Overview of a method that can help select and refine the optimum human-powered vehicle frame design

by Hei Wei (Don) Chan

INTRODUCTION

A human-powered-vehicle designer is often faced with the challenge of designing a vehicle frame from scratch, with little or nothing existing previously to draw as a reference. To have to actually build everything that looks suitable may be prohibitively expensive and very time consuming. What is needed is a "quick and dirty" method to compare frames with different geometries and dimensions to help refine and select the optimum design.

The following is an overview of one "quick and dirty" method called formally, "matrix structural analysis using the displacement method" or more simply, the "finite-element method (FEM)". The method described here is a particular subset of finite-element analysis where the elements are *complete beams, struts or tubes* instead of small but finite pieces in continua. It is particularly suitable for use on a microcomputer and the computing time of the program is usually less than a minute after all the inputs are keyed in.

MODELING OF THE VEHICLE FRAME

To use the program, one needs to establish a "matchbox" model of the vehicle frame and to determine the loading conditions. A "matchbox" model is one consisting solely of uniform beams, struts or tubes as elements, set in between load-bearing nodes. For this FEM program, elements have to have equal bending stiffness in all directions. In addition, the torsional stiffness of each element is expressed in terms of GI_p for a circular section and, therefore, non circular sections will have to use an"equivalent" GI_p. G is the shear elastic modulus and I_p is the polar moment of inertia.

WHAT THE PROGRAM LOOKS LIKE TO THE USER

Once activated, the FEM program will ask as inputs the number of nodes and elements. For every node, it will ask whether each of the six degrees of freedom (DOF) is active or inactive. Active means the node can be moved along that DOF and inactive means the node is fixed in that DOF. The six DOF are the orthogonal axes x, y, z and the three rotational axes about these orthogonal axes. The program will also ask for the external forces applied in the x, y, and z directions and the external moments about the three orthogonal axes. Finally, it will ask for the coordinates of the node.

For every element, the program will ask between which two nodes the element is situated and the EA, EI and GI_p of the element. E stands for the elastic modulus, A the cross-sectional area, I the bending moment of inertia, I_p the polar moment of inertia and G, the shear elastic modulus.

To accomodate situations where some of the elements are force fitted or heated or cooled to be fitted onto the frame, the program will ask for the initial mechanical and thermal strain.

After all the above information is obtained, the program calculates the displacements of the six DOF of each node. The forces and moments at the two ends of each element are also calculated and displayed.

If the same loading conditions are used to test several frame designs, the stiffer one is simply the one with the least linear or angular displacements in the various degrees of freedom at the nodes of concern. Besides comparing stiffness, this program is also useful for gaining insight into how loads are distributed throughout the frame. With some additional calculations, we can even obtain values of stresses at the ends of the elements. This information can help detect possible failure of joints at the loading conditions we are dealing with.

AN EXAMPLE OF ACTUAL USAGE

To illustrate the complete procedure of using this method, I cite my attempt to design a stiffer tandem-bicycle frame. The intention of this project is to increase the overall stiffness through the use of a different frame configuration and different tube dimensions while keeping the material and the weight the same as a "reference frame". The reference frame is a commercial aluminum tandem frame on which I have acquired information about its geometry and its tube dimensions.

I start by establishing the loading conditions I wish to test the frame in. In my case, I determine the typical maximum values of forces and moments applied to every node of the tandem-frame model in three situations: an out-of-saddle-sprinting situation, a steady-pedaling situation and a frontal-impact situation.

I obtained the values for the forces and moments through a variety of ways: *general industrial standards*, as in "a strong healthy individual can exert a force of up to two-and-a-half times his own body weight on the pedal"; *calculations*, as in "the force on the chain is equal to the force on the pedal multiplied by the length of the crank and divided by the radius of the chainring"; and *estimations*, like "the instantaneous maximum pull on the handlebar from a very strong rider would be slightly over 100 lbf".

The next step is to model a tandem bicycle frame as a space frame consisting entirely of nodes and elements (Figure 1).

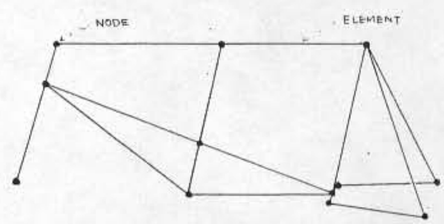

Figure 1. A matchbox model of a tandem frame: 12 nodes, 17 elements

Loads will have to be assigned to the respective nodes and the rest of the computer input, like the coordinates and the degrees of freedom of the nodes, EA, EI, GI_p of the elements etc., will have to be determined. Here is where the designing comes in. Besides the several nodes that are common and essential in all my tandem designs, the placement of the rest of the nodes and elements as well as the element dimensions are entirely up to my whim. The whole process is basically one of trial, error and luck. For those interested, the common nodes in my tandem-frame models are the front-axle and rear-axle nodes, the stem node, the two saddle nodes and the two bottom-bracket nodes.

I run the program and first obtain the output for the reference frame. I then rerun the program and obtain an output of my design frame. I compare the displacements in the 6 DOF of all the common nodes and note the displacements on my frame that are an arbitrary 10% more and 10% less than those on the reference one. I repeat the procedure for a second design and a third design . . . and so on. I continue until the number of DOF that are stiffer is more than the number of DOF that are less so by a happy amount.

For that stiffest frame, I will proceed to test for fatigue failure using the forces and moments at the ends of the elements as calculated by the computer program. If the stresses are too high, I will have to modify the design and rerun the program.

Note that this program does not take into account fillet radius and stress concentration and therefore a safe matchbox model does not guarantee a safe real-life frame. However, a safe matchbox frame is a better start than an unsafe matchbox frame to base a prototype design upon.

FINITE-ELEMENT METHOD, THE APPROACH

The mechanics of the FEM program relies basically on the equation:

stiffness x displacement = force applied.

Let me explain this using a two-dimensional truss element as an example (Figure 2).

Every node of this truss element has two degrees of freedom, one in the x-direction and one in the y-direction. The element in our example has one node at each of its ends and therefore a total of four degrees of freedom possible. A force

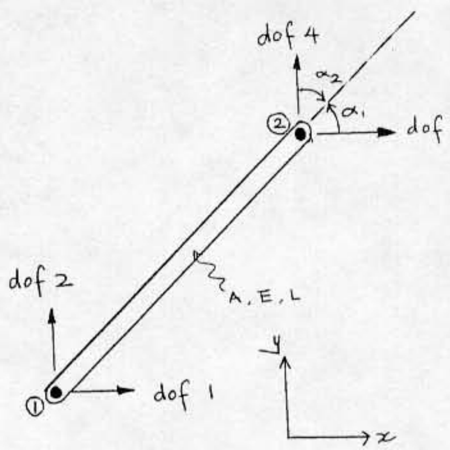

Figure 2. A 2-D truss element

applied in the direction of any one of these DOF will have an effect on all DOF throughout the element.

Now let us assume only DOF4 in our truss element is active and we apply a force F that has a horizontal component $F_x = F\cos\alpha_1$ in the direction of DOF3. We can write a stiffness coefficient k_{34} where

$$k_{34} = \frac{\text{force along DOF3}}{\text{unit displacement of DOF4}}$$

To find the value of k_{34}, we give DOF4 a unit displacement. In order for DOF4 to have a unit displacement, the truss element needs to extend a length of

$d = 1\cos\alpha_2$ (Figure 3). For the truss element to extend that much, we need a force $F = AEd/L = AE\cos\alpha_2/L$ where L=length of the element, A=cross-sectional area and E=modulus of elasticity.

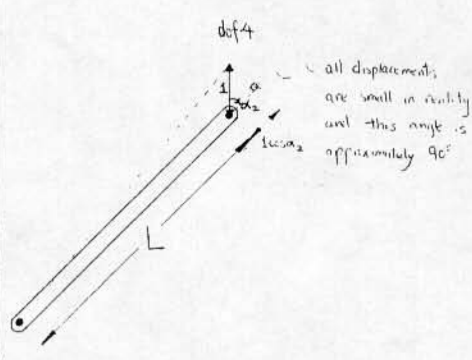

Figure 3. Truss element has to extend $1 \cdot \cos\alpha_2$ for a unit displacement in the direction of DOF 4

The component of this F in the direction of DOF3 is $F\cos\alpha_1$ and therefore,

$$k_{34} = F\cos\alpha_1/1 = AE\cos\alpha_1\cos\alpha_2/L$$

Having found k_{34}, we know that if there is a force in DOF3 that measures R_3, there will be a displacement of $d_3 = R_3/k_{34}$.

Since forces and displacements are additive, if DOF2 is also active, this force R_3 will have to be shared by both DOF2 and DOF4. We will get an equation:

$$R_3 = k_{34}d_4 + k_{32}d_2$$

If all DOF are active, we will get the equation:

$$R_3 = k_{31}d_1 + k_{32}d_2 + k_{33}d_3 + k_{34}d_4$$

Actually even when some DOF are inactive, we can still write the equation this way. The reason is inactive DOF have $d_i=0$ and their terms will drop out of the equation naturally.

It is now obvious that by calling a force component in the direction of DOF n, R_n, and repeating the above argument for forces in all DOFs, we can obtain a set of linear equations in R_i, k_{ij} and d_j. Sets of linear equations lend themselves easily to matrix representation and in our case, we have

$$[R] = [k][d]$$

k_{34} is $AE\cos\alpha_1\cos\alpha_2/L$. Doing the analysis that we did for k_{34} for all other k_{ij}'s will get us the contents of the matrix [k]:

$$[k] = AE/L \times \begin{bmatrix} C_1C_1 & C_1C_2 & -C_1C_1 & -C_1C_2 \\ C_2C_1 & C_2C_2 & -C_2C_1 & -C_2C_2 \\ -C_1C_1 & -C_1C_2 & C_1C_1 & C_1C_2 \\ -C_2C_1 & -C_2C_2 & C_2C_1 & C_2C_2 \end{bmatrix}$$

where $C_1 = \cos\alpha_1$, and $C_2 = \cos\alpha_2$

This matrix is what we call the "local stiffness matrix". It is a matrix that belongs to one truss element. The next step is to assemble local matrices for all the elements we are dealing with and combine them to form a single "global stiffness matrix".

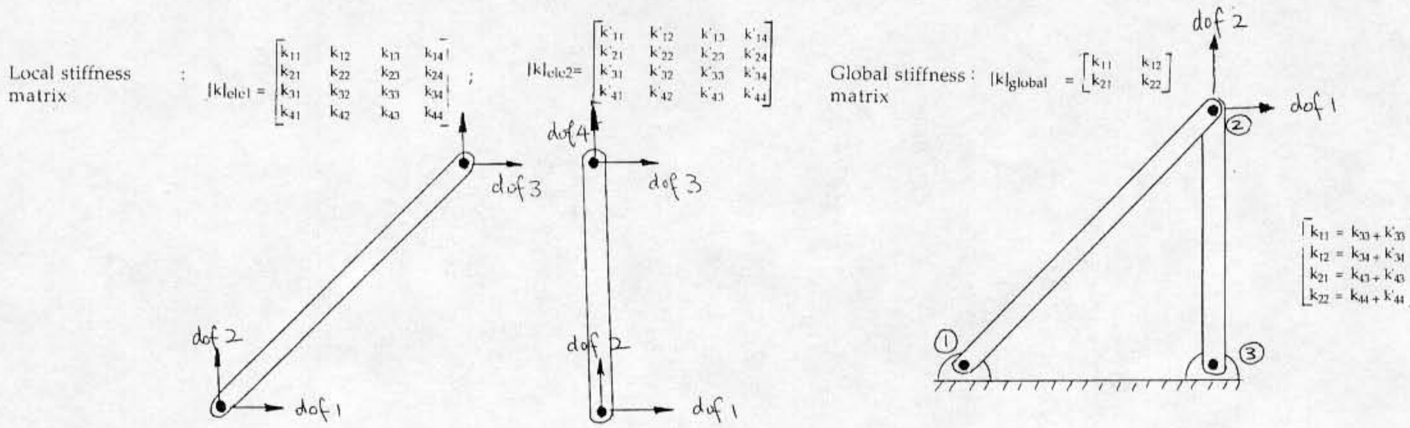

Figure 4. How local matrices combine to form the global matrix in a truss

Assume we have a truss with two elements as shown in Figure 4. Node 1 and node 3 are pinned to the ground and are completely fixed in position. Node 2 is active in two degrees of freedom. In assembling a global stiffness matrix, we ignore DOF that are completely inactive. Hence we have a 2x2 global stiffness matrix in this example. The entries for this matrix are simply the sum of the entries of the corresponding local stiffness matrices (Figure 4).

With this global matrix known and the external forces given, we can calculate the displacements of all active DOF. The computer program uses gaussian elimination to find the solution of the displacement matrix. Knowing all displacements, we go back to the local stiffness matrix and calculate the forces at each DOF using simple matrix multiplication.

In three-dimensional truss and beam problems, the only major differences are larger local stiffness matrices (12x12) because of a greater number of DOF and the use of several different equations for stiffness—e.g. in torsion we use

$$\text{stiffness} = M_t/\phi = GI_p/L$$

where M_t is the torsional moment and ϕ is the angle of twist.

The basic procedure, however, of first finding the local and global stiffness matrices and then the global displacement matrix and finally the local forces remains unchanged.

FURTHER INFORMATION

This finite-element method is very well documented in the book written by Nathan H. Cook called *Mechanics and Materials for Design*, 1984, McGraw-Hill Inc. This book not only contains a detailed description of FEM in chapters 4, 9 and 11, but also has a complete listing of the computer program in BASIC from p. 359 to p. 368.

If you are interested in obtaining a copy of the program for your IBM PC or compatible machine on a 5" diskette, please send $10.00 payable to the IHPVA to David Gordon Wilson, 21 Winthrop St., Winchester, MA 01890, USA.

This program is for private use only and should not be copied for commercial distribution.

Don Chan
63 Bowdoin St.
Newton Highlands, MA 02161
USA

(Don Chan did this summer project as an MIT junior in mechanical engineering. He is currently (1989) working on his senior thesis at Cannondale. Don is from Hong Kong—ed.) ❑

Daedalus—the aircraft

(continued from page 5)

Mark Drela stressed the importance of the seats for so long a flight. The pilots did some of their training on a Ryan Vanguard recumbent, and on a recumbent ergometer, but used their regular bikes for much of their conditioning.

Their long-distance stamina was improved through the development of a glucose-polymer salt-water mixture by Ethan Nadel of Yale, who worked with Steve Bussolari on the human factors of the flight. The drink inevitably became known as "Ethan-ol". One of the tangible results of this remarkable flight may be the commercial development of this drink.

In the question period, Mark was asked about the final crash of Daedalus. He said that the plane could not have landed in any case: a crash was inevitable. The black pebble beach was almost too hot for bare feet. The ocean was very cold. A roller convection cell developed that imposed "g" forces much higher than those for which the plane was designed. The same type of crash had happened to the other Daedalus aircraft when it crashed over the Mojave desert last year.

—Reported by Dave Wilson

❑