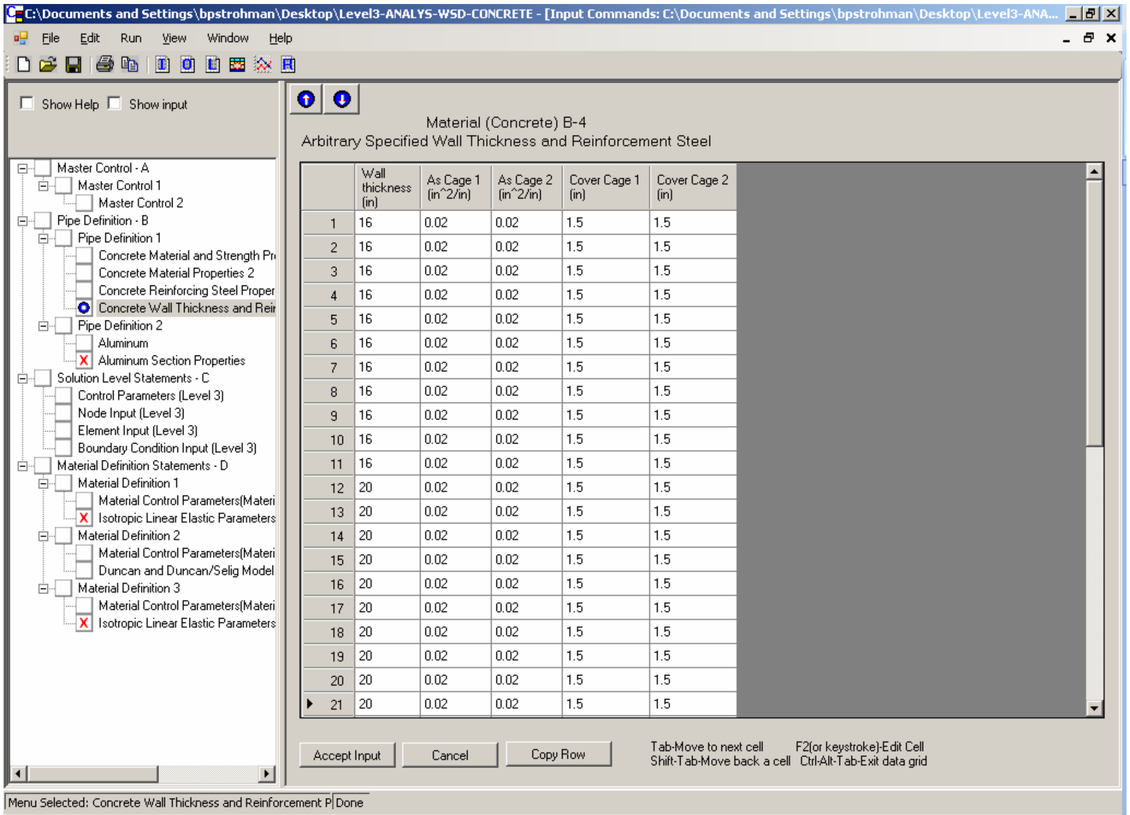
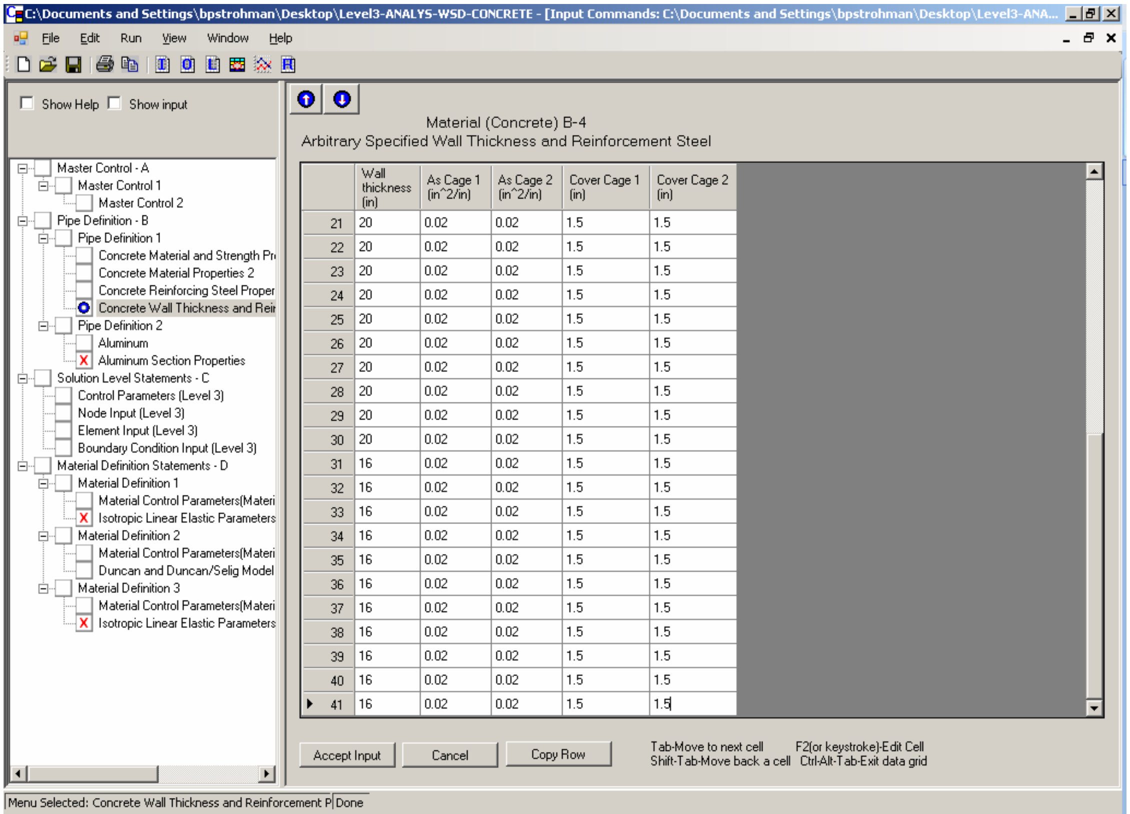


Figure 14-12 – Input Screen B-4: Pipe Material 1 - Concrete

Enter the material information in the grid from the table provided in the 'Problem Definition'. To expedite the entry, fill in the first row, use the 'Copy Row' button to rows 2-41, and modify the wall thicknesses and inner/outer cage areas that are different.



	Wall thickness (in)	As Cage 1 (in^2/in)	As Cage 2 (in^2/in)	Cover Cage 1 (in)	Cover Cage 2 (in)
21	20	0.02	0.02	1.5	1.5
22	20	0.02	0.02	1.5	1.5
23	20	0.02	0.02	1.5	1.5
24	20	0.02	0.02	1.5	1.5
25	20	0.02	0.02	1.5	1.5
26	20	0.02	0.02	1.5	1.5
27	20	0.02	0.02	1.5	1.5
28	20	0.02	0.02	1.5	1.5
29	20	0.02	0.02	1.5	1.5
30	20	0.02	0.02	1.5	1.5
31	16	0.02	0.02	1.5	1.5
32	16	0.02	0.02	1.5	1.5
33	16	0.02	0.02	1.5	1.5
34	16	0.02	0.02	1.5	1.5
35	16	0.02	0.02	1.5	1.5
36	16	0.02	0.02	1.5	1.5
37	16	0.02	0.02	1.5	1.5
38	16	0.02	0.02	1.5	1.5
39	16	0.02	0.02	1.5	1.5
40	16	0.02	0.02	1.5	1.5
41	16	0.02	0.02	1.5	1.5

Figure 14-13 – Input Screen B-4 (continued) : Pipe Material 1 - Concrete

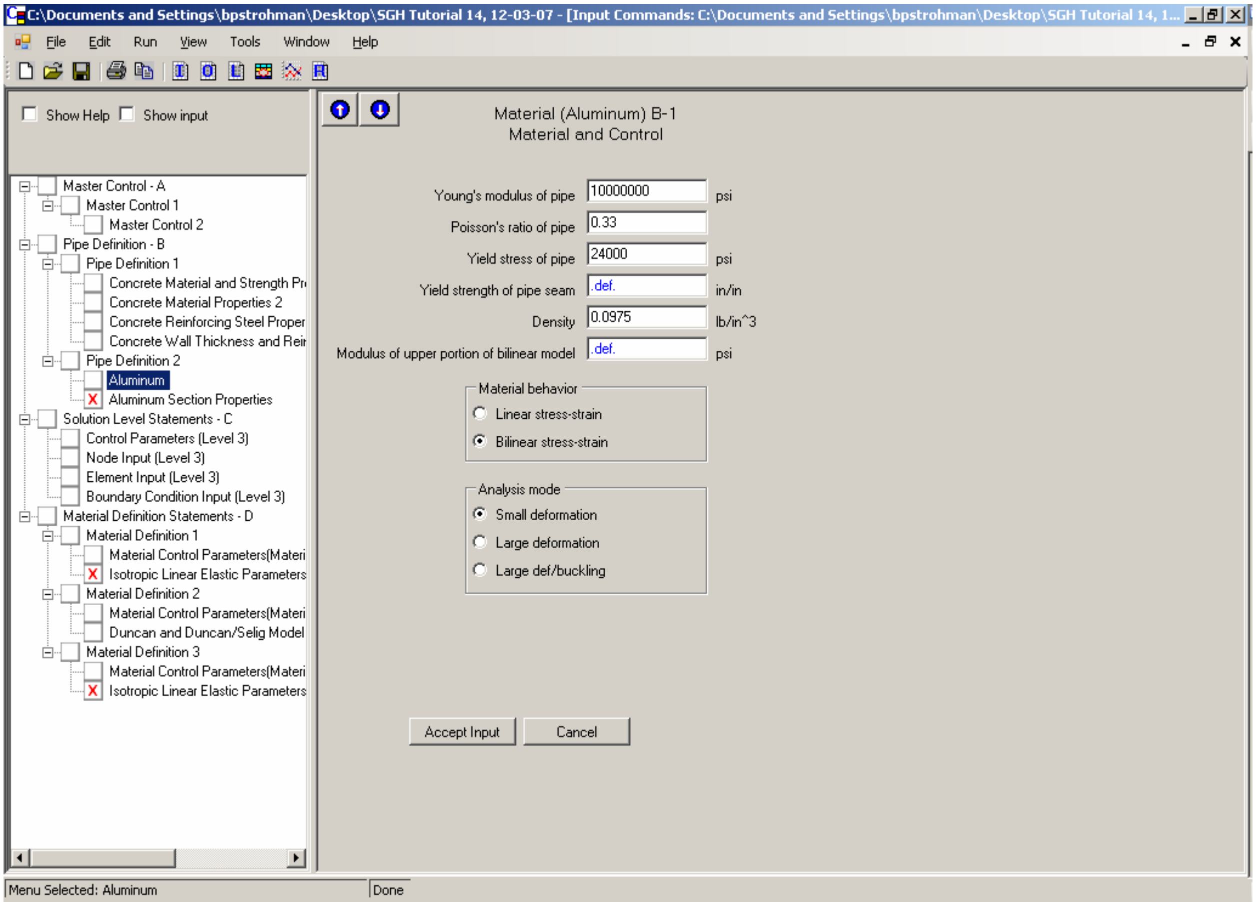


Figure 14-14 – Input Screen B-1: Pipe Material 2 - Aluminum

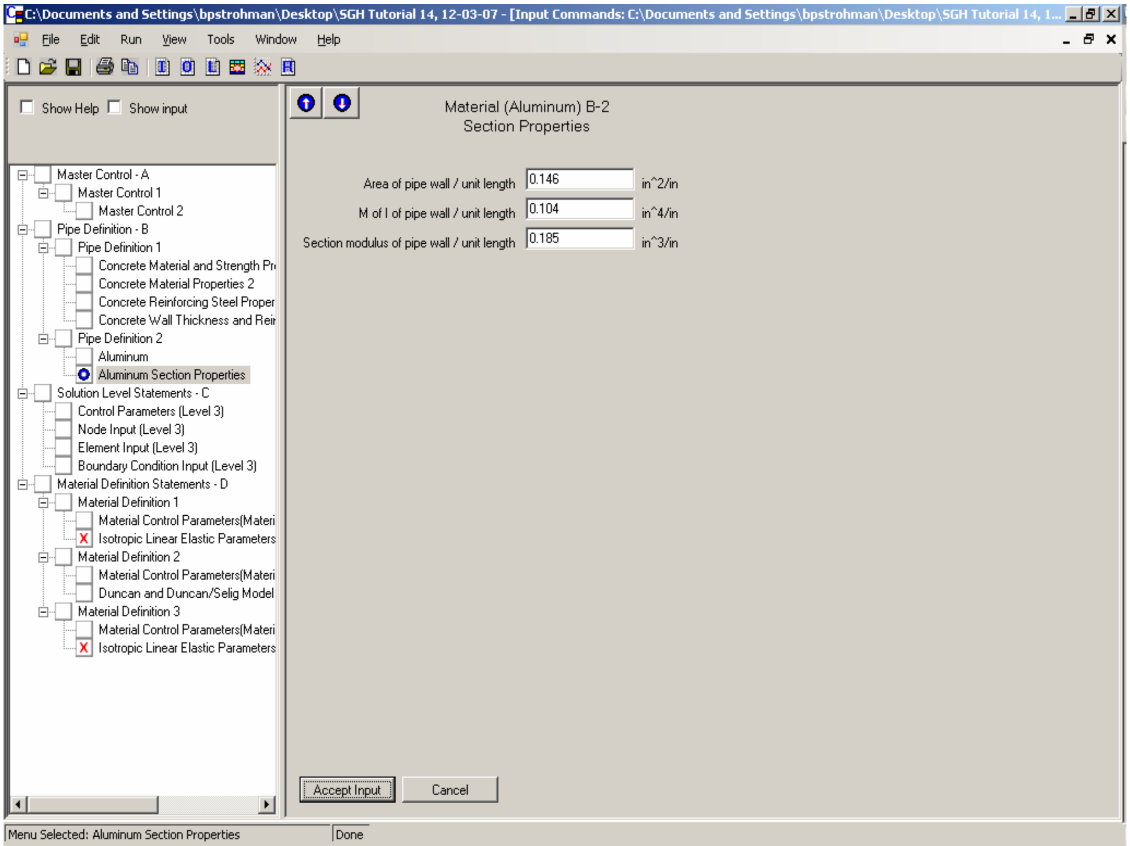


Figure 14-15 – Input Screen B-2: Pipe Material 2 - Aluminum

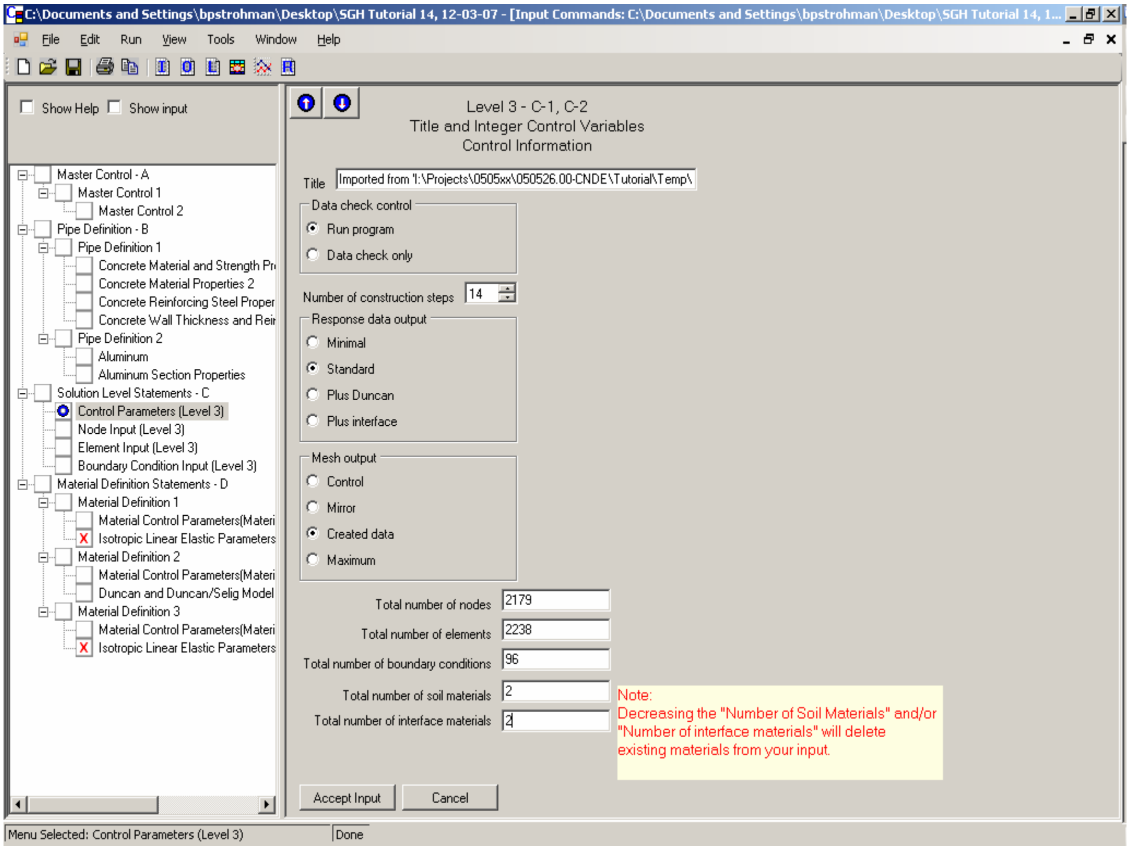


Figure 14-16 – Input Screen C-1, C-2

Boundary conditions are not listed in the xml mesh file provided for this example. Thus, boundary conditions must be added manually. Since this is a user-defined Level 3 analysis, node numbers are not predefined in the *User Manual.* The user must input the remaining standard input parameters (Figures 14-17 and 14-18) to define the problem and then use the "View/Mesh Plot" to determine which nodes necessitate boundary conditions. This is demonstrated in Figure 14-27.

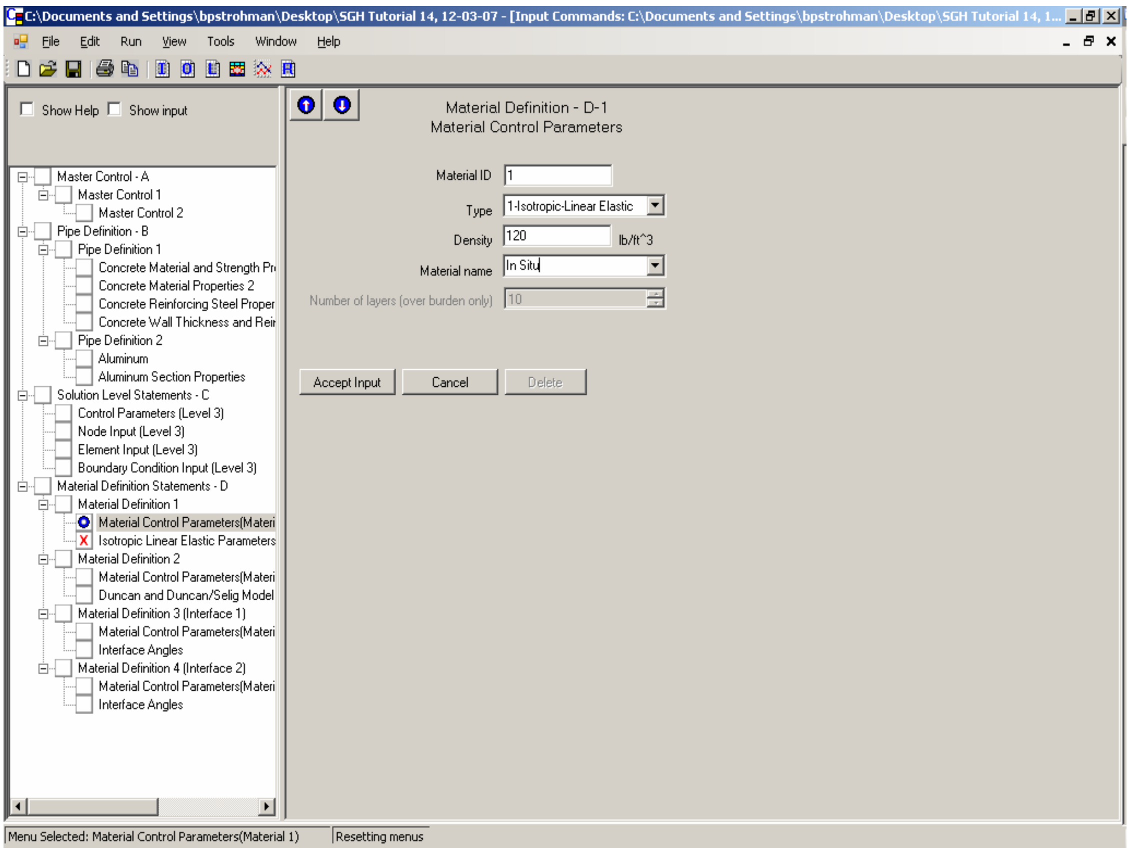


Figure 14-17 – Input Screen D-1 for Material 1
(Note: Repeat for Material 2 with values provided in the 'Problem Definition')

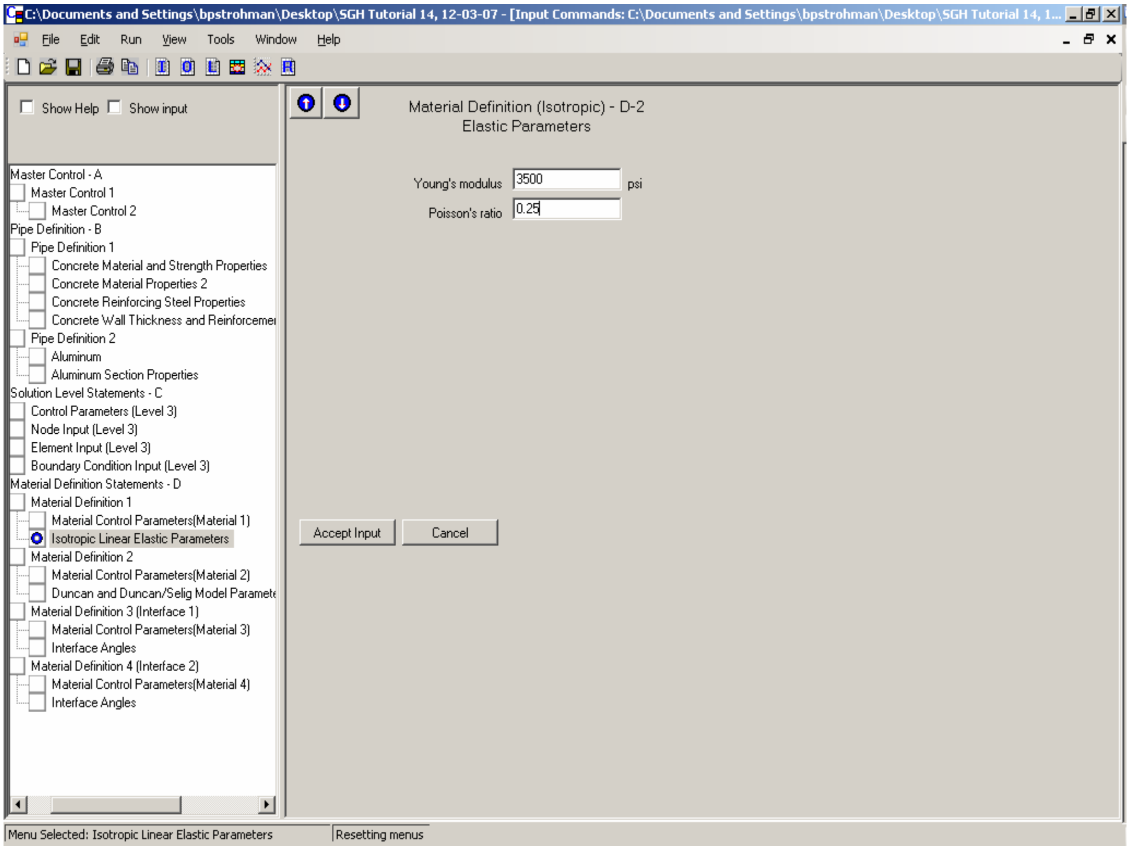


Figure 14-18 – Input Screen D-2 for Material 1
(Note: Repeat for Material 2 with values provided in the 'Problem Definition')

Similar to the boundary conditions, the user must use the “View/Mesh Plot” to determine which nodes define the interface elements. Figures 14-19 and 14-20 display in the input screens for the interface property definition.

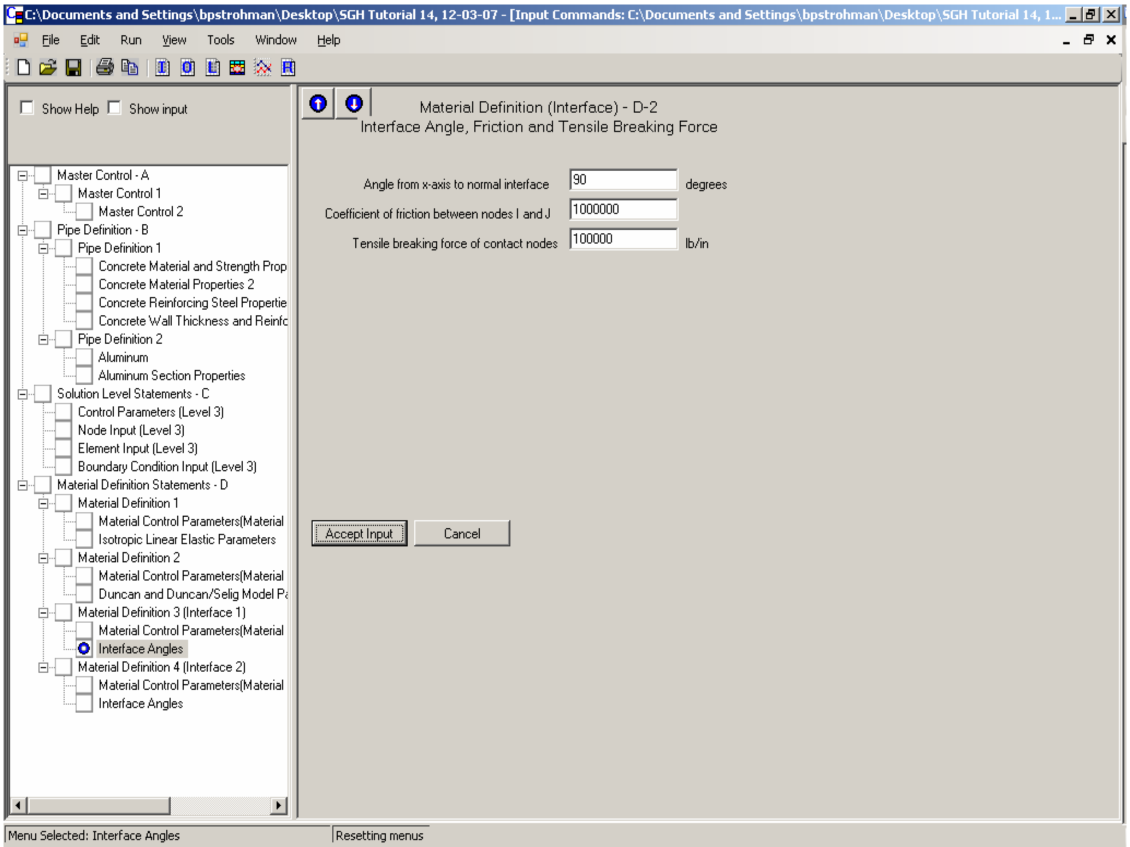


Figure 14-19 – Input Screen D-2 for Material 3

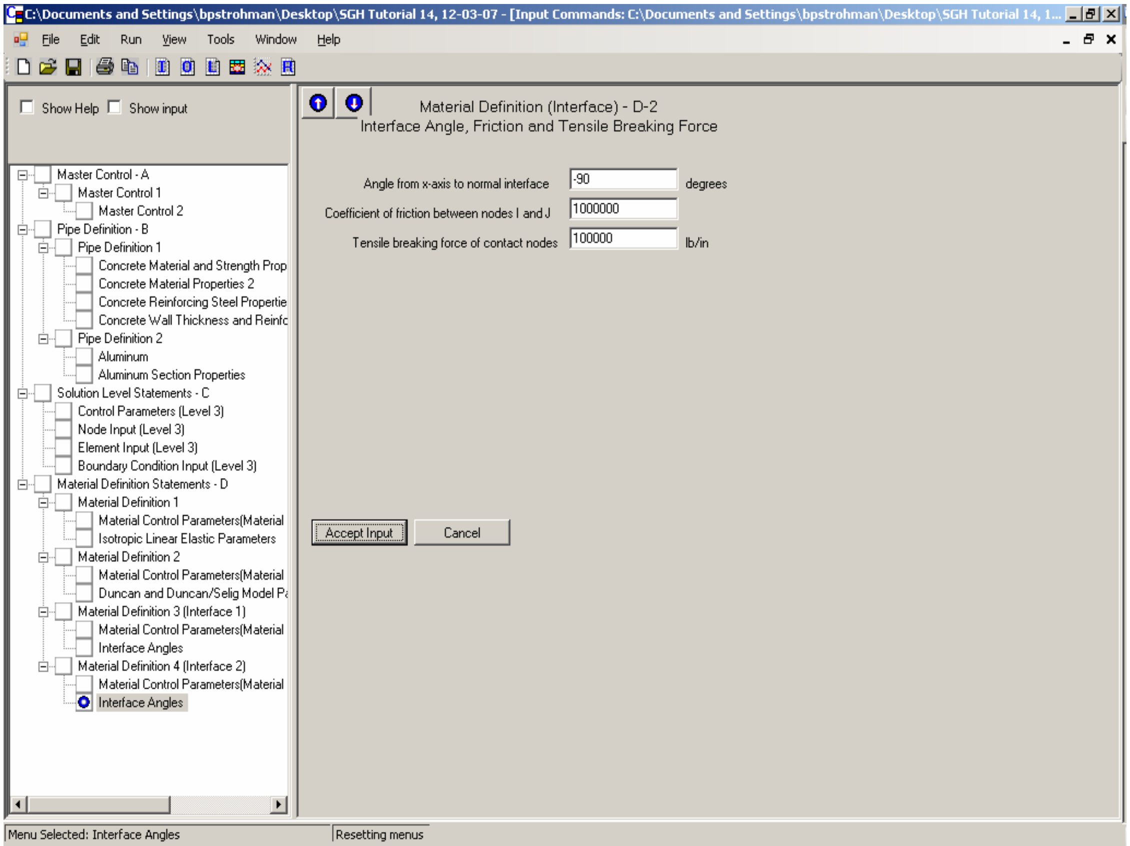


Figure 14-20 – Input Screen D-2 for Material 4

At this point, the user must run the mesh to determine which nodes necessitate boundary conditions and interface elements. When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

Now proceed to the mesh plot. Click “View” and then “Mesh Plot” to open the mesh plot. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot.

Figure 14-21 shows a mesh plot displaying the node numbers of the elements activated in load step 1. Horizontal roller boundary conditions need to be applied on the side boundaries of the model and vertical pin boundary conditions on the bottom boundary as depicted in Figure 14-21. Node numbers for these boundary conditions are shown on the mesh plot.

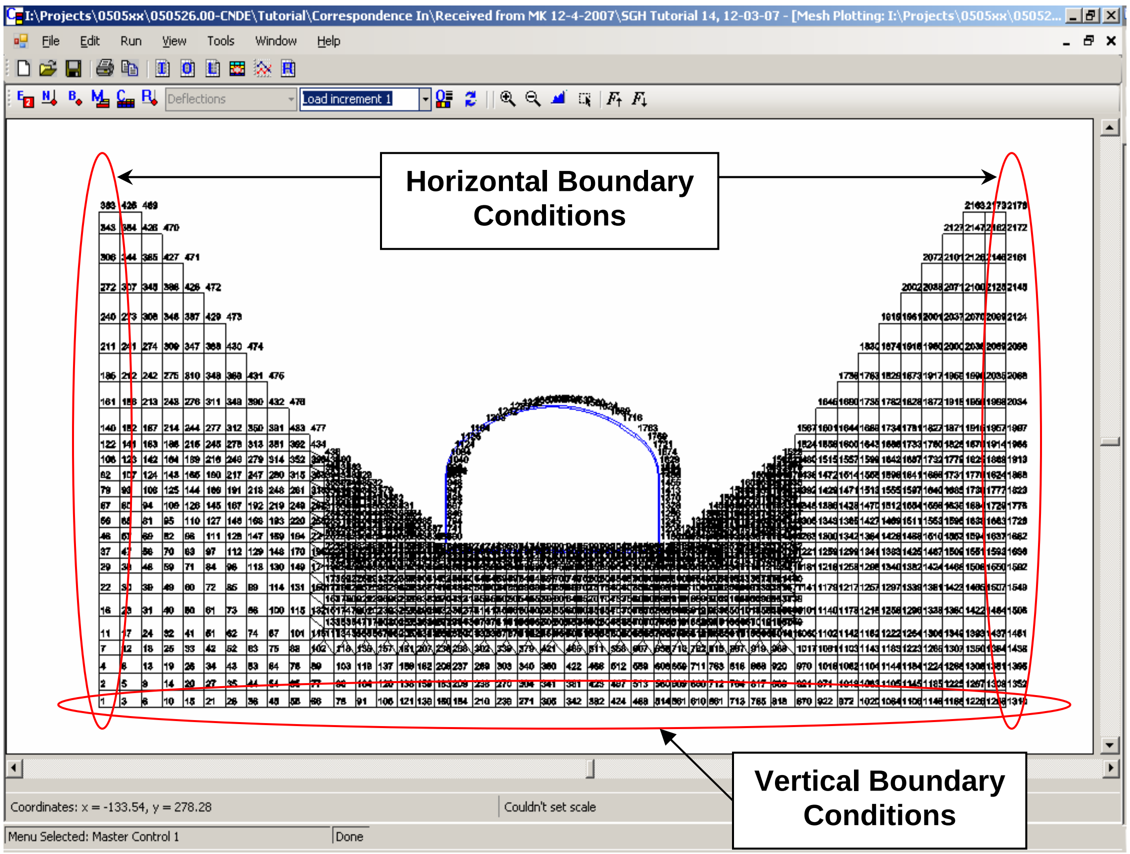


Figure 14-21 – Mesh Plot for Load Step 1, Used to Determine Nodes for Boundary Conditions

Figure 14-22 shows a close-up of Figure 14-21 to display the node numbers for the interface elements. Node numbers for the interface elements are shown on the mesh plot.

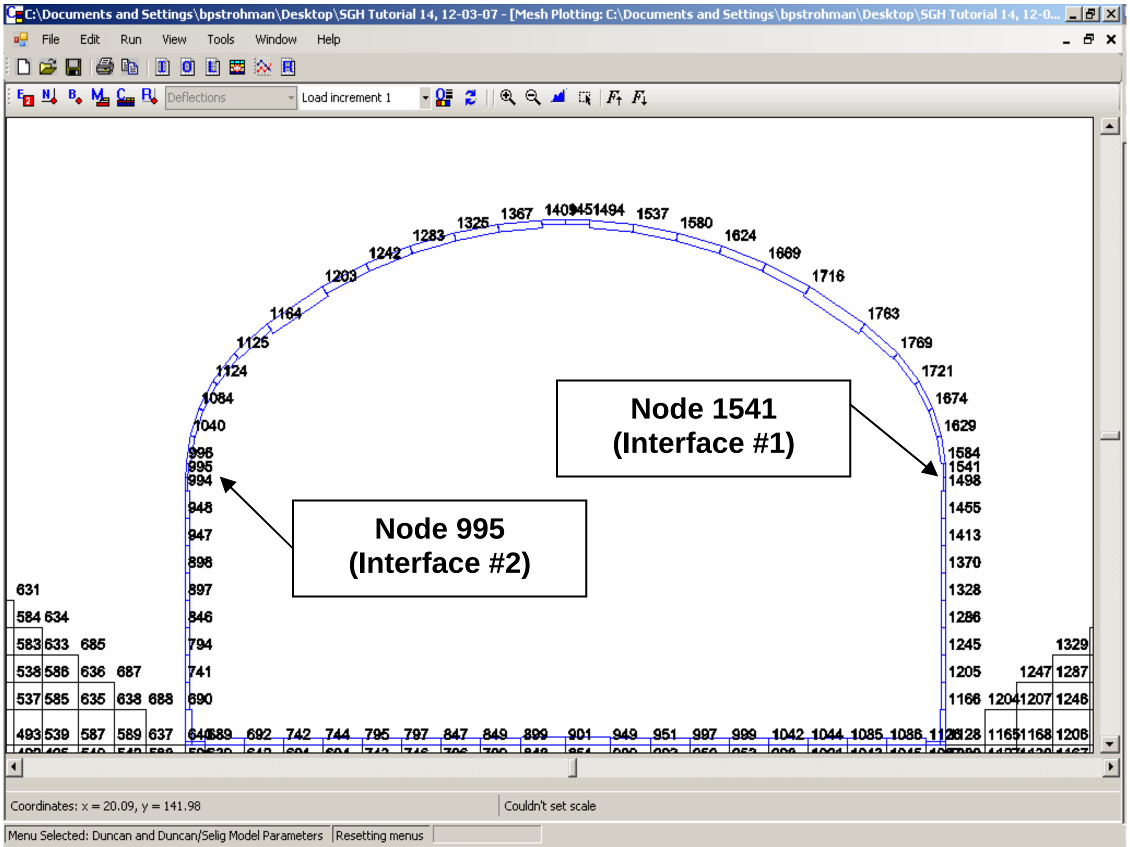


Figure 14-22 – Close-up of Mesh Plot for Load Step 1, Used to Determine Nodes for Interface Elements

Now proceed to the input screen ‘Control Parameters (C-1, C-2)’. In CANDE, interface elements are defined by three nodes with the same coordinates. In this example, one node defines the concrete, one node defines the aluminum, and one node defines the interface between the two nodes. As a result, 2 new nodes need to be defined with the same coordinates as nodes 995 and 1541. This increases the ‘Total Number of Nodes’ to 2,183 and the ‘Total Number of Elements’ to 2,240. Furthermore, based on Figure 14-21, the ‘Total Number of Boundary Conditions’ is 92. Figure 14-23 displays an updated C-1, C-2 input screen.

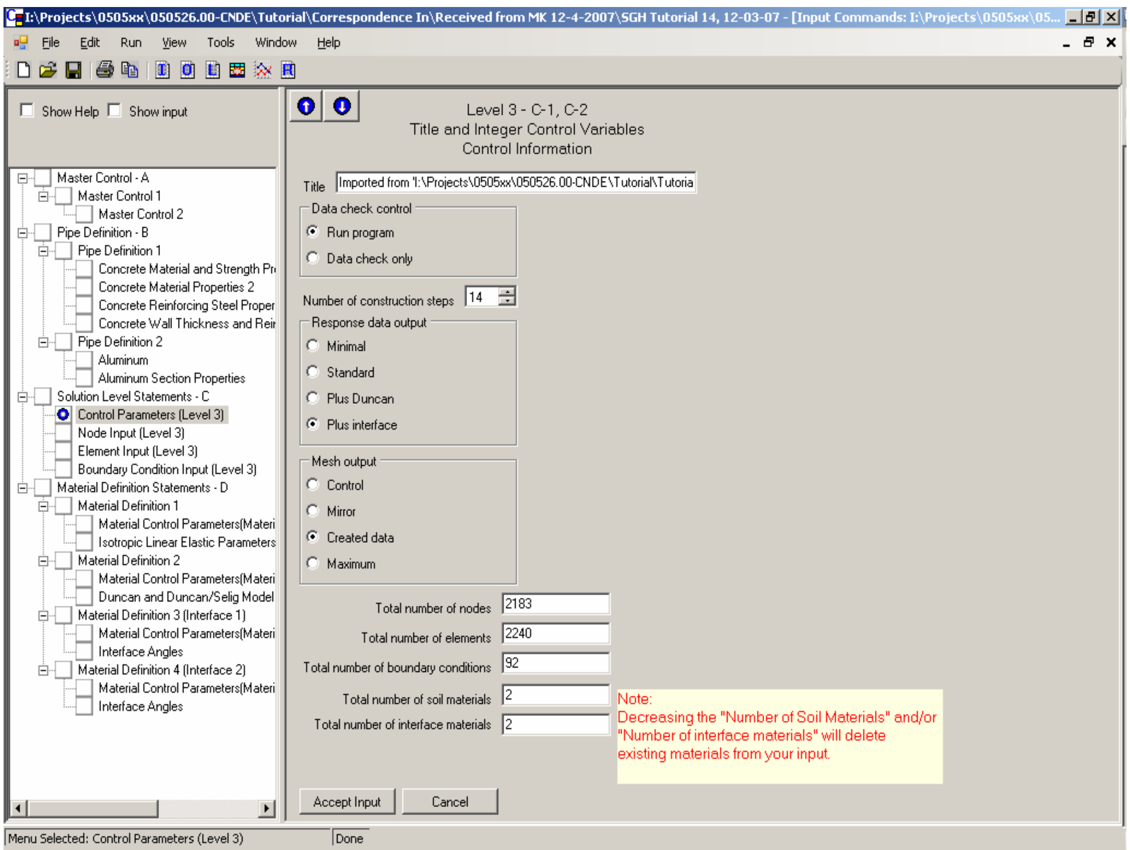


Figure 14-23 – Updated Input Screen C-1, C-2

Now proceed to the inputs screen 'Node Input (C-3)'. Find the coordinates of nodes 995 and 1541. The coordinates for node 995 are (-90, 60.83) and node 1541 are (90, 60.83). Input coordinates for 2 new nodes (2180, 2181) with the coordinates of node 995 and 2 new nodes (2182, 2183) with the coordinates of node 1541. Figure 14-24 displays the last twenty lines of an updated C-3 input screen.

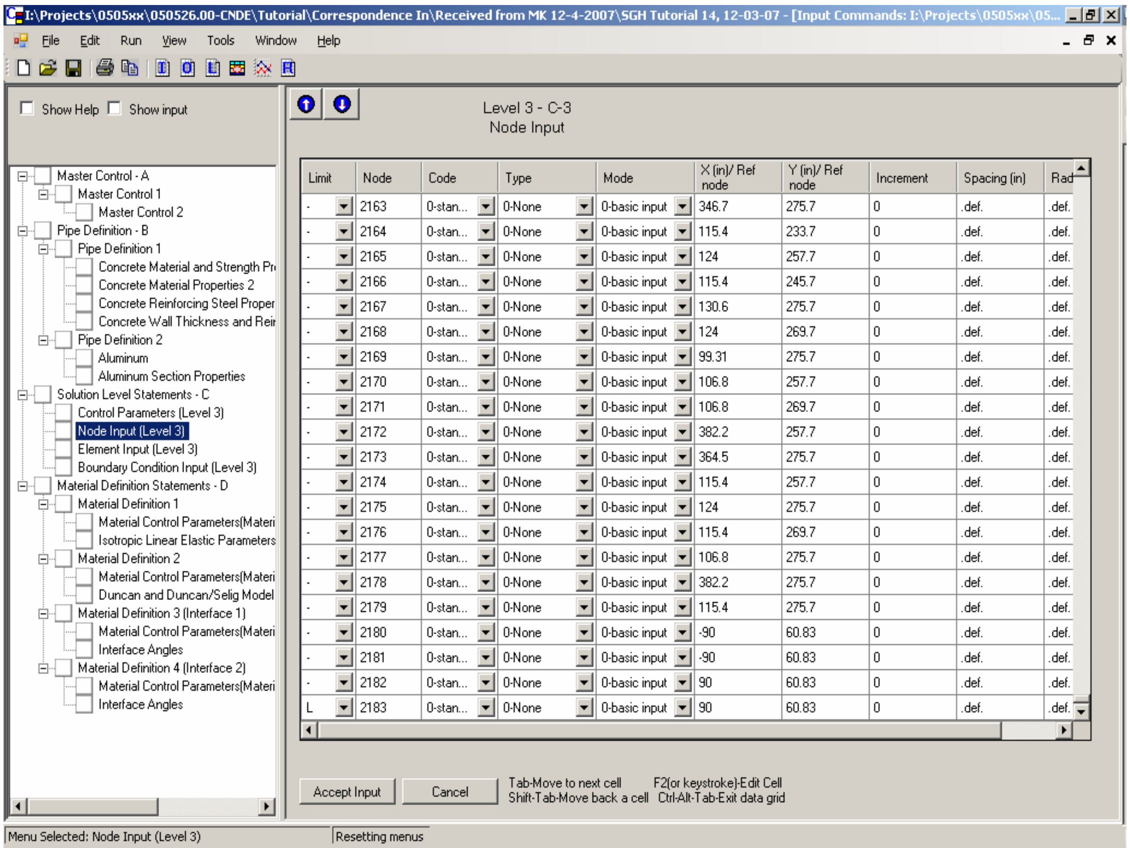


Figure 14-24 – Updated Input Screen C-3

Now proceed to the input screen ‘Element Input (C-4)’. Elements 2173 to 2238 define the beam elements for this problem. For element 2212, change node I from 995 to 2181 and for element 2238, change node I from 1541 to 2183. This changes the connectivity of the beam elements at the point where the two materials join from aluminum-concrete to aluminum-interface-concrete. Input interface element connectivity for two elements (2239, 2240). For element 2239, define node I = 1541, node J = 2183, and node K = 2182. Define element 2240 in the same manner, node I = 995, node J = 2181, and node K = 2180. For elements, 2239 and 2240, the interface material I.D. is 1 and 2 respectively. For additional information regarding defining interface elements and sign convention, see Chapter 5, C-4 of the *User Manual.* Figures 14-25 and 14-26 display the last forty lines of an updated C-4 input screen.

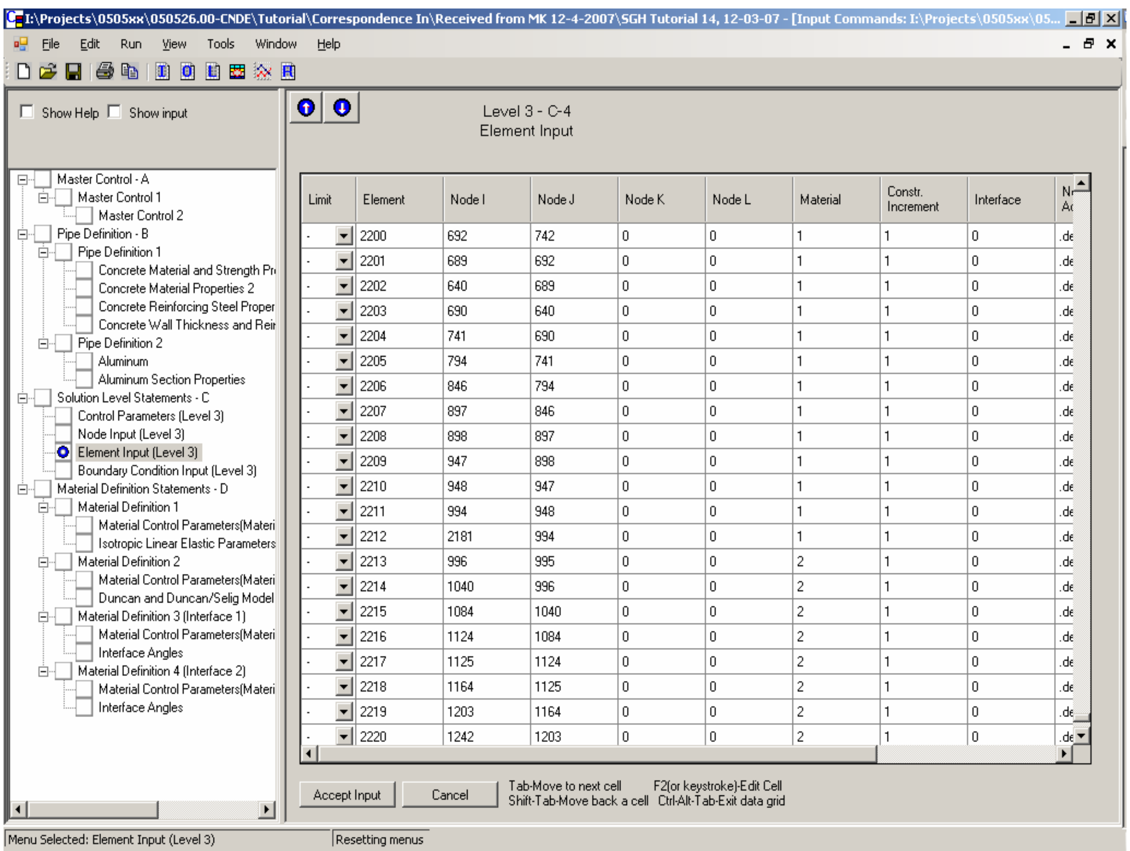


Figure 14-25 – Updated Input Screen C-4

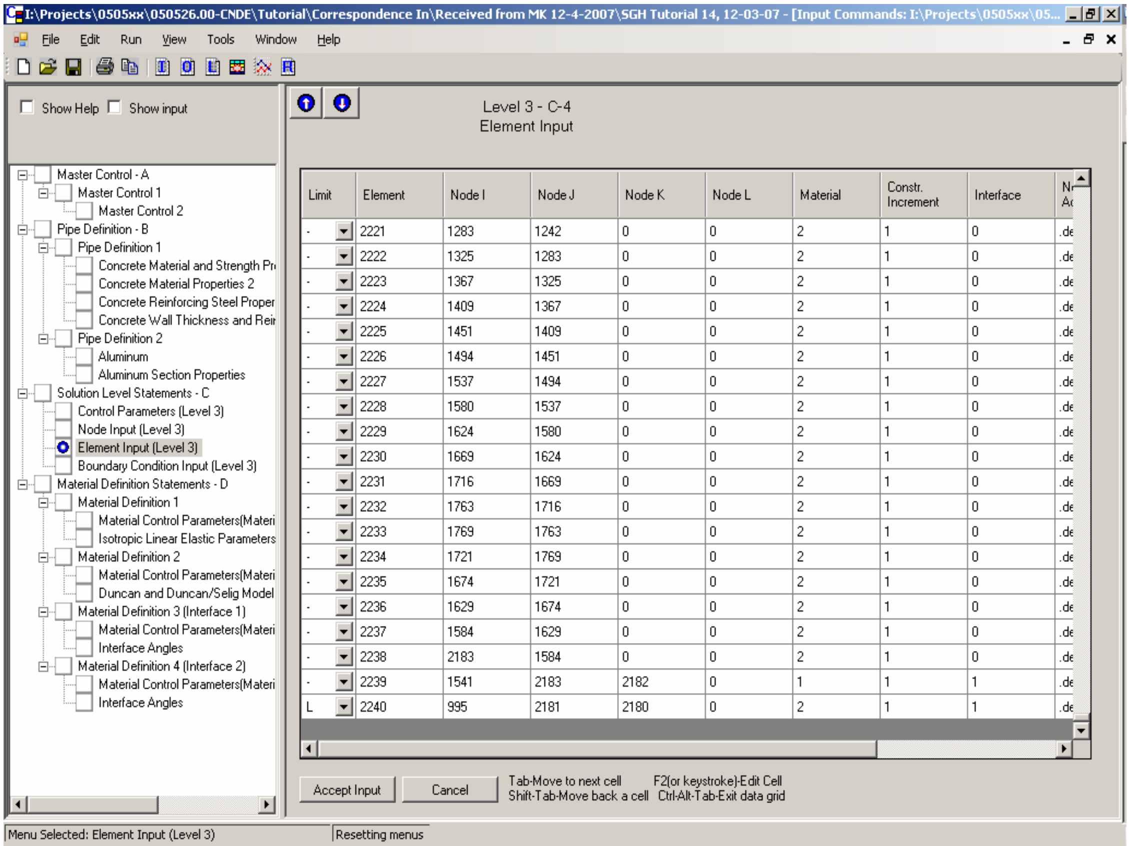


Figure 14-26 – Updated Input Screen C-4 (continued)

Now proceed to the input screen ‘Boundary Condition Input (C-5)’. Input boundary condition nodes displayed in Mesh Plot (Figure 14-21). Figures 14-27 and 14-28 display the first and last twenty lines, respectively of an updated C-5 input screen.

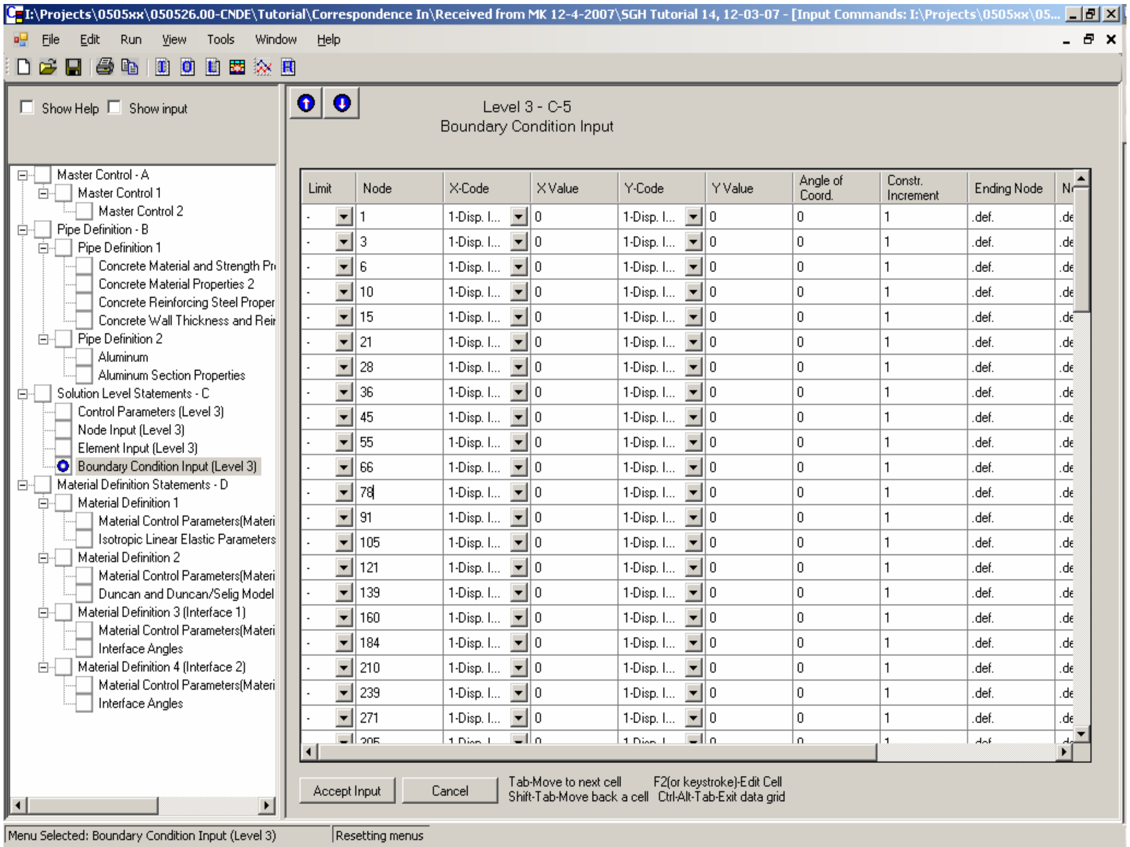


Figure 14-27 – Updated Input Screen C-5

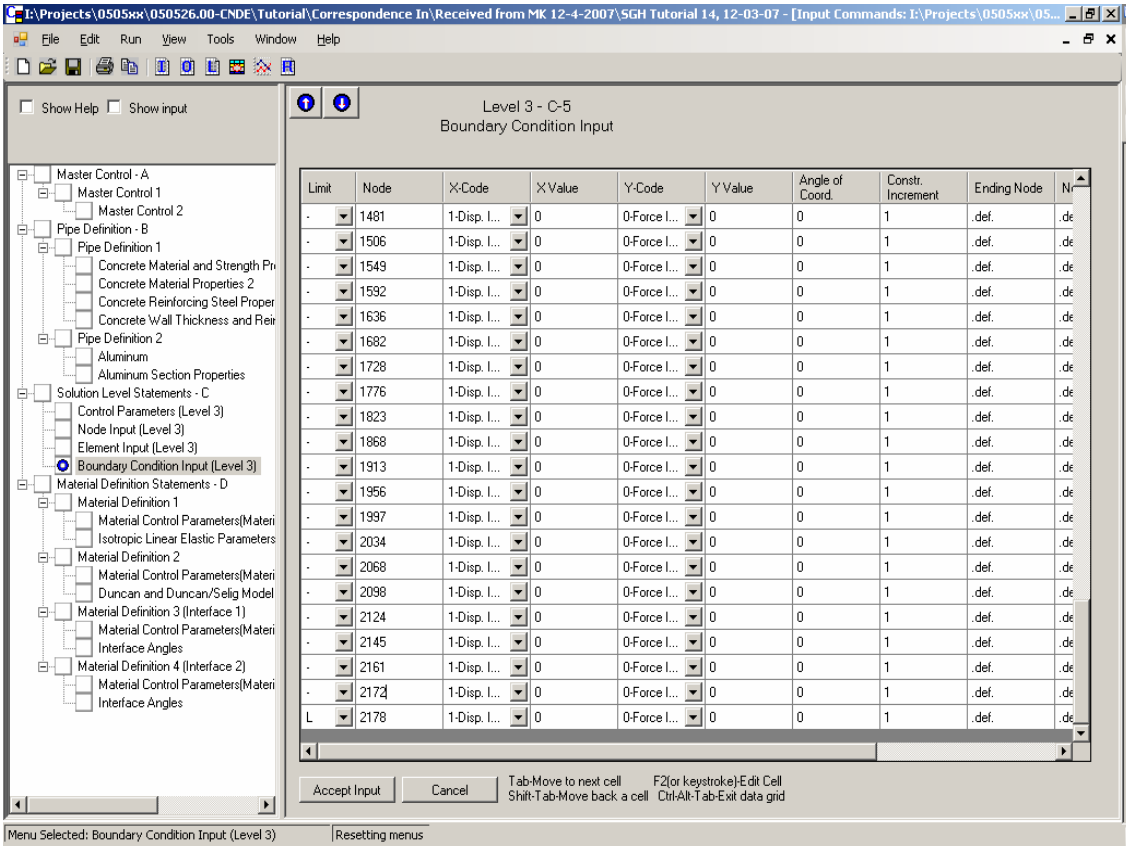


Figure 14-28 – Updated Input Screen C-5 (continued)

When all input is complete and saved, click “Run” and “CANDE-2007” on the main toolbar to execute the program. This will open the “Running CANDE” window which will allow you to monitor the run as it progresses. When the program is complete, a check box “CANDE Analysis Complete” window will appear. Click “OK” and check if the “Running CANDE” window states “Normal Exit From CANDE.” If an error has occurred, the program will identify the input line of the error. Reopen the input file and review for mistakes – using the find option may assist in locating the problem as the line numbers correspond to the CANDE error message. Click the “Close” button in the “Running CANDE” window to return to the user interface.

14.3 Reviewing and Interpreting the Output

Now proceed to check the output file. Tools to assist in this process are the mesh plot and graphs available under the "View" menu on the main toolbar. Users should explore the use of the buttons on the toolbars of the mesh plot screen. Element and node numbering, material information, boundary conditions, and construction increments may all be added or removed from the plot. The "Plotting Parameters" button allows the user to:

- change the magnification of the deflections,
- eliminate the magnitude of early deflection/stress values - for example the deflections and stresses due to the self weight of the in situ soil may not be of interest, or the user may wish to see only the deflection/stress due to a live load condition, and
- change colors or add increment identifiers.

Begin with the mesh plot. Click "View" and then "Mesh Plot" to open the mesh plot. Open the plotting parameters menu and click the check box "Show constr. increment numbers." Click "OK," set Load Increment to 14 to show the entire mesh and click the toolbar icon to turn on the construction steps. The mesh plot should look like Figure 14-29.

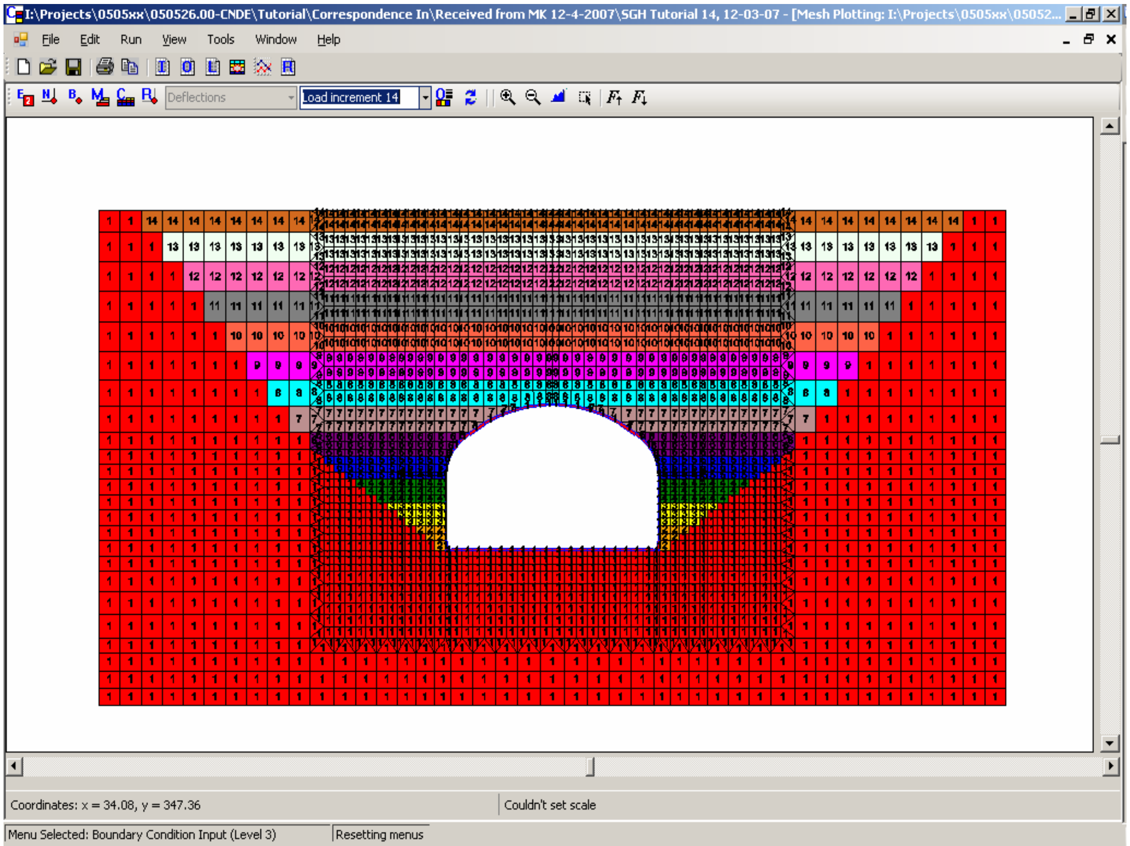


Figure 14-29 – Mesh Plot for Load Steps 1 to 14

To view deflection, open the plotting parameters window and set the deflection magnification factor to 10 (see Figure 14-30).

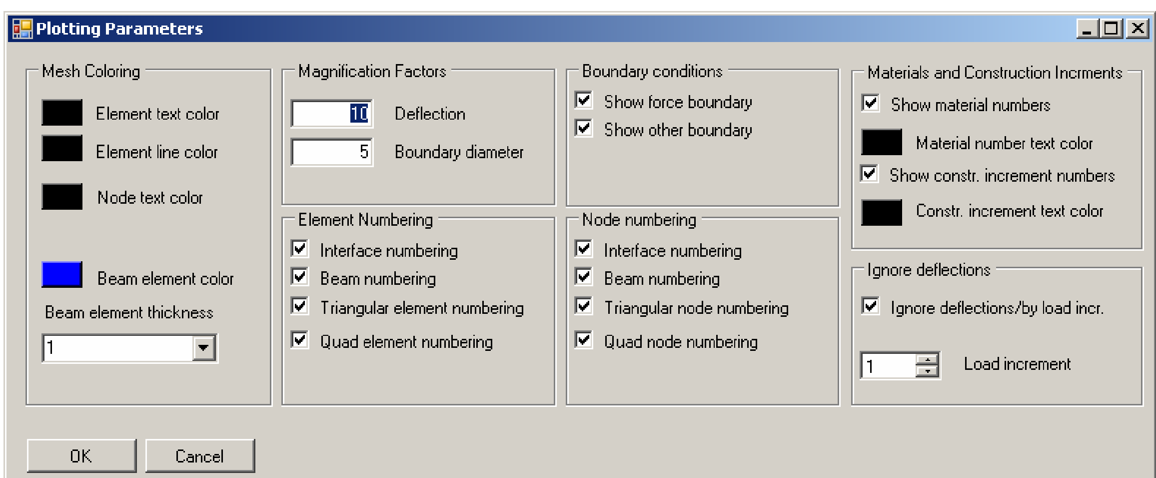


Figure 14-30 – Mesh Window Plotting Parameters

Select "OK" to close the window. Click the icon to "Turn on/off selected output results" and set the drop down box to Deflections. Set the increment to 14 and the screen should look like Figure 14-31. Other mesh output parameters can also be inspected to determine if the results are consistent with the design intent.

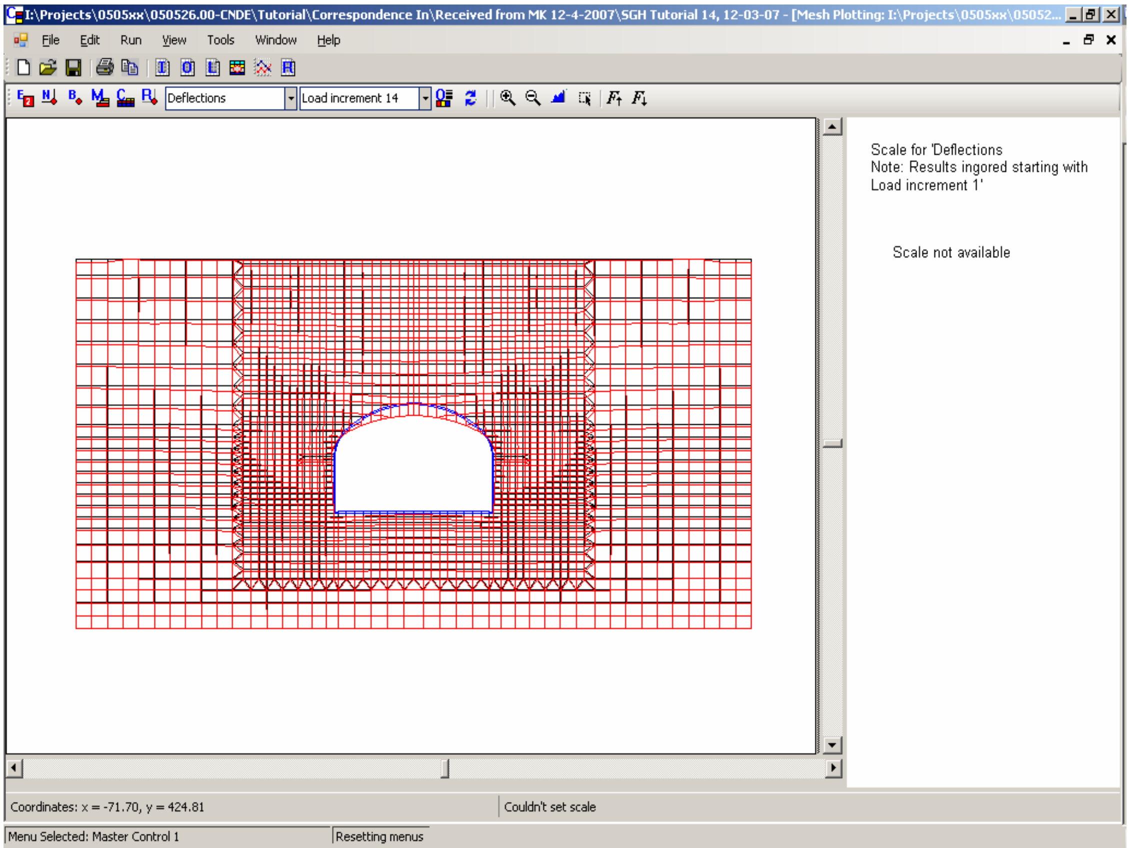


Figure 14-31 – Deflections, Excluding Those Due to Self Weight of In Situ Soil

Similarly, using the "View CANDE Graphs" button on the toolbar or selecting "View/Graphs" the user can explore the forces in the culvert itself. Figure 14-32 compares the bending moments in the reinforced concrete U-shaped base after three increments. This is accomplished using the "Plot multiple load steps" icon at the far left of the toolbar to uncheck "Show single load step" and check Load Steps 5, 10, and 14, then "OK." The "Turn on/off view of pipe" icon on the right side of the toolbar, allows showing the pipe and the node numbering to assist in interpreting the graph. Turn on the view of the pipe and set the drop down box to "Bending Moment" and the screen should appear as Figure 14-32. The beam group drop down box allows the user to view the forces in other 'beam groups' if a problem contains multiple pipe element groups. Figure 14-33 displays the bending moments in the aluminum arch after Load Steps 5, 10, and 14. Note that in both Figures 14-32 and 14-33 the bending moment at the pin connection between the two pipe element groups is 0, as expected.

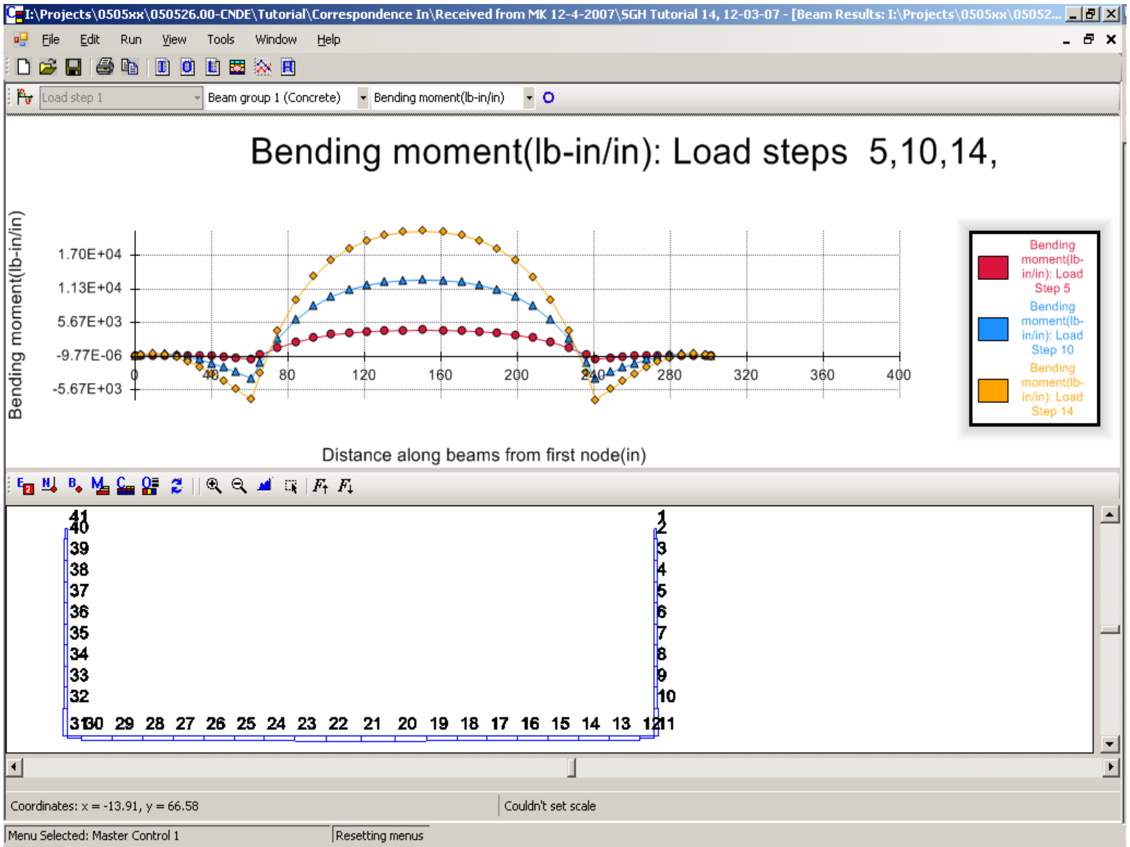


Figure 14-32 – Bending Moment in Reinforced Concrete U-Shaped Base for Load Steps 5, 10, and 14

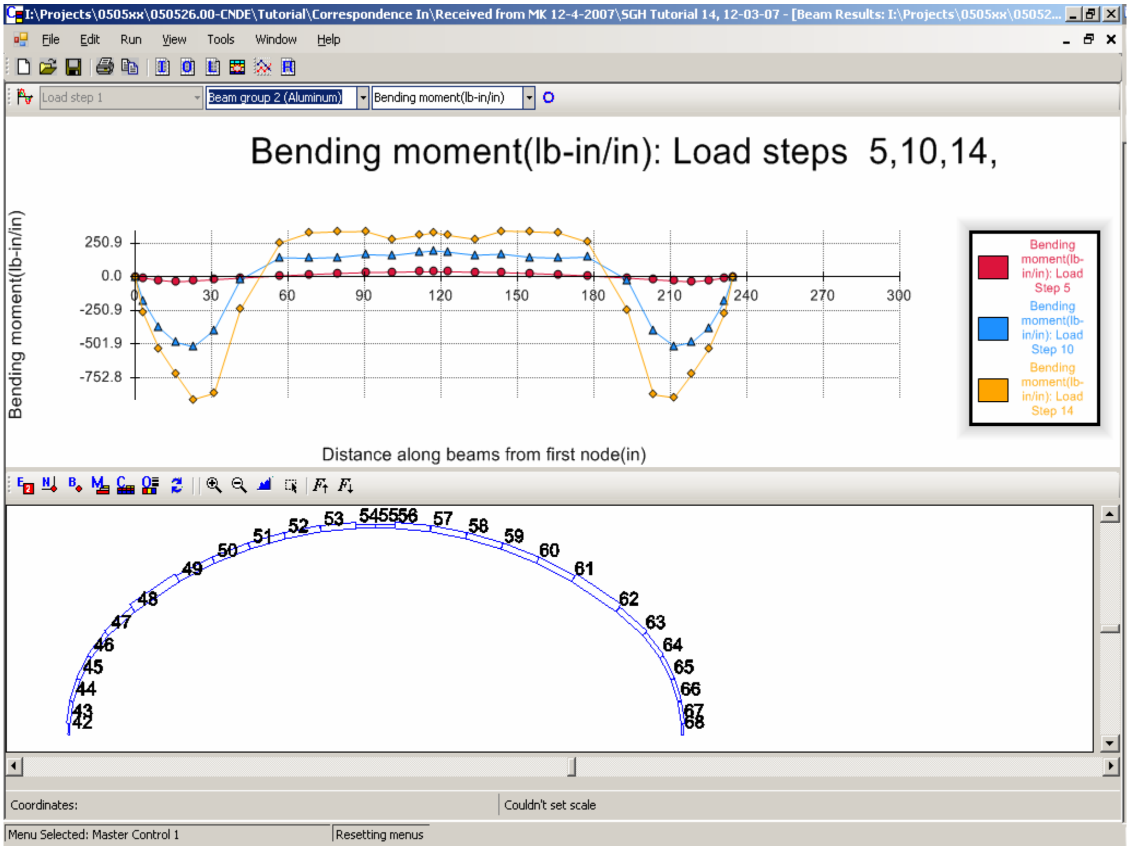


Figure 14-33 – Bending Moment in Aluminum Arch for Load Steps 5, 10, and 14

After an initial review of the output using the mesh and graph options, the user can select "View/Output Report (CANDE)" or select the "View CANDE output" button to look at the output file. Again the file should be inspected for signs of errors or incorrect input. Using the control window on the left the user can select to review the system input data or the design assessment (Figure 14-34) by clicking on the appropriate node of the 'Table of Contents' browser shown on the left side of the output viewer. The design assessment is printed after every load step so that the designer can assess the progress of the design. Figure 14-34 shows the final assessment printed at the end of the file for the aluminum arch.

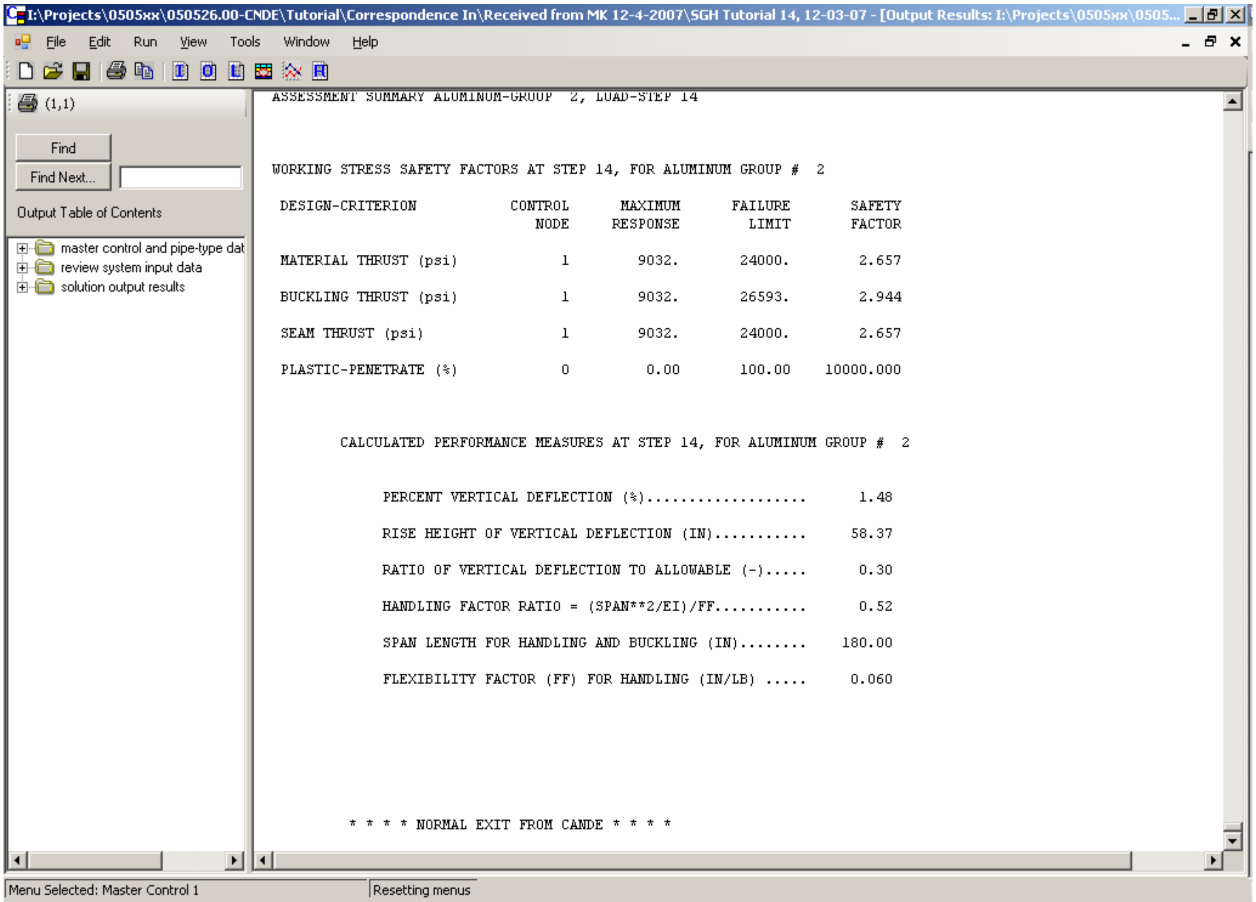


Figure 14-34 – Design Assessment Summary – Load Step 14

One additional tool is the Results Generator which can be used by selecting "View/Results Generator" or the "Start CANDE Results Generator" button. Figure 14-35 shows the Results Generator input screen set to obtain deflection and moment data from the full output file. After selecting the desired output criteria, click the "Generate/Preview" button in the lower left corner of the window. A portion of the report with the desired data is shown in Figure 14-36. The three tabs shown in Figures 14-35 and 14-36 are described in the following:

- General/Mesh Output – allows the user to request to view general mesh output such as thrusts, shears, bending moments and nodal displacements in the beam elements, interface element forces and displacements, and stresses and strains in the remaining mesh elements.

- Beam Output – allows the user to select to view more detailed output within the beam elements such as maximum fiber stresses, strain ratios, etc.

- Report Preview – displays the desired report.

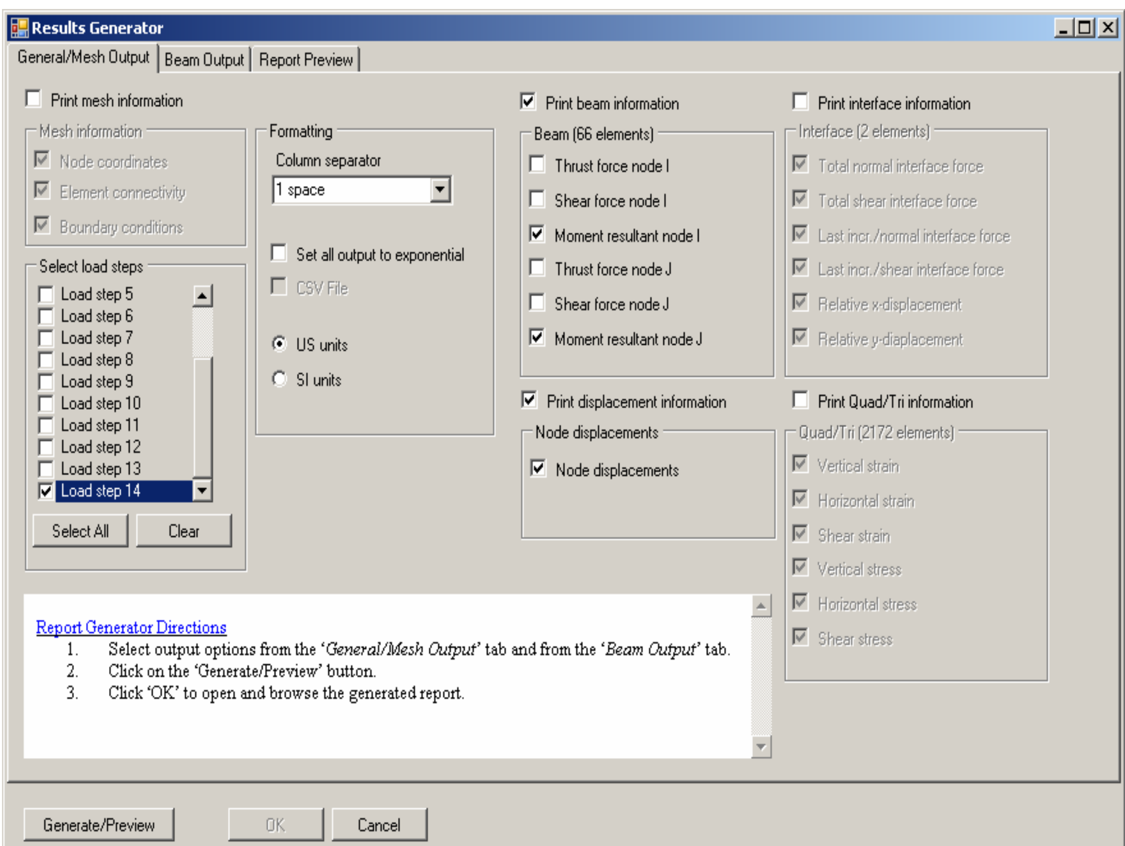
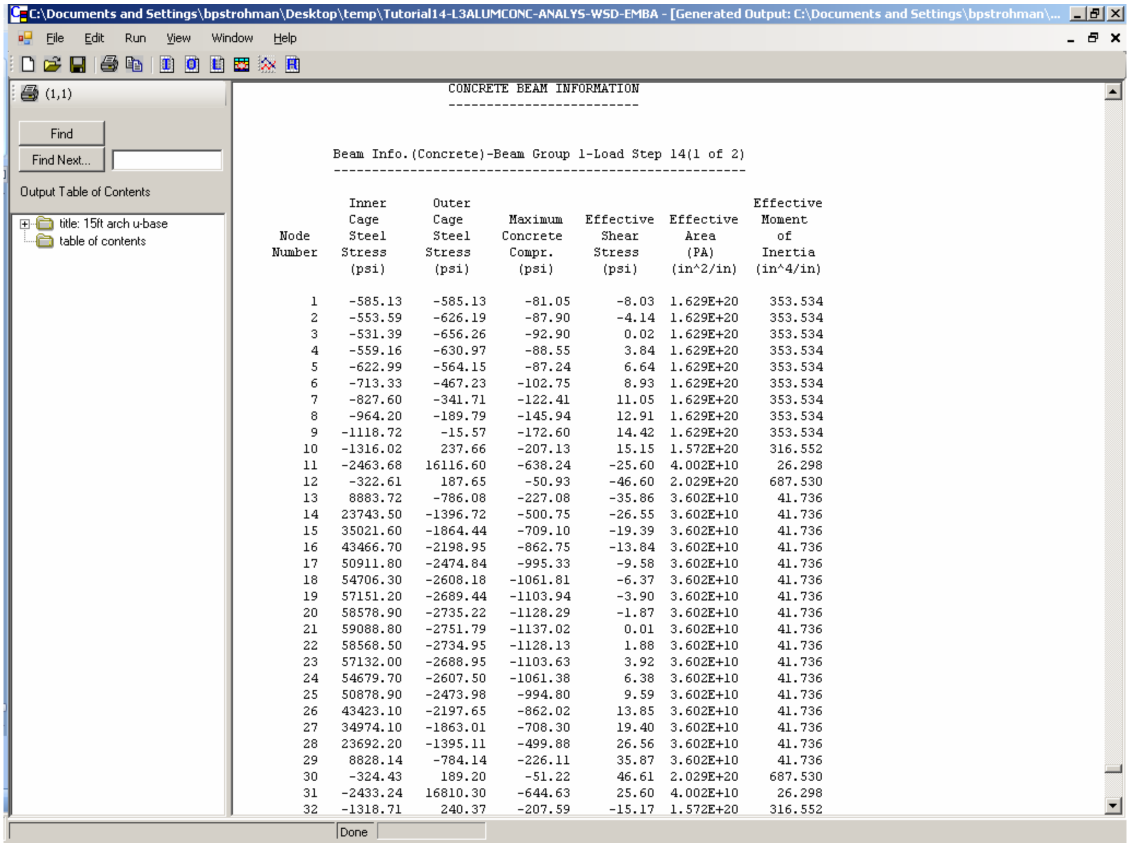


Figure 14-35 – Results Generator Input Screen – Load Step 14 Bending Moments and Deflections



CONCRETE BEAM INFORMATION

Beam Info.(Concrete)-Beam Group 1-Load Step 14(1 of 2)

Node Number	Inner Cage Steel Stress (psi)	Outer Cage Steel Stress (psi)	Maximum Concrete Compr. (psi)	Effective Shear Stress (psi)	Effective Area (PA) (in^2/in)	Effective Moment of Inertia (in^4/in)
1	-585.13	-585.13	-81.05	-8.03	1.629E+20	353.534
2	-553.59	-626.19	-87.90	-4.14	1.629E+20	353.534
3	-531.39	-656.26	-92.90	0.02	1.629E+20	353.534
4	-559.16	-630.97	-88.55	3.84	1.629E+20	353.534
5	-622.99	-564.15	-87.24	6.64	1.629E+20	353.534
6	-713.33	-467.23	-102.75	8.93	1.629E+20	353.534
7	-827.60	-341.71	-122.41	11.05	1.629E+20	353.534
8	-964.20	-189.79	-145.94	12.91	1.629E+20	353.534
9	-1118.72	-15.57	-172.60	14.42	1.629E+20	353.534
10	-1316.02	237.66	-207.13	15.15	1.572E+20	316.552
11	-2463.68	16116.60	-638.24	-25.60	4.002E+10	26.298
12	-322.61	187.65	-50.93	-46.60	2.029E+20	687.530
13	8883.72	-786.08	-227.08	-35.86	3.602E+10	41.736
14	23743.50	-1396.72	-500.75	-26.55	3.602E+10	41.736
15	35021.60	-1864.44	-709.10	-19.39	3.602E+10	41.736
16	43466.70	-2198.95	-862.75	-13.84	3.602E+10	41.736
17	50911.80	-2474.84	-995.33	-9.58	3.602E+10	41.736
18	54706.30	-2608.18	-1061.81	-6.37	3.602E+10	41.736
19	57151.20	-2689.44	-1103.94	-3.90	3.602E+10	41.736
20	58578.90	-2735.22	-1128.29	-1.87	3.602E+10	41.736
21	59088.80	-2751.79	-1137.02	0.01	3.602E+10	41.736
22	58568.50	-2734.95	-1128.13	1.88	3.602E+10	41.736
23	57132.00	-2688.95	-1103.63	3.92	3.602E+10	41.736
24	54679.70	-2607.50	-1061.38	6.38	3.602E+10	41.736
25	50878.90	-2473.98	-994.80	9.59	3.602E+10	41.736
26	43423.10	-2197.65	-862.02	13.85	3.602E+10	41.736
27	34974.10	-1863.01	-708.30	19.40	3.602E+10	41.736
28	23692.20	-1395.11	-499.88	26.56	3.602E+10	41.736
29	8828.14	-784.14	-226.11	35.87	3.602E+10	41.736
30	-324.43	189.20	-51.22	46.61	2.029E+20	687.530
31	-2433.24	16810.30	-644.63	25.60	4.002E+10	26.298
32	-1318.71	240.37	-207.59	-15.17	1.572E+20	316.552

Figure 14-36 – Results Generator – Load Step 14 Beam Stresses