Texas Instruments
TI-99/4 Home Computer

Cassette Software

Model PHT 6016

Structural Engineering Library

Five powerful programs which enable you to solve common problems in structural engineering.

- **MOMENT OF INERTIA—Calculates the composite area, moment of inertia, and distance from baseline to centroid of a composite structure.**
- **DYNAMIC LOADING/SINGLE DEGREE OF FREEDOM—Determines the elastic response of a system with a single degree of freedom when subjected to an impulse load.**
- **CONCRETE BEAM STRESS ANALYSIS—Computes the resistant moment of rectangular sections and the steel area required for structural flexure members.**
- **TRUSSES—Determines the distribution of forces in Warren, Howe, and Pratt trusses.**
- **FOUR-SPAN DISTRIBUTION—Calculates the final bending moments for continuous beams with five supports.**

Requires the use of a cassette tape recorder (not included) for loading the program contents into the TI-99/4 Home Computer memory.

As this manual was designed for the U.S. market, the warranty conditions described herein are not applicable in the U.K. The only valid Guarantee Conditions are those set forth in the "Users Reference Guide" accompanying the Home Computer.

Table of Contents

Author: Texas Instruments

Language: TI-99/4 BASIC

Hardware: TI-99/4 Home Computer
Disk Controller and Drive or Cassette Tape Recorder

Media: Diskette or Cassette

The Structural Engineering Library is a multipurpose package, offering five programs which solve problems commonly encountered in structural engineering and relieve you of the tedium of calculations.

- The Moment of Inertia program calculates the moment of inertia for any combination of two or more structural or rectangular members.

- The Dynamic Loading/Single Degree of Freedom program determines the elastic response of a system with a single degree of freedom when subjected to an impulse load.

- The Concrete Beam Stress Analysis program calculates the resistant moment of rectangular sections and the area of steel required for structural flexure members based on the strength of design method.

- The Trusses program calculates the distribution of forces for three types of trusses: Warren, Howe, and Pratt.

- The Four-Span Moment Distribution program determines the bending moments at supports for continuous beams or girders with five supports. The loading conditions can be uniform, concentrated, or combined with variable end conditions.

The Moment of Inertia program computes the composite area, calculates the total moment of inertia about the horizontal centroidal axis, and locates the centroid for any of the following:

1. two structural members;
2. one structural member and one rectangular member;
3. two rectangular members; or
4. any of the above combinations with additional structural or rectangular members.

This program assumes that the composite shapes are symmetric about the vertical axis.

Numeric data can be entered in integer or real form. To remove an area from a composite plate section, use negative values for rectangular width. The program retains all significant digits during calculation although the final answer is rounded. You can enter English or metric measurements provided you are consistent. (For example, if the distances are measured in inches, the moments should also be given in inches.)

Three formulas are used in calculating the moment of inertia. The first one calculates the moment of inertia of a rectangular area about its horizontal centroidal axis.

$$I = \frac{bd^3}{12}$$

where

b = width of rectangular member

d = height of rectangular member

The second formula calculates the moment of inertia transfer from the centroid to another point for any area.

$$I' = I + Ay^2$$

where

A = area

y = distance from baseline to centroid

The composite moment of inertia of two areas is calculated using this formula.

$$\overline{I} = I_1 + A_1 (\overline{y} - y_1)^2 + I_2 + A_2 (\overline{y} - y_2)^2$$

where

$$\overline{y} = \frac{A_1 y_1 + A_2 y_2}{\overline{A}}$$

$\overline{A} = A_1 + A_2$ = total area

A_1 = area of member #1

A_2 = area of member #2

I_1 = moment of inertia of member #1

I_2 = moment of inertia of member #2

$\overline{I}$ = total moment of inertia

y_1 = distance from baseline to centroid of member #1

y_2 = distance from baseline to centroid of member #2

$\overline{y}$ = total distance from baseline to centroid

STEP 1: If the computer is not already in the BASIC mode, select TI BASIC. To load the program from a diskette, insert the diskette into the disk drive, type

OLD DSK1.MOMENT

and press ENTER.

To load the program from a cassette tape, refer to the "Loading Cassettes" section in this manual for instructions on determining the program's position on the cassette tape. When you have inserted the cassette and positioned the tape counter, type

OLD CS1

and press ENTER. The computer then displays directions for loading the tape. Refer to "Loading Cassettes" if you have difficulty in loading the program from the cassette.

STEP 2: When the cursor reappears, type RUN, and press ENTER. The MOMENT OF INERTIA title screen appears briefly, followed by a description of the program. Press ENTER to continue.

STEP 3: Then the following list of options is displayed.

1 TWO STRUCTURAL MEMBERS
2 ONE STRUCTURAL AND RECTANGULAR MEMBER
3 TWO RECTANGULAR MEMBERS
4 EXIT

Now select the type of members for which you wish to calculate the moment of inertia by entering the appropriate number.

After you have performed the calculation for two members, you may choose to return to the list of options by pressing SHIFT Z (BACK) or to add a member to the composite structure by pressing SHIFT V (PROC'D). Refer to the section entitled "ADDING ADDITIONAL MEMBERS" for instructions.

OPTION 1: TWO STRUCTURAL MEMBERS
If you select Option 1, you may enter data for two structural members. First, enter the cross-sectional area, moment of inertia, and distance from baseline to the centroid for one member. Then enter the same data for the second member. The program displays the area, moment of inertia, and distance from baseline to centroid for the composite structure. Then respond to the prompt PRESS BACK, PROC'D.

OPTION 2: ONE STRUCTURAL AND ONE RECTANGULAR MEMBER
If you select Option 2, you may enter data for one rectangular and one structural member. First, enter the width of the rectangular member, its height, and the distance from the baseline to its centroid. Then enter the cross-sectional area of the structural member, its moment of inertia, and the distance from the baseline to its centroid. The program displays the area, the moment of inertia, and the distance from the baseline to the centroid of the composite structure. Then respond to the prompt PRESS BACK, PROC'D.

OPTION 3: TWO RECTANGULAR MEMBERS
If you select Option 3, you may enter data for two rectangular members. Enter the width of the first member, its height, and the distance from the baseline to its centroid. Then enter the same data for the second rectangular member. The program displays the area, the moment of inertia, and the distance from the baseline to the centroid of the composite structure. Then respond to the prompt PRESS BACK, PROC'D.

OPTION 4: EXIT
To leave the program, select Option 4. The message ** DONE ** is displayed, and the program stops.

ADDING ADDITIONAL MEMBERS

To add additional members to a composite structure, press SHIFT V (PROC'D) after the calculations are complete. Next indicate the type of member you are adding; enter R for a rectangular member or S for a structural member. For a rectangular member, enter the width of the rectangle, its height, and the distance from baseline to centroid. If adding a structural member, enter the cross-sectional area of the member, its moment of inertia, and the distance from its baseline to the centroid. The program displays the area, the moment of inertia, and the distance from the baseline to the centroid for the composite structure. You may return to the list of options by pressing SHIFT Z (BACK) or add an additional member to the structure by pressing SHIFT V (PROC'D).

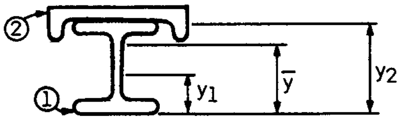
To help you understand how the program works, an example is provided for each option. First, load and run the program as described in Steps 1 and 2 of the "User Instructions."

Two Structural Members

Select Option 1 to calculate the area, moment of inertia, and the distance from the baseline to the centroid of a composite figure with two structural members.

Find the moment of inertia, the composite area, and the distance from the baseline to the centroid of a crane girder composed of a W24 x 76 with a cap channel of C12 x 20.7.

Structural Member # 1	Structural Member # 2
$A_1 = 22.4$	$A_2 = 6.09$
$I_1 = 2100$	$I_2 = 3.88$
$y_1 = 23.91 \div 2 = 11.95$	$y_2 = 23.91 + .282 - .698 = 23.494$



(Reference data from the Manual of Steel Construction, 7th Edition, American Institute of Steel Construction, 1973.)

Now enter the data in response to the prompts on the screen:

PROMPT	ENTER	COMMENTS
AREA(#1):	22.4	Cross-sectional area of girder
MOMENT OF INERTIA(#1):	2100	Moment of inertia of girder
BASELINE TO CENTROID(#1):	11.955	Distance from baseline to girder's centroid
AREA(#2):	6.09	Cross-sectional area of cap channel
MOMENT OF INERTIA(#2):	3.88	Moment of inertia of channel
BASELINE TO CENTROID(#2):	23.494	Distance from baseline to channel's centroid

The following answers are displayed. (Answers are rounded to two decimal places.)

COMPOSITE AREA	28.49
MOMENT OF INERTIA	2741.42
BASELINE TO CENTROID	14.42

To return to the list of options, press SHIFT Z (BACK).

One Structural and One Rectangular Member

Select Option 2 to calculate the area, moment of inertia, and the distance from the baseline to the centroid of a composite figure made up of one structural and one rectangular member as described below.

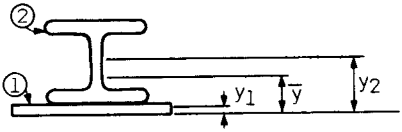
Find the moment of inertia, the area, and the distance from the baseline to the centroid for a lintel composed of a W8 x 17 with a bottom PL 3/8 x 11-1/2.

Rectangular Member

$b = 11.5$
$d = .375$
$y_1 = .375 \div 2 = .1875$

Structural Member

$A_1 = 5.01$
$I_1 = 56.6$
$y_2 = 4.00 + .375 = 4.375$



(Reference data from the Manual of Steel Construction, 7th Edition, American Institute of Steel Construction, 1973.)

Now enter the data as shown below:

PROMPT	ENTER	COMMENTS
WIDTH(REC):	11.5	Width of rectangular member
HEIGHT(REC):	.375	Height of rectangular member
BASELINE TO CENTROID(REC):	.1875	Distance from baseline to rectangle's centroid
AREA(STR):	5.01	Cross-sectional area of lintel
MOMENT OF INERTIA(STR):	56.6	Moment of inertia of lintel
BASELINE TO CENTROID(STR):	4.375	Distance from baseline to lintel's centroid

The following answers are displayed. (Answers are rounded to two decimal places.)

COMPOSITE AREA	9.32
MOMENT OF INERTIA	97.28
BASELINE TO CENTROID	2.43

To return to the list of options, press SHIFT Z (BACK).

Three Rectangular Members

Select Option 3 to calculate the composite area, moment of inertia, and the distance from the baseline to the centroid for the composite figure made up of three rectangular members shown below.

Find the moment of inertia, the composite area, and the distance from the baseline to the centroid for a plate girder composed of a top flange PL 2 x 14, bottom flange PL 1 1/2 x 24, and web PL 1/2 x 72.

Rectangular Member # 1

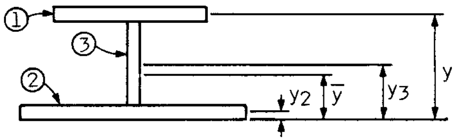
$b_1 = 14$
$d_1 = 2$
$y_1 = 1.5 + 72 + (2 \div 2) = 74.5$

Rectangular Member # 2

$b_2 = 24$
$d_2 = 1.5$
$y_2 = 1.5 \div 2 = .75$

Rectangular Member # 3

$b_3 = .5$
$d_3 = 72$
$y_3 = 1.5 + (72 \div 2) = 37.5$



(Reference data from the Manual of Steel Construction, 7th Edition, American Institute of Steel Construction, 1973.).

Now enter the data in response to the prompts on the screen:

PROMPT	ENTER	COMMENTS
WIDTH(#1):	14	Width of rectangle #1
HEIGHT(#1):	2	Height of rectangle #1
BASELINE TO CENTROID(#1):	74.5	Distance from baseline to the centroid of rectangle #1
WIDTH(#2):	24	Width of rectangle #2
HEIGHT(#2):	1.5	Height of rectangle #2
BASELINE TO CENTROID(#2):	.75	Distance from baseline to the centroid of rectangle #2

The following answers are displayed. (Answers are rounded to two decimal places.)

COMPOSITE AREA	64
MOMENT OF INERTIA	85681.31
BASELINE TO CENTROID	33.01

To add the third member to this structure, press SHIFT V (PROC'D). Now enter R to add a rectangular structure; then enter the data for this member.

WIDTH(REC):	.5	Data for additional member
HEIGHT(REC):	72	
BASELINE TO CENTROID(REC):	37.5	

The following answers are displayed for the structure that has three rectangular members. (Answers are rounded to two decimal places.)

COMPOSITE AREA	100
MOMENT OF INERTIA	101696.64
BASELINE TO CENTROID(REC)	34.63

To return to the list of options, press SHIFT Z (BACK). Select option 4 to exit the program. The message ** DONE ** is displayed, and the program stops.

The Dynamic Loading/Single Degree of Freedom program determines the elastic response of a single degree of freedom system subjected to an impulse load. The program calculates the acceleration, velocity, and deflection of the system based on your input of the following data: initial deflection, initial velocity, initial acceleration, weight, damping coefficient, spring constant, time increment, initial time, and force as a function of time. Many structures can be treated as a single mass and spring, subjected to dynamic loads, such as earthquakes, wind gusts, or blast loads. Once the elastic response is determined, the maximum stresses as a result of dynamic loading can be evaluated.

Numeric data can be entered in integer or real form. The program retains all significant digits during calculation although the final answer is rounded.

The numerical analysis used assumes two conditions. First it assumes that the acceleration varies linearly between time stations. Secondly, it assumes that structural damping of the viscous type occurs. The damping force is inversely proportional to the velocity.

The elastic response is calculated using the following formula.

$$\ddot{y}_{n+1} = \frac{F(t)_{n+1} - k(y_n + \dot{y}_n \Delta t + \frac{1}{3}\ddot{y}_n \Delta t^2) - C(\dot{y}_n + \frac{1}{2}\ddot{y}_n \Delta t)}{M + \frac{k\Delta t^2}{6} + \frac{C}{2}\Delta t}$$

$$\dot{y}_{n+1} = \dot{y}_n + \frac{1}{2}(\ddot{y}_n + \ddot{y}_{n+1})\Delta t$$

$$y_{n+1} = y_n + \dot{y}_n \Delta t + \frac{1}{3}\ddot{y}_n \Delta t^2 + \frac{1}{6}y_{n+1}\Delta t^2$$

where

y = displacement (ft)

$\dot{y}$ = velocity (ft/sec)

$\ddot{y}$ = acceleration (ft/sec^2)

k = spring constant (lb/ft)

M = mass (lb — sec^2/ft)

C = damping coefficient (lb/fp)

Δt = time movement (sec)

$F(t)$ = force (lb) as a function of time (sec)

(Reference: John M. Biggs, Introduction to Structural Dynamics, McGraw-Hill Book Company, 1964.)

STEP 1: If the computer is not already in the BASIC mode, select TI BASIC. To load the program from a diskette, insert the diskette into the disk drive, type

OLD DSK1.DYNAMIC

and press ENTER.

To load the program from a cassette tape, refer to the "Loading Cassettes" section in this manual for instructions on determining the program's position on the cassette tape. When you have inserted the cassette and positioned the tape counter, type

OLD CS1

and press ENTER. The computer then displays directions for loading the tape. Refer to "Loading Cassettes" if you have difficulty in loading the program from the cassette.

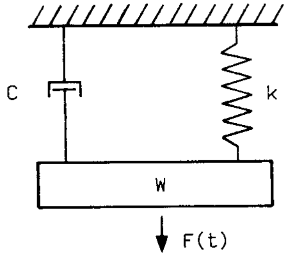
STEP 2: When the cursor reappears, type RUN, and press ENTER. The DYNAMIC LOADING (SINGLE DEGREE OF FREEDOM) title screen appears briefly, followed by a description of the program. Press ENTER to begin entering data.

STEP 3: Enter the initial deflection, initial velocity, initial acceleration, the weight in kips, the damping coefficient, the spring constant, the time increment, and the initial time. To calculate the accleration, deflection, and velocity at various points in time, enter the force that is being applied at a moment in time [f(t) at time t]. The time, accleration, deflection, and velocity are displayed. Continue to enter a value for force for each calculation of accleration, deflection, and velocity that you desire. When you have all the values you require, press ENTER without entering a value for FORCE.

STEP 4: You may then perform another calculation by entering Y in response to the prompt REDO? (Y/N), or you may stop the program by entering N.

To help you understand how this program functions, work the following example.

Given:



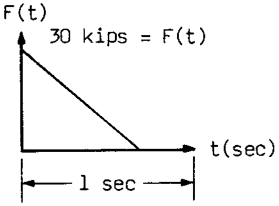
k = 20 kips/ft
W = 16.1 kips

10% Critical Damping

$$C = \frac{\%}{100} = \frac{20 \times 32.2}{w_n}$$

$$\text{where } w_n = \frac{kg}{W} = \frac{20 \times 32.2}{16.1}$$

so, C = .6325



$$\text{Request} \quad t = \frac{\text{natural perio}}{10}$$

$$= \frac{2}{10\ w_n} = .0994$$

therefore, use t = .05

y = y = 0, since starting from rest.

$y = [F(t)_0 - ky_0 - Cy_0\ ] \div M$; $M = 16.1 \div 32.2 = .5$
$y = 30 \div .5 = 60$ fps @ t=0

First, load and run the program as described in Steps 1 and 2 of "User Instructions." As the computer prompts you, enter the data shown below:

PROMPT	ENTER	COMMENTS
INITIAL DEFLECTION:	0	Required data
INITIAL VELOCITY:	0	
INITIAL ACCELERATION:	60	
WEIGHT:	16.1	
DAMPING COEFFICIENT(C):	.6325	
SPRING CONSTANT(K):	20	
TIME INCREMENT:	.05	
INITIAL TIME:	0	

Next enter the force which is the value of the function f(t) at t = .05.

FORCE:	28.5	Enter F(t) @ t = .05

The following values for acceleration, deflection, and velocity at that force are displayed. (Answers are rounded to four decimal places.)

TIME:	.05
ACCELERATION:	50.6562
DEFLECTION:	.0711
VELOCITY:	2.7664

Now enter the force which is the value of the function f(t) at t = .10.

FORCE:	27	Enter F(t) @ t = .10

The following values for acceleration, deflection, and velocity at that force are displayed. (Answers are rounded to four decimal places.)

TIME:	.1
ACCELERATION:	37.0439
DEFLECTION:	.267
VELOCITY:	4.9589

Enter the force, which is the value of the function f(t) at t = .15.

FORCE:	25.5	Enter F(t) @ t = .15

The following values for acceleration, deflection, and velocity at that force are displayed. (Answers are rounded to four decimal places.)

TIME:	.15
ACCELERATION:	20.7192
DEFLECTION:	.5545
VELOCITY: .	6.4029

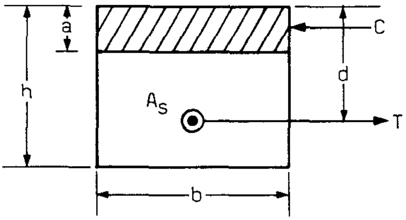
Because no other values are required, press ENTER without entering a value for FORCE. Enter N in response to the prompt REDO? (Y/N). The message ** DONE ** is displayed, and the program stops.

The Concrete Beam Stress Analysis program determines the resistant moment of rectangular flexural sections of concrete and the steel area required for structural flexure members based on the strength design method. (This program does not consider the minimum reinforcement requirements specified by the A.C.I. code of March 8, 1971.) Beams requiring only tension steel are evaluated further for stress block height, minimum coverage of steel, and tension/compression forces.

To perform the analysis, you enter the following data: compressive strength of the concrete, yield strength of the reinforcement, factored design load moment, height of the cross section, the distance from the top to the reinforcement centroid, and the width of the cross section.

Numeric data can be entered in integer or real form. The program retains all significant digits during calculations although the final answer is rounded.

The Concrete Beam Stress Analysis performs its analysis using the strength design method as illustrated in the following diagram and equations.



STRENGTH DESIGN METHOD
(A.C.I. 3/8/1971)

where

h = total height of cross section

a = height of equivalent rectangular stress block

b = width of cross section

A_S = tension reinforcement

d = distance from extreme compression fiber to centroid of tension reinforcement

C = compressive force at the centroid of compression block

T = tensile force at the centroid of tension reinforcement

EQUATIONS:

$$B_1 = 0.85 - 0.05\,(f'_c - 4) \leq 0.85$$

where

B_1 = concrete distribution factor

f'_c = specified compressive strength of concrete

$$P_{max} = \left(\frac{0.75 + 0.85\ B_1 f_c'}{f_y}\right)\left(\frac{87}{87 + f_y}\right) = \frac{55.4625\ B_1 f_c'}{f_y\ (87 + f_y)}$$

where

P_{max} = maximum reinforcement ratio

f_y = specified yield strength of reinforcement

$$\frac{a}{d_{max}} = P_{max} \frac{f_y}{0.85 f_c'}$$

where

a = height of equivalent rectangular stress block

d_{max} = maximum distance from extreme compression fiber to centroid of tension reinforcement

$$K_{u\ max} = 0.765\ f_c' \left(\frac{a}{d_{max}}\right)\left(1 - 0.5 \frac{a}{d_{max}}\right)$$

where

$K_{u\ max}$ = resistant moment coefficient

$$\frac{a}{d} = 1 - \sqrt{1 - \frac{M_u}{MR}\left(1 - 0.5 \frac{a}{d_{max}}\right) \frac{a}{d_{max}}}$$

where

d = distance from extreme compression fiber to centroid of tension reinforcement

M_u = factored design moment

MR = resistant moment of cross-section

$$A_u = \frac{0\ 9 f_y \left(1 - 0.5 \frac{a}{d}\right)}{12}$$

where

A_u = coefficient for steel calculation

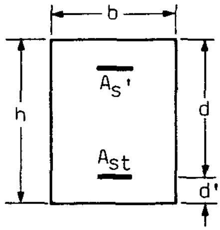
FOR MEMBERS WITH TENSION REINFORCEMENT ONLY:

$$A_s = \frac{M_u}{A_u\, d}$$

where

A_s = tension reinforcement

FOR MEMBERS WITH TENSION AND COMPRESSION REINFORCEMENT:



$$M_t' = \frac{0.9\, f_y\, (d - d')}{12}$$

where

M_t' = moment to be used for design of tension reinforcement on beams with compression and tension reinforcement

d' = difference between total height of cross-section and distance from extreme compression fiber to centroid of tension reinforcement

$$M_c' = \frac{0.9\, (f_y - 0.85\, f_c')\, (d - d')}{12}$$

where

M_c' = moment to be used for design of compression reinforcement

$$MR = K_{u\ max}\, bd^2$$

$$A_{st} = P_{max}\,bd + \frac{M'}{M_t'}$$

where

A_{st} = tension reinforcement for tension and compression reinforcement members

M' = difference between factored design moment and resistant moment of cross-section

$$A_s' = \frac{M'}{M_c'}$$

where

A_s' = compression reinforcement

$$d_{min} = \frac{12\,M_u}{K_{u\,max}\,b}$$

where

d_{min} = minimum distance from extreme compression fiber to the centroid of tension reinforcement

$$a = \frac{(A_{st} - A_s')\,f_y}{0.85\,F_c'\,b}$$

$$M_{(nominal)} = \frac{M_u}{0.9}$$

$$A_{s\,max} = P_{max}\,bd$$

$$\text{Tension} = \text{Compression} = \frac{M_u}{0.9z}$$

where

z = difference between the distance from extreme compression fiber to centroid of tension reinforcement (d) and one-half of the height of the equivalent rectangular stress block (a/2)

$$\text{Concrete Stress} = \frac{\text{Compression}}{0.85\,ab}$$

$$\text{Steel Stress} = \frac{\text{Tension}}{A_s}$$

(Reference: Paul Rogers, Reinforced Concrete Design for Buildings, Van Nostrand Reinhold Company, 1973.)

STEP 1: If the computer is not already in the BASIC mode, select TI BASIC. To load the program from a diskette, insert the diskette into the disk drive, type

OLD DSK1.CONCRETE

and press ENTER.

To load the program from a cassette tape, refer to the "Loading Cassettes" section in this manual for instructions on determining the program's position on the cassette tape. When you have inserted the cassette and positioned the tape counter, type

OLD CS1

and press ENTER. The computer then displays directions for loading the tape. Refer to "Loading Cassettes" if you have difficulty in loading the program from the cassette.

STEP 2: When the cursor reappears, type RUN, and press ENTER. The CONCRETE BEAM STRESS ANALYSIS title screen appears briefly, followed by a description of the program. Press ENTER to begin entering data.

STEP 3: Enter the following data: the compressive strength of the concrete, the yield strength of the reinforcement, the factored design load moment, the height of the cross section, the distance from the top to the reinforcement centroid, and the width of the cross section.

STEP 4: The program then displays the resistant moment of the cross section and the area of the tension steel. The program also determines whether compression steel is necessary. If it is, the compression steel area and block height are displayed. If compression steel is not necessary, the following values are displayed: the block height, the minimum value of d (the distance from extreme compression fiber to the centroid of tension reinforcement), the maximum steel area, the nominal moment, the concrete compression, the concrete stress, and the steel stress.

STEP 5: You may perform another analysis by entering Y to the prompt REDO? (Y/N). If you enter N, the message ** DONE ** is displayed, and the program stops.

Two examples are provided to help you understand how the prograı works. The first example does not require compression steel. The second example uses compression reinforcement. First, load and run the program as described in Steps 1 and 2 of the "User Instructions." Then begin entering data in response to the prompts for the following example.

Given:	M_u = 200 ft-kips	f_c' = 4 ksi
	f_y = 60 ksi	h = 22.5 in
	d = 20 in	b = 10 in

Compute the steel area for the beam; no compression reinforcement is used.

PROMPT	ENTER	COMMENTS
COMPRESSIVE STRENGTH OF CONCRETE(KSI):	4	Required data
YIELD STRENGTH OF REINFORCEMENT(KSI):	60	
FACTORED DESIGN LOAD MOMENT (FT-KIPS):	200	
H: TOTAL HEIGHT OF CROSS SECTION(IN.):	22.5	
D: DISTANCE FROM TOP TO REINFORCEMENT CENTROID(IN.):	20	
B: WIDTH OF CROSS SECTION (IN.):	10	

The following answers are displayed. (Answers are rounded to two decimal places.)

RESISTANT MOMENT OF CROSS SECTION FT-KIPS)	312.24
TENSION STEEL AREA(SQ. IN.)	2.34

----NO COMPRESSION STEEL----		No compression steel necessary
A, BLOCK HEIGHT(IN.)	4.13	Compression block height
D,MINIMUM(IN.)	16	
STEEL AREA-MAX(SQ. IN.)	4.27	Maximum steel area (tension)
NOMINAL MOMENT(FT-KIPS)	222.22	
TENS. =COMP. (KIPS)	148.7	Steel tension must equal concrete compression
CONC. STRESS(KSI)	4.23	Concrete stress
STEEL STRESS(KSI)	63.45	Steel stress

To calculate the second example, enter Y in response to the prompt REDO? (Y/N).

Compute the steel area for the beam with the following description; compression reinforcement is used.

Given: M_u = 360 ft-kips f_c' = 4 ksi
f_y = 60 ksi h = 22.5"
d = 20" b = 10"

Enter the data in response to the prompts shown below.

PROMPT	ENTER	COMMENTS
COMPRESSIVE STRENGTH OF CONCRETE(KSI):	4	Required data
YIELD STRENGTH OF REINFORCEMENT(KSI):	60	
FACTORED DESIGN LOAD MOMENT (FT-KIPS):	360	
H: TOTAL HEIGHT OF CROSS SECTION(IN.):	22.5	
D: DISTANCE FROM TOP TO REINFORCEMENT CENTROID(IN.):	20	
B: WIDTH OF CROSS SECTION (IN.):	10	

The following results are displayed. (Answers are rounded to two decimal places.)

RESISTANT MOMENT OF CROSS SECTION (FT-KIPS)	312.24	
TENSION STEEL AREA(SQ. IN.)	4.88	
COMPRESSION STEEL AREA(SQ. IN.)	.64	Area of upper steel membe (compression)
A: COMPR. BLOCK HEIGHT(IN.)	7.48	

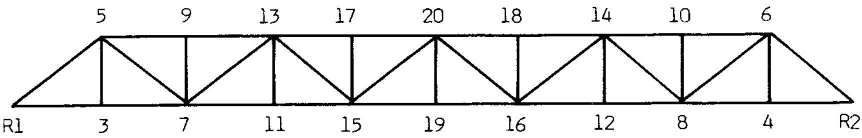
If no further calculations are desired, enter N in response to the prompt REDO? (Y/N). The message ** DONE ** is displayed, and the program stops.

The Trusses program determines the distribution of forces in three types of trusses--Warren, Howe, and Pratt--and labels them in terms of tension (+) or compression (-). You enter the number of bays (only an even number of bays greater or equal to four can be used), the width of the bay, the truss height, and the force at each joint. The program then calculates the force and the reactions for each horizontal, diagonal, or vertical member.

To enter the forces for each truss, use the following order, regardless of the number of bays or the type of truss.

1. Begin with joint 3
2. Enter left bottom
3. Enter right bottom
4. Enter left top
5. Enter right top
6. Repeat steps 2-5 for each additional bay

For example, in the ten-bay Warren truss drawn below, the joints are labeled in the order in which the forces should be entered.



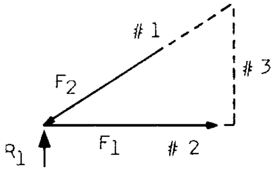
The program assumes that the only forces acting at the extreme left and right ends are the reaction forces R1 and R2. All positive forces are considered to be in a downward direction.

Bay width and truss height may be entered in either feet or meters as long as the units are consistent. Forces should be entered in units consistent with the length measurements. The sign convention of downward (+) and upward (-) is used for all upright trusses, but inverted trusses may be computed by using downward (-) and upward (+).

The program assumes that the truss is in equilibrium. It splits all diagonal forces into their vertical and horizontal components. It then cuts a free-body diagram at each joint and calculates the forces to equilibrium. All joints are pinned joints.

The following example is a free-body calculation.

Reaction at R1 is upward 20.250 Kips. It must be balanced by a downward force provided by the vertical component of member #1.



#1 = 20 ft
#2 = 16 ft
#3 = 12 ft
R1 = 20.25 K

Because the bay width (#2) is 16 ft and the truss height (#3) is 12 ft, the Pythagorean theorem may be used to find the ratio of member #1 to its vertical component.

$$(12)^2 + (16)^2 = 400; \text{ Member \#1} = \sqrt{400} = 20 \text{ ft.}$$

The relation of the vertical component to the member is 12 in 20. Thus, the force relation is:

$$\frac{F_2}{20} = \frac{20.250}{12}$$

Therefore, $F_2 = 33.750$

The member is in compression because it is pressing against the joint. The force in the compressed member #1 is -33.750 Kips.

STEP 1: If the computer is not already in the BASIC mode, select TI BASIC. To load the program from a diskette, insert the diskette into the disk drive, type

OLD DSK1.TRUSS

and press ENTER.

To load the program from a cassette tape, refer to the "Loading Cassettes" section in this manual for instructions on determining the program's position on the cassette tape. When you have inserted the cassette and positioned the tape counter, type

OLD CS1

and press ENTER. The computer then displays directions for loading the tape. Refer to "Loading Cassettes" if you have difficulty in loading the program from the cassette.

STEP 2: When the cursor reappears, type RUN, and press ENTER. The TRUSSES title screen appears briefly, followed by a list of options.

1 WARREN TRUSS SOLUTION
2 HOWE TRUSS SOLUTION
3 PRATT TRUSS SOLUTION
4 EXIT

STEP 3: Select the type of truss you wish to analyze. (Each option operates in the same manner.) Once you have selected an option, a brief description of the option's function is displayed. Press ENTER to begin entering data.

STEP 4: Enter the number of bays, the width of the bay, the truss height, and the forces at each joint, beginning with joint 3.

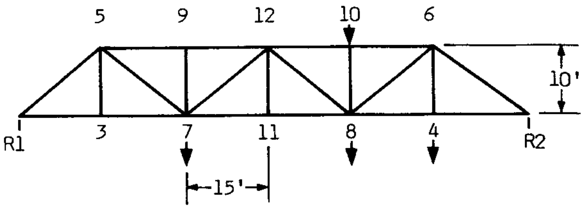
STEP 5: The program performs its calculations and then displays the left and right reactions and the forces on the horizontal, diagonal, and vertical members.

STEP 6: After the results have been displayed, the prompt PRESS REDO, BACK is displayed. If you press SHIFT R (REDO), you may enter new data for the same type of truss. If you press SHIFT Z (BACK), you return to the list of options. You may then evaluate another type of truss or select option 4 to exit the program.

To help you understand how the program works, an example is provided for each type of truss. First, load and run the program as described in Steps 1 and 2 of the "User Instructions." To calculate the forces on the following six-bay Warren truss, select Option 1.

WARREN TRUSS

Truss Height = 10 ft.
Bay Width = 15 ft.
Joint 4 = 100lbs.
Joint 7 = 150 lbs.
Joint 8 = 150 lbs.
Joint 10= 225 lbs.



After the program description is displayed, press ENTER. Then, enter the data shown below:

PROMPT	ENTER	COMMENTS
NUMBER OF BAYS(4-10):	6	Must be an even number from 4 through 10.
BAY WIDTH:	15	Use English or metric
H, TRUSS HEIGHT:	10	measurements as long as they are consistent.
FORCE 3:	0	Enter forces beginning
FORCE 4:	100	with joint #3.
FORCE 5:	0	
FORCE 6:	0	
FORCE 7:	150	
FORCE 8:	150	
FORCE 9:	0	
FORCE 10:	225	
FORCE 11:	0	
FORCE 12:	0	

The following answers are displayed.

R1: LEFT REACTION	241.66666	Computed forces at each
R2: RIGHT REACTION	383.33333	end
MEMBER 3-5	0	Vertical members
MEMBER 4-6	100	
MEMBER 7-9	0	
MEMBER 8-10	-225	
MEMBER 11-12	0	

MEMBER 1-5	-435.6707791	Diagonal members
MEMBER 5-7	435.6707791	
MEMBER 7-12	-165.2544335	
MEMBER 2-6	-691.0639945	
MEMBER 6-8	510.7864307	
MEMBER 8-12	165.2544335	

Press ENTER to display the remaining calculations.

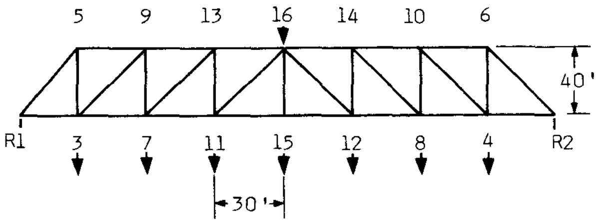
MEMBER 1-3	362.5	Lower horizontal members
MEMBER 3-7	362.5	
MEMBER 7-11	862.5	
MEMBER 2-4	575	
MEMBER 4-8	575	
MEMBER 8-11	862.5	
MEMBER 5-9	-725.	Upper horizontal members
MEMBER 9-12	-725.	
MEMBER 6-10	-1000.	
MEMBER 10-12	-1000.	

Next press SHIFT Z (BACK) to return to the list of options.

Select Option 2 to calculate the forces on the following eight-bay Howe truss.

HOWE TRUSS

Bay Width = 30 ft.
Truss Height = 40 ft.
Joint 3 = 100 lbs
Joint 4 = 100 lbs
Joint 7 = 150 lbs
Joint 8 = 150 lbs
Joint 11 = 75 lbs
Joint 12 = 75 lbs
Joint 15 = 25 lbs
Joint 16 = 400 lbs



After the program description is displayed, press ENTER. Then enter the data shown below:

PROMPT	ENTER	COMMENTS
NUMBER OF BAYS(4-10):	8	Must be an even number from 4 through 10.
BAY WIDTH:	30	Use English or metric
H, TRUSS HEIGHT:	40	measurements as long as they are consistent.
FORCE 3:	100	Enter forces beginning
FORCE 4:	100	with joint #3.
FORCE 5:	0	
FORCE 6:	0	
FORCE 7:	150	
FORCE 8:	150	
FORCE 9:	0	
FORCE 10:	0	
FORCE 11:	75	
FORCE 12:	75	
FORCE 13:	0	
FORCE 14:	0	
FORCE 15:	25	
FORCE 16:	400	

The following answers are displayed.

R1: LEFT REACTION	537.5	Computed forces at each end
R2: RIGHT REACTION	537.5	
MEMBER 1-3	403.125	Horizontal members
MEMBER 5-9	-403.125	
MEMBER 3-7	731.25	
MEMBER 9-13	-731.25	
MEMBER 7-11	946.875	
MEMBER 13-16	-946.875	
MEMBER 11-15	1106.25	
MEMBER 2-4	403.125	
MEMBER 6-10	-403.125	
MEMBER 4-8	731.25	
MEMBER 10-14	-731.25	
MEMBER 8-12	946.875	
MEMBER 14-16	-946.875	
MEMBER 12-15	1106.25	
MEMBER 1-5	-671.875	Diagonal members

Press ENTER to display the remaining calculations.

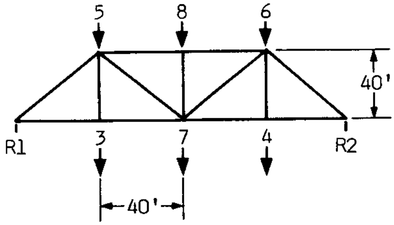
MEMBER 3-9	-546.875	
MEMBER 7-13	-359.375	
MEMBER 11-16	-265.625	
MEMBER 2-6	-671.875	
MEMBER 4-10	-546.875	
MEMBER 8-14	-359.375	
MEMBER 12-16	-265.625	
MEMBER 3-5	537.5	Vertical members
MEMBER 7-9	437.5	
MEMBER 11-13	287.5	
MEMBER 4-6	537.5	
MEMBER 8-10	437.5	
MEMBER 12-14	287.5	
MEMBER 15-16	25	

Now press SHIFT Z (BACK) to return to the list of options.

Select Option 3 to calculate the forces on the following four-bay Pratt truss.

PRATT TRUSS

Bay Width = 40 ft.
Truss Height = 40 ft.
Joint 3 = 175 lbs
Joint 4 = 220 lbs
Joint 5 = 400 lbs
Joint 6 = 370 lbs
Joint 8 = 180 lbs



After the program description is displayed, press ENTER. Then enter the data shown below:

PROMPT	ENTER	COMMENTS
NUMBER OF BAYS(4-10):	4	Must be an even number from 4 through 10.
BAY WIDTH:	40	Use English or metric
H, TRUSS HEIGHT:	40	measurements as long as they are consistent.
FORCE 3:	175	Enter forces beginning
FORCE 4:	220	with joint #3.
FORCE 5:	400	
FORCE 6:	370	
FORCE 7:	200	
FORCE 8:	180	

The following answers are displayed.

R1: LEFT REACTION	768.75	Computed forces at each
R2: RIGHT REACTION	776.25	end
MEMBER 1-3	768.75	Horizontal members
MEMBER 3-7	768.75	
MEMBER 5-8	-962.5	
MEMBER 2-4	776.25	
MEMBER 4-8	776.25	
MEMBER 6-8	-962.5	

MEMBER 1-5	-1087.176676	Diagonal Members
MEMBER 5-7	274.0038777	
MEMBER 2-6	-1097.783278	
MEMBER 6-7	263.397276	
MEMBER 3-5	175	Vertical members
MEMBER 4-6	220	
MEMBER 7-8	-180	

To return to the list of options, press SHIFT Z (BACK). Then select option 4 to exit the program. The message ** DONE ** is displayed and the program stops.

The Four-Span Distribution program uses the moment-distribution method of analysis, an iterative procedure, to determine final bending moments at supports for continuous beams or girders with five supports. The loading conditions can be uniform, concentrated, or combined and may have variable end conditions. This program can be used only with prismatic members which have the constant I throughout the length.

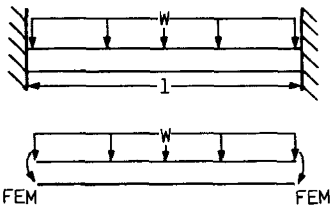
To perform the calculation, you enter the following data: moments for each span, the continuous load for each span, the length of the span, the concentrated load for each span, its distance from the span's left edge, and any additional loads.

Numeric data can be entered in integer or real form. The program retains all significant digits during calculations although the final answer is rounded.

FOUR-SPAN DISTRIBUTION — Background

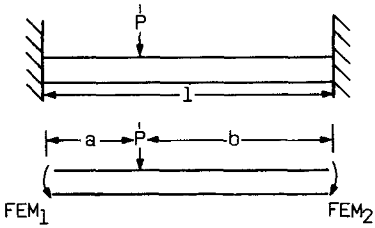
The moment-distribution method analyzes statically indeterminate beams and frames by relatively simple calculations. This iterative procedure assumes that all joints of the structure are temporarily restrained against displacement and evaluates the moments (called fixed end moments or FEM's) in the members corresponding to this condition. After a number of cycles of releasing the joints and distributing the member moments, the true values of moments in the members are obtained.

DISTRIBUTED LOAD



$FEM = wl^2 : 12$

CONCENTRATED LOAD



$FEM_1 = Pab^2 : l^2$

$FEM_2 = Pa^2b : l^2$

When a joint is unlocked, it rotates if the algebraic sum of the FEM's acting on the joint does not equal zero; this resulting moment acting on the joint is called the unbalanced moment. When the unlocked joint rotates under this unbalanced moment, end moments are developed in the ends of the members meeting at the joint. These finally restore equilibrium at the joint and are called distributed moments. After release, end moments are likewise developed at the far ends of each member; these are called carry-over moments.

STEP 1: If the computer is not already in the BASIC mode, select TI BASIC. To load the program from a diskette, insert the diskette into the disk drive, type

OLD DSK1.4SPAN

and press ENTER.

To load the program from a cassette tape, refer to the "Loading Cassettes" section in this manual for instructions on determining the program's position on the cassette tape. When you have inserted the cassette and positioned the tape counter, type

OLD CS1

and press ENTER. The computer then displays directions for loading the tape. Refer to "Loading Cassettes" if you have difficulty in loading the program from the cassette.

STEP 2: When the cursor reappears, type RUN, and press ENTER. The FOUR-SPAN DISTRIBUTION title screen appears briefly, followed by a description of the program. Press ENTER to begin entering data.

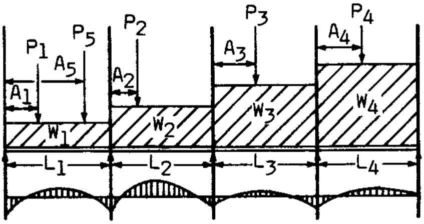
STEP 3: Enter the following data: moments for each span, continuous load and length for each span, concentrated load for each span and its distance from the span's left edge. (Refer to the graph displayed on your screen for assistance in entering data.) If you have more than one concentrated load per span, you may enter the second load for each span after responding Y to the prompt MORE THAN ONE CONCENTRATED LOAD PER SPAN? (Y/N). If a particular span does not have a load, enter zero. Enter all additional loads in this manner. When you have entered all concentrated loads, enter N to the prompt MORE THAN ONE CONCENTRATED LOAD PER SPAN? (Y/N).

STEP 4: Computing the distributed moments for the fixed ends begins and may take 15-20 seconds. The results are rounded to two decimal places.

STEP 5: After the moments are displayed, you may calculate new moments for released end moments or cantilevered ends by responding Y to the prompt DO YOU WISH TO RELEASE END MOMENTS AND/OR HAVE CANTILEVER ENDS? (Y/N). Then, enter the new moments for the left and right ends. The moments are calculated and displayed.

STEP 6: If you respond N to the prompt above or after you have calculated the additional moments, the prompt REDO? (Y/N) is displayed. If you wish to make additional calculations, enter Y. If you enter N, the message ** DONE ** is displayed, and the program stops.

By working the following example, you can better understand how the program operates.



I_1 = 40 ft-kips
I_2 = 54 ft-kips
I_3 = 47.5 ft-kips
I_4 = 30 ft-kips

W_1 = 2 kips
W_2 = 3 kips
W_3 = 4 kips
W_4 = 5 kips

L_1 = 20 ft
L_2 = 18 ft
L_3 = 19 ft
L_4 = 20 ft

P_1 = 10 kips
P_2 = 8 kips
P_3 = 0
P_4 = 12 kips
P_5 = 5 kips

A_1 = 8 ft
A_2 = 9 ft
A_3 = 0
A_4 = 12 ft
A_5 = 12 ft

First, load and run the program as described in Steps 1 and 2 of the "User Instructions." Then enter the data in response to the prompts.

PROMPT	ENTER	COMMENTS
I1:	40	Enter moments of inertia
I2:	54	for each span.
I3:	47.5	
I4:	30	
LOAD(W1):	2	Enter continuous loads
LENGTH(L1):	20	and length for each span.
LOAD(W2):	3	
LENGTH(L2):	18	
LOAD(W3):	4	
LENGTH(L3):	19	
LOAD(W4):	5	
LENGTH(L4):	20	

FORCE(P1):	10	Enter concentrated loads and the distance from the span's left edge. When more than one concentrated load occurs on a span, enter the first load only.
DISTANCE(A1):	8	
FORCE(P2):	8	
DISTANCE(A2):	9	
FORCE(P3):	0	
DISTANCE(A3):	0	
FORCE(P4):	12	
DISTANCE(A4):	12	
MORE THAN ONE CONCENTRATED LOAD PER SPAN? (Y/N)	Y	Enter Y to enter the second loads.
FORCE(P1):	5	Enter the second loads; enter zero for a no-load condition.
DISTANCE(A1):	12	
FORCE(P2):	0	
DISTANCE(A2):	0	
FORCE(P3):	0	
DISTANCE(A3):	0	
FORCE(P4):	0	
DISTANCE(A4):	0	
MORE THAN ONE CONCENTRATED LOAD PER SPAN? (Y/N)	N	All loads have been entered; computation takes 15-20 seconds.

The following results, which have been rounded to two decimal places, are displayed.

DISTRIBUTED MOMENTS FOR
FIXED ENDS

MOMENT A:	-105.33
MOMENT B:	-99.76
MOMENT C:	-98.64
MOMENT D:	-163.7
MOMENT E:	-214.24

To calculate new moments for released end moments or cantilevered ends, enter the following data:

PROMPT	ENTER	COMMENTS
DO YOU WISH TO RELEASE END MOMENTS AND/OR HAVE CANTILEVER ENDS?(Y/N)	Y	Allows you to calculate free or cantilever ends.

NEW MOMENTS AT "A" (LEFT END)	-300	Input new end moments.
NEW MOMENTS AT "E" (RIGHT END)	0	

The following new moments are calculated and displayed:

MOMENT B:	-42.46	New moments.
MOMENT C:	-93.13	
MOMENT D:	-231.68	

To end calculations and exit the program, press N in response to the prompt REDO? (Y/N). The message ** DONE ** is displayed, and the program stops.