CME-TRUSS (Version 2.1)
User's Manual

INTRODUCTION

CME-TRUSS is an interactive program for structural analysis of two-dimensional trusses. The software is written in the Microsoft **Visual Basic for Windows** programming language. The solution algorithm is based on the direct stiffness method of matrix structural analysis. This program is intended for educational use only.

CME-TRUSS gives the user the capability to:

- ❑ create a truss model interactively, with immediate graphical display of joints, members, supports, and loads, as the model is being developed;
- ❑ define cross-section properties by selecting AISC standard rolled shapes from a menu;
- ❑ define multiple load cases and load case combinations;
- ❑ modify, add to, or delete any part of the structural model directly, without having to re-enter the entire input file;
- ❑ save your structural model at any time in the development process, whether or not the model is complete;
- ❑ run multiple analyses without ever exiting the program;
- ❑ perform a cost estimate for a steel truss composed of AISC standard rolled shapes.

SYSTEM REQUIREMENTS

CME-TRUSS requires an IBM-compatible personal computer running Microsoft Windows 3.0 or better (Windows 3.1 is preferred); a VGA or better monitor; and a mouse or equivalent pointing device. It does not require a math coprocessor but runs considerably faster with one present.

INSTALLATION

The ***CME-TRUSS*** package consists of the following:

Program files

CMETRUSS.EXE	Executable program file
WSHAPES.TXT	Text file containing cross-section data on AISC wide-flange (W) shapes
WTSHAPES.TXT	Text file containing cross-section data on AISC WT shapes
LLSHAPES.TXT	Text file containing cross-section data on AISC double-angle (LL) shapes

System files

VBRUN300.DLL	Visual Basic run-time module
COMMDLG.DLL	Visual Basic common dialog dynamic-link library
CMDIALOG.VBX	Visual Basic common dialog custom control

To install CME-TRUSS: RUN "SETUP" FROM FILE MANAGER

CREATING A STRUCTURAL MODEL IN CME-TRUSS

To start ***CME-TRUSS***, double-click the program-item icon, or run **`CMETRUSS.EXE`**. Once the program has loaded, you will see an information screen. Read it, and hit RETURN or click on the **OK** button to proceed. You will then be asked whether or not you want to use AISC standard steel shapes for input of member properties. If you answer **Yes**, the shapes data files are loaded. (This takes a while; be patient.) If **No**, the data files are not loaded, and the steel shapes input option is disabled. At this point, the ***CME-TRUSS*** **Main Menu** should fill the screen.

THE MAIN MENU

The **Main Menu** window consists of four regions:

(1) The *menu bar*, at the top, which is used to control the program.
(2) The *tool bar*, immediately below, which is used to control the appearance of the plotted structural model and to set the current load case.
(3) The *status bar*, at the bottom, which provides the current file name and the status of the current structural model.
(4) The *client area*, where various other windows will be displayed as the structural model is developed.

ENTERING JOINTS

From the *menu bar* on the **Main Menu**, select **Structure**, then **Joints.** ***CME-TRUSS*** will prompt you for a title, to be used as a heading for your analysis output. Enter a title, and click **OK** to display the **Enter Joints** window. Note that the **Current Structure** window also appears at this time. This window will display a plot of your truss as you develop it. In the **Enter Joints** window, type the x-coordinate of your first joint. (The cursor will already be in the correct position.) Then click inside the y-coordinate box (or hit the TAB key) to move the cursor, and type the y-coordinate. If this joint is a support, click the appropriate check box(es) to indicate restraint in the x-direction or y-direction (rollers), or both (pin). If you make an error in entering coordinates, click the appropriate box (or TAB to that location), then edit using standard keystrokes (BACKSPACE, DELETE, arrow keys, etc.) Once all joint data are correct, click the **Enter** button, or hit the **Enter** key. Your entry is immediately appended onto the **CURRENT JOINT LIST** below, indicating that it is now a defined joint in your structural model. In the **Enter Joints** window, the joint number box is automatically incremented by 1, and the coordinate entry boxes are cleared, ready for the next input. Enter the remaining joints in the same manner.

You do not have to enter joints in numerical order; however, your *completed* structural model must have continuously numbered joints, from 1 through the highest joint number, with no gaps. If you enter joints out of sequence, ***CME-TRUSS*** will generate the missing joint numbers automatically, and include them in the **CURRENT JOINT LIST** as a reminder.

If you wish to edit a joint (i.e., change its coordinates or support conditions), highlight that entry in the **CURRENT JOINT LIST**, and click on the **Edit** button. (A double-click on the list entry will accomplish the same thing a bit more quickly.) The coordinates and support information for that joint will appear in the boxes above, where they can be edited. When the changes are complete, click the **Enter** button or hit the **Enter** key to re-enter the modified joint.

To delete a joint, highlight the entry in the **CURRENT JOINT LIST**, and click on the **Delete** button. (*Be careful!* Any previously defined members or loads associated with that joint will also be deleted.)

Note that, as each new joint is entered, it is also plotted automatically in the **Current Structure** window. For more information on this feature, see ***USING THE CURRENT STRUCTURE WINDOW*** below.

When you are done entering joints (or when you are tired of entering joints and want to do something else), click the **Done** button, and return to the **Main Menu**.

ENTERING MEMBERS

From the **Main Menu**, select **Structure**, then **Members** to display the **Enter Members** window. Using the same procedure as for joint entry, type the joint numbers corresponding to the ends of each member in the **From** and **To** boxes. ***CME-TRUSS*** will not allow you to define a member end if the corresponding joint has not yet been defined.

Define the member cross-section properties by *either*:

(1) entering the modulus of elasticity, E, and cross-sectional area into the appropriate boxes, or

(2) selecting the appropriate AISC standard shape from the **Steel Shape** list. (This feature is only available if you elected to load the steel shapes data when you started the program.) To view the list, first click on the appropriate type of shape (**W**, **WT**, or **LL**); then click the down-arrow. The list of standard shapes will drop down. Move through it using the scroll bar on the right side of the list. To select a steel shape, click the appropriate list entry. ***CME-TRUSS*** will automatically enter the appropriate area and modulus of elasticity (E=29000 ksi) into the corresponding boxes.

IMPORTANT
When you define cross-sectional properties by selecting steel shapes, you *must* use **inches** for all dimensions and **kips** for all loads in your structural model.

Once a member entry is complete, click the **Enter** button or hit the **Enter** key to append the member to the **CURRENT MEMBER LIST**. Again, each new member is automatically plotted in the **Current Structure** window.

Members are edited and deleted in the same manner as joints.

ENTERING LOADS AND LOAD CASE COMBINATIONS

From the **Main Menu**, click **LoadCase**, then **New,** to display the **Load Case** window. Type a load case name (e.g., Snow, Roof Live), then enter the joint number and the x- and y-components of the corresponding joint load. Click **Enter** to append the load to the **CURRENT LOADS** list, and repeat the process until all loads are entered. The **Edit**, and **Delete** buttons work the same way as in previous windows. When all loads are entered, click the **Done** button. As each new load case is entered, it is automatically appended to the **Current Load Case** list box, located on the right side of the tool bar.

Repeat this process for each new load case. The allowable number of load cases is limited to 20. You can edit or delete an existing load case using the corresponding items in the **Load Case** menu. Edit and delete operations always apply to the "current load case"--the one displayed in the **Current Load Case** list box. To change the current load case, click the down arrow on the list box, and select a new load case or load combination.

CME-Truss will also analyze up to 10 linear combinations of load cases. To enter load case combinations, return to the **Main Menu**, then click **Load Combination**, and **New**. You will be prompted for the number of load case combinations. Enter the number and click **Enter** to display the **Load Combinations** window. This

window consists of a matrix with load case combinations on the vertical axis and load cases on the horizontal. Within the boxes, enter the multipliers (load factors) for each load case. For example, if Load Case 1 is dead load (D) and Load Case 2 is live load (L), then enter 1.2 and 1.6 in the first row to define Load Case Combination 1 as 1.2D + 1.6L.

USING THE CURRENT STRUCTURE WINDOW AND TOOL BAR

The **Current Structure** window is automatically loaded whenever you enter joints or open an existing structural data file. Normally it is displayed *underneath* the **Enter Joints, Enter Members,** and **Load Case** windows; thus at any given time, the plot may be partially obscured. If you wish to view the entire structural model, you can do so by:

(1) clicking anywhere on the **Current Structure** window, to bring it to the front,
(2) changing the **Current Structure** window to the full-screen mode by clicking its **Maximize** button, or
(3) Minimizing the **Enter Joints** window by clicking its **Minimize** button.

At any time, you can view the x-y coordinate axes, the overall dimensions of the model, the joints, joint numbers, member numbers, loads, and displacements by clicking the appropriate button on the **Main Menu** toolbar. (The displacement button only functions after a structural analysis has been performed.) When the load button and displacement button always cause the "current load case" to be displayed.

Clicking the printer button causes the currently displayed structure to be printed on the default printer. Ensure that the printer is connected and on line before using this feature.

SAVING THE STRUCTURAL MODEL

You can save the structural model to a file *at any time during the development process*. The saved input file can then be loaded into ***CME-TRUSS*** later, to continue development or to run additional analyses.

To save, select **File**, then **Input**, then **Save** from the **Main Menu.** If you have not previously saved the model**, y**ou will be prompted for a filename and path. To save under a different name, use **Save As**. If you do not specify a filename extension, ***CME-TRUSS*** uses **`*.cme`** as a default.

To open a previously saved file, select **File**, then **Input**, then **Open.**

Though it is not necessary to save your model before running the structural analysis, *frequent saving is highly recommended*. ***CME-TRUSS*** 2.0 is the first Windows-compatible version of this program; its author is a civil engineer, not a software developer. Its error-handlers are not robust, and bugs are inevitable.

PERFORMING A STRUCTURAL ANALYSIS

The status bar at the bottom of the **Main Menu** has three status windows--one each for joints, members, and loads. When all three windows are green, the structural model is complete (though not guaranteed to be correct or fully consistent), and ready for analysis. Click **Run**, then **Structural Analysis**. If you have more than one load case, the **Output Control** window will be displayed, and you will be given the opportunity to designate specific load cases and load case combinations for output. Check the appropriate items, and click the **GO** button to run the analysis.

When the analysis is complete, the **Analysis Results** window will display the output file for the first available load case. Use the scroll bar to view portions of the output which are not visible initially. The **Current Structure** window will show the corresponding displacements. To view the results for any other load case or combination, change the selected item in the **Current Load Case** list box.

Your analysis results are *not* automatically saved. If you want to save them (along with an echo-print of your input data), select **File**, then **Output**, then **Save** (or **Save As**) from the **Main Menu.** *Do not use the same file name as you used for your input file*. The default extension for output files is **`*.txt`**. (This facilitates subsequent editing of the output file or insertion of the file into another document.)

You can send the output file directly to the default printer by selecting **File**, then **Output**, then **Print** from the **Main Menu.**

OTHER FEATURES

☞ Enter or change the title of your structural model by clicking **Structure**, then **Title**.

☞ Dump the current structural model and create a new one by clicking **Structure**, then **Start Over**.

☞ Perform a cost estimate for the current truss by clicking **Run**, then **Cost Analysis**. This option will only work if AISC steel shapes have been loaded and assigned to *all* members in the model. You must enter the cost per pound of steel, the cost per connection, and the cost per steel shape before executing the analysis.

☞ View or remove the tool bar, status bar, **Current Structure** window, or **Analysis Results** window by clicking the appropriate item in the **View** menu.

If you experience problems with this software, or if you have questions or comments, please contact MAJ Stephen J. Ressler, Department of Civil and Mechanical Engineering, West Point, NY 10996.

CME-TRUSS + LRFD LOADTEST (Version 2.0)
User's Manual Supplement

INTRODUCTION

CME-TRUSS+LRFD LOADTEST is an enhanced version of ***CME-TRUSS***, developed for use by the instructor in an undergraduate structural steel design course. The ***LRFD LOADTEST*** feature provides a comprehensive LRFD-based strength and serviceability evaluation of student truss designs, created via the standard version of ***CME-TRUSS.*** The evaluation includes a simulated load test of the student's truss, displayed as a full-color animation.

For obvious reasons, this program should not be given to students.

SYSTEM REQUIREMENTS

System requirements for this program are the same as for ***CME-TRUSS***; however, for fully effective display of the animated load test, the system should also include a monitor and video card capable displaying 256-color graphics.

INSTALLATION

Installation of ***CME-TRUSS+LRFD LOADTEST*** is accomplished in the same manner as is ***CME-TRUSS***, except that the executable program file is **`LOADTEST.EXE`**. The remaining three program files (**`*SHAPES.TXT`**) and the three system files must be installed as before.

USING *CME-TRUSS+LRFD LOADTEST*

Using this program to evaluate student truss designs requires three actions:

(1) Create a design requirement for the students.
(2) Create a set of evaluation parameters.
(3) Run the simulated load test on student truss designs.

These are described in the following sections.

CREATING THE DESIGN REQUIREMENT

LRFD LOADTEST permits a great deal of flexibility in creating a design requirement for subsequent evaluation. Due to certain limitations in the program, however, the design requirement cannot be entirely unconstrained. The design requirement should have the following characteristics:

- The scenario should require the design of a ***roof truss***. (In its current form, ***LRFD LOADTEST*** assumes that the controlling factored load combination is LRFD (A4-3), ***1.2D + 1.6(L_r or S) + 0.5L***. Future versions of the program will consider other load combinations.)

- The truss should have a specified ***span length***.

- The truss should have a specified ***maximum depth***, measured from centerline to centerline of the top and bottom chords.

- The truss design should use a single, specified grade of steel throughout. (Mixing two or more steel grades in the same structure is not permitted.)

- The truss design must use welded connections, with a shear lag reduction coefficient (U) of 0.85. (In calculating fracture strength, ***LRFD LOADTEST*** cannot account for net section effects in bolted connections.)

- The truss should have specified support conditions (e.g., pin-roller, pin-pin).

- All loads must be uniformly distributed over the entire span length. The magnitudes of the service (unfactored) loads must be specified, or sufficient information must be provided for students to calculate them. Specified loads are:

    - Distributed Dead Load - a uniformly distributed load applied to the top chord of the truss, ***not including self-weight of the truss***. This load could include the weight of roofing material, a slab, and roof framing.

    - Concentrated Dead Load - a point load applied at every top chord joint. Its magnitude is not dependent on top-chord joint spacing. This load is intended to represent the weight of roof framing members running perpendicular to the truss, connected at the top-chord panel-points.

    - Roof Live Load - a uniformly distributed load applied to the top chord of the truss. Calculation of this load is specified in ANSI/ASCE 7-88, *Minimum Design Loads for Buildings and Other Structures*.

    - Snow Load - a uniformly distributed load applied to the top chord of the truss. Calculation of this load is specified in ANSI/ASCE 7-88, *Minimum Design Loads for Buildings and Other Structures*.

    - Live Load - a uniformly distributed load applied to the top chord of the truss. (This load is likely to be zero for a roof truss.)

- A maximum service-load deflection (calculated for $\boldsymbol{L_r + S + L}$) should be specified.

- Cost factors should be specified as follows:

    - Cost per pound of steel, in dollars

    - Cost per connection, in dollars, where one connection is defined as one member connected to one joint

    - Cost per steel shape used in the design, in dollars.

It is quite possible to create a design scenario which imposes these constraints without significant loss of realism.

As part of the design project, students must be required to perform their structural analyses using ***CME-TRUSS***; and with their final project reports, they must submit a floppy disk containing a saved copy of their structural input data files.

CREATING THE EVALUATION PARAMETER FILE

Once the design requirement is developed, the instructor must develop the set of evaluation parameters which ***LRFD LOADTEST*** will use to assess the adequacy of student truss designs. The parameters must be consistent with the design requirement described above. They take the form of a formatted data file, created, saved, and subsequently opened by ***LRFD LOADTEST.***

To create a parameter file, run ***LRFD LOADTEST,*** and from the **Main Menu,** select **File**, then **LRFD Parameters**, then **New**. The **LRFD Parameters** window will be displayed. In the appropriate text box, enter the following information:

- ❑ Span length
- ❑ Maximum depth
- ❑ Yield stress of steel
- ❑ Ultimate stress of steel
- ❑ Required number of pinned supports
- ❑ Required number of roller supports
- ❑ All loads described above
- ❑ All cost factors described above
- ❑ Maximum allowable service-load deflection
- ❑ Tolerance (a decimal value which allows for a "margin of error" in all checks performed by ***LRFD LOADTEST.*** The suggested value of 0.02 provides a 2% margin of error. This feature gives students an incentive not to "play it safe" in their efforts to develop optimal designs.

Once all the parameters are entered, click the **Enter and Save** button to save the parameter file. The default filename extension for parameter files is **`*.par`**.

RUNNING THE SIMULATED LOAD TEST

To evaluate a student truss design, perform the following actions:

- ♦ Run ***LRFD LOADTEST.***
- ♦ Open a student's input data file, using **File, Input, Open** on the **Main Menu**.
- ♦ From the **Main Menu**, select Run, then **LRFD Load Test**.
- ♦ You will be prompted for an output file name. When the simulated load test is complete, this file will contain a member-by-member summary of the LRFD strength and serviceability analysis, for all relevant limit states. The default file name is **REPORT.TXT**.
- ♦ You will then be asked to enter the loaded joints, in order from left to right. This information permits ***LRFD LOADTEST*** to convert uniformly distributed loads to equivalent concentrated loads, applied at the appropriate joints. Once the joints are entered, the truss is plotted again, with the loaded joints highlighted in red. You will be asked to verify them.
- ♦ Once the loaded joints are verified, ***LRFD LOADTEST*** performs the following actions:
  - ❑ Calculates the self-weight of the truss and converts to appropriate joint loads.
  - ❑ Calculates the total factored load (LRFD A4-3) and converts to appropriate joint loads.
  - ❑ Calculates the total service live load (L_r+S+L) and converts to appropriate joint loads.
  - ❑ Updates the structural model and runs a structural analysis to determine member forces and deflections for the two load cases described above.

- Calculates the tensile strength of each member in the truss, considering yielding and fracture.
- Calculates the compressive strength of each member, considering flexural buckling, flexural-torsional buckling, and local buckling.
- Checks the adequacy of each member with respect to the strength limit states listed above.
- Checks the adequacy of each member with respect to the LRFD-specified upper limits on slenderness ratio (L/r<300 for tension member; L/r<200 for compression members).
- Checks the calculated max service load deflection against the specified max allowable deflection.

When these checks are complete, the truss is displayed again, but this time on a black background. Click the **Load Test** button in the upper left-hand corner to begin the animated load test simulation. As the test proceeds, load increases from zero to the maximum required strength; members change color (from light blue to bright red) to indicate to the proportion of their capacity "used up" at a given point in the load cycle. If a member fails, it turns black and thus disappears from the screen.

Once the load test is complete, click the **Done** button, and the **LRFD Load Test Results** window will display a summary of the evaluation. A series of colored boxes show whether the truss satisfied the length, depth, support condition, strength, and serviceability requirements specified in the design requirement.

Click **Done** to end the LRFD load test. At this point, a new input file can be opened, and the load test repeated.