Massachusetts Institute of Technology
Department of Aeronautics and Astronautics
Cambridge, MA 02139

16.01/16.02 Unified Engineering I, II
Fall 2003

Systems Problem 9

Name: _____

Due Date: 11/6/03

	Time Spent (min)
SP9	
Study Time	

Announcements: NOTE: There is no SP8

Systems Problem #9

The Truss

Issued October 30 2003   Due 5pm November 6 2003

Objectives

After completing this laboratory exercise you should be able to:

• Perform an analysis of a two dimensional truss under applied loads

• Verify the calibration of a laboratory instrument

• Use calibration data to convert measured micro-strain readings into corresponding forces

• Execute an empirical study to relate applied loads to forces in truss members

• Write a computer code to perform an automatic analysis of an arbitrary truss

• Compare and contrast the use of analytical and experimental models

Overview

A three-dimensional truss is tested under transverse loading and the load-deflection characteristics and the internal bar/member strains are measured. From the latter, the internal bar/member loads are determined. These latter derived results are compared to results obtained from analysis using a two-dimensional model of the truss. The deflection results and the strain results are retained for use later in Unified when these concepts are introduced in lecture. Thus, comparison can be made between the model of the truss and the performance of an actual truss and therefore the validity of some of the assumptions made in forming the model can be examined.

In performing the laboratory, each student is introduced to a number of types of instrumentation and associated concepts. These are strain gages and the measurement of strain; deflection meters (dial gauges); loads cells; load-displacement relations; and the use of experimental/empirical calibration to obtain indirect measurements of one parameter via the direct measurement of another parameter (in this case member loads via member strains).

In addition to the laboratory work, each student will generate a computer code, from a template provided, that will perform automatic calculation of the bar forces in a truss, given an arbitrary applied loading. This will enforce the concept of the utility of computational methods to solve engineering problems

Background

As was discussed in lecture, trusses are used for a number of different structures. They are structurally very efficient (i.e. they can carry high loads for a given mass of structure), and so they are widely used for space structures, and were used for early airframes. In the context of Unified Engineering they are also useful to us in introducing you to structural analysis as they are relatively simple structures to analyze. However, it is important to realize, that for trusses, and for any other structure that we design or analyze, the analysis is performed on an idealization of the actual, physical structure. In conducting any structural analysis it is crucial at some point to assess whether the modeling assumptions required to perform the analysis are justified. This laboratory exercise will give you a chance to explore the assumptions we have made in order to analyze trusses.

Truss Model
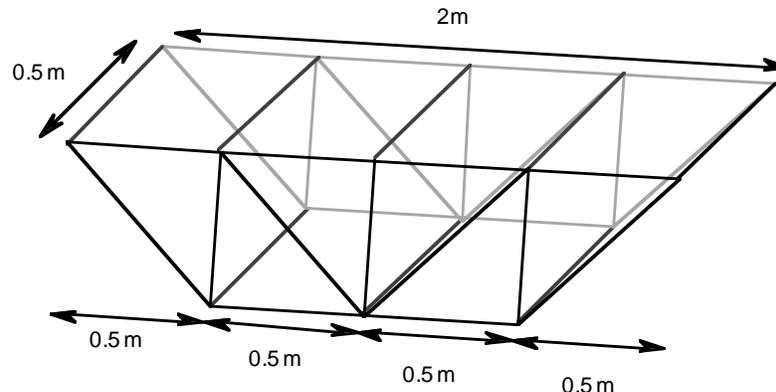
The basic truss configuration, is shown in figure 1

Figure 1.   Illustration of the truss configurations used in the truss lab

The Analytical Model

In order to perform an analysis, the real physical device must be modeled such that the *analytical model* can be relatively simply analyzed but the model must also capture the essential physics of the situation. In this case, it is assumed that the three-dimensional connections are such that the two two-dimensional trusses are loaded evenly.

In addition the normal assumptions associated with "idealized planar (two-dimensional) trusses"

1. All bars are straight
2. Bar joints are frictionless pins
3. Bars are massless and perfectly rigid for loading analysis
4. All loads and reactions are applied at the joints
5. Loads in members are colinear
6. Thus, bars carry only axial forces

This results in the analytical model configuration shown in Figure 2. You should have analyzed this truss in problem set question M7 before coming to the lab to perform the experiment.
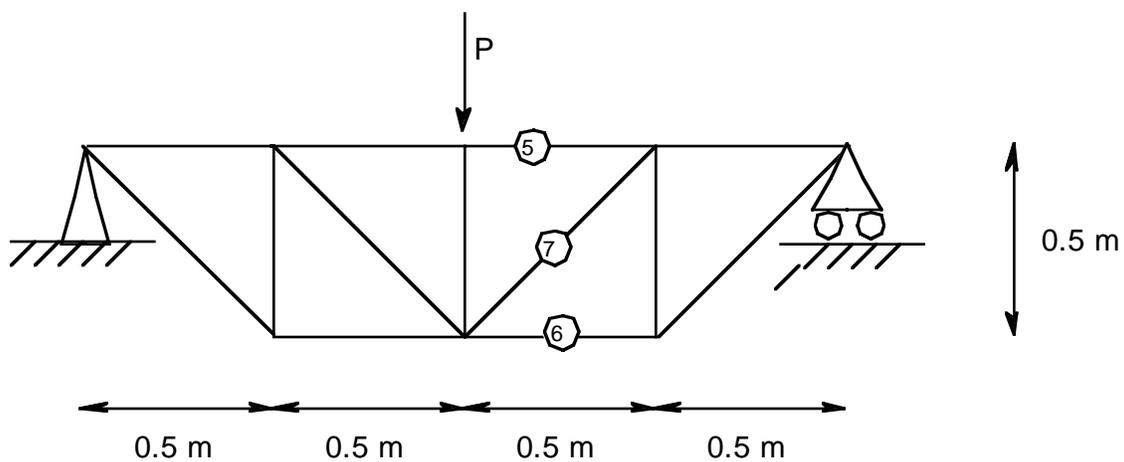

Figure 2.　　Illustration of two-dimensional model of truss to be utilized for analysis.

Experimental Apparatus and Setup

There are three separate sets of experimental apparatus. There are two sets of apparatus corresponding to the truss shown in figure 1. One has steel bars and the other has aluminum bars.

Loading Setup

The experimental trusses are supported on a pair of unistrut supports with a horizontal separation of 2 meters as shown in Figure 3.
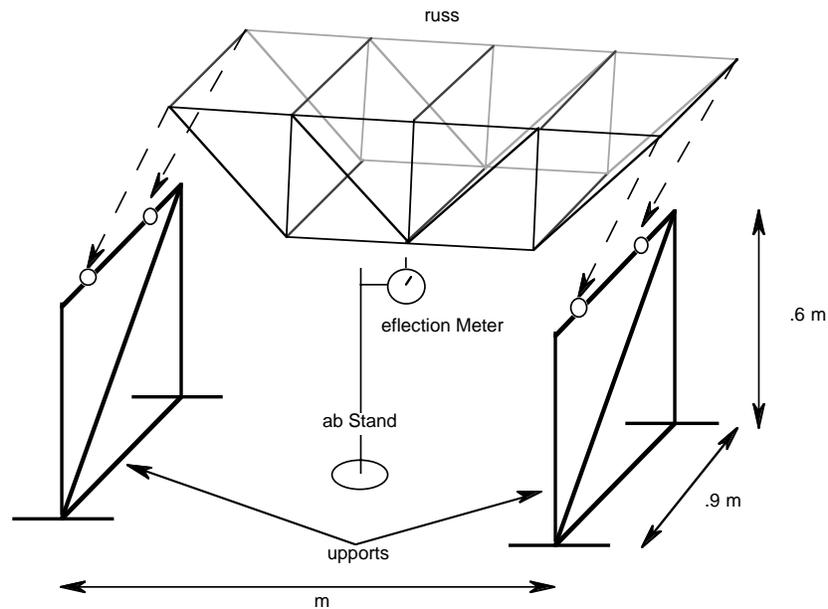
Figure 3. Illustration of experimental supports and truss configuration for truss. Note that the dashed arrows indicate where the truss will actually rest on the supports.

The truss is loaded using the jack and pulley set-up shown in Figure 4. This set-up is placed at the joints between the second and third bay so that the load is applied at the same point as shown in Figures 2.
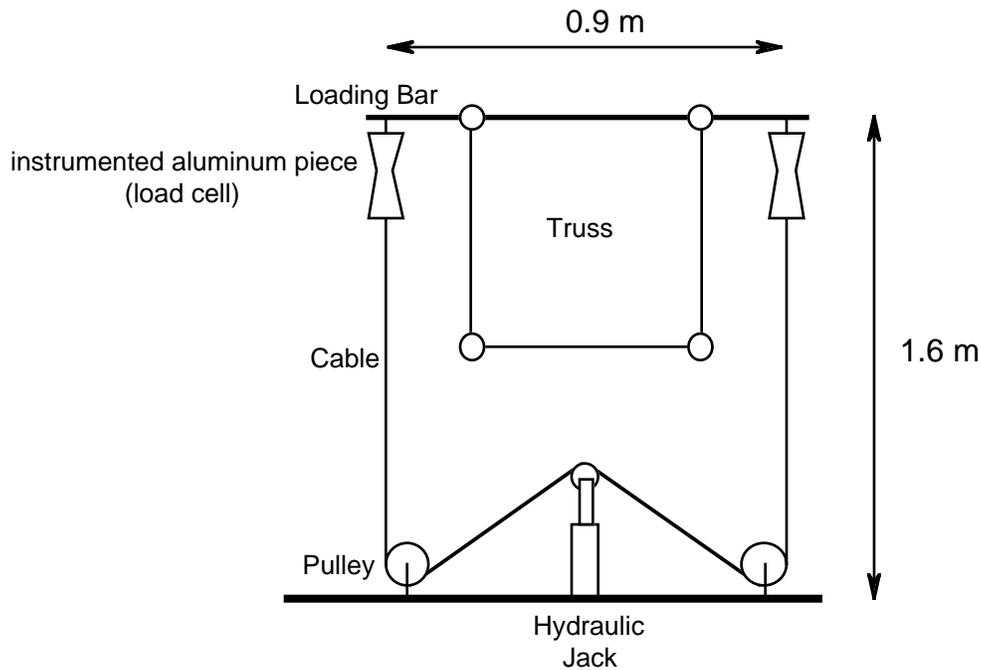
Figure 4.   Cut-away side view, at loading point, of experimental loading configuration
with jack and cable pulley set-up

Load Cells and Instrumented Bars

In this experiment we will measure the load applied to the overall structural
configuration as well as load in specific members of the truss. This is done via an *indirect
measurement and calibration technique*.

For the truss, the two *load cells* (labeled 8 and 9) are attached to the loading bar and
to the loading cable and are thus in the line of load on each end. The load cells are
instrumented with strain gauges and give a direct reading of the strain. Prior to the assembly
of the test setup, the load cells were calibrated by hanging known weights from them and
reading the output from the strain gauges. A calibration curve of load versus microstrain was
thus established. In this way, readings of the strain can be taken and the load on the bar
determined from the calibration curve. This, then gives the load on that side of the truss.
This is the *indirect measurement and calibration technique - we do not directly measure
forces, but infer them from the resultant strain*.

It is important in using such a technique to occasionally check that the calibration is still
valid. This will be done in each case for the load cells.

Three of the truss members (labeled 5, 6, and 7 in Figure 2) have also been
instrumented to allow you to read the strain that each member experiences when the truss

is loaded. These instrumented members were also calibrated in the same way. Once again, strain will be directly measured and load in the members determined from the calibration curves, one for each instrumented member. In this case, since the truss is already assembled, calibration will not be able to be checked.

Item	mV per lb (positive = tension)
Member #5	0.1
Member #6	0.1
Member #7	0.1
Load Cell #8	1.0
Load Cell #9	1.0

Table: Calibration constants for bars and load cells. Note do not alter the gain on the strain gauge amplifiers - this will change the calibration factor and render your results useless!.

Strain Gages and Strain Measurement

Strain gages are basically long pieces of a conductor in the configuration shown in Figure 6. the conductor is on a carrier which allows application to the material of interest via an adhesive.
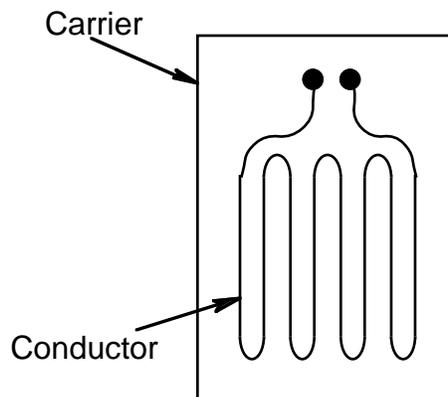

Figure 6.   Illustration of strain gage

As the base material, to which the strain gage is bonded, changes length, the length of the strain gage changes. This changes the resistance of the strain gage material (as you learned in Physics). A calibration has been done by the manufacturer to determine the relationship

between the changes in the resistance of the strain gage and the measured strain. This is the basic concept of the strain gage.

These changes in resistance are very small. A simple resistor circuit, known as the *Wheatstone bridge*, is utilized to measure small changes in voltage which is then converted into a displayed microstrain value.

Deflection Meter

The deflection meter/gage consists of a dial readout with a spring-loaded protrusion. This protruding piece is placed normal to a flat surface at the point where deflection is to be measured. The gauge is rigidly mounted so that deflection is measured relative to a known surface. The spring-loaded piece is *preloaded* (i.e. placed tightly against the surface so that it already registers a deflection) so that there is no dead space between it and the surface of interest. As the structure deflects, the protruding piece moves, causing the dial gauge to move. The change in this reading gives the deflection of the structure at that point.

Experimental Procedure

The lab takes place in the Hangar at the back of building 33.

Before you begin the lab, you must perform an analysis of the truss so that you have an expectation for which truss members should be in tension or compression, and for the relative loads each member will carry. This is problem set question M7. Step 1 should be completed before arriving in the lab.

1. Analyze the truss shown in Figure 3 to determine the loads in bars #5, #6 and #7 as a function of the load P, applied to the structure. This will then give the slope of a line, to later be plotted, relating the individual members to the applied load.

The steel truss will be loaded from 0 to 1000 pounds (i.e. 500 lbs per load cell) in ten even increments of 100 pounds. The aluminum truss will be loaded from 0 to 500 pounds (250 lbs per load cell).   At each load level, six readings will be taken: readings of the strain in the two load cells and the three instrumented members, and a reading of the deflection at the center of the truss.

Measurements of microstrain for each strain gage are obtained using a digital readout box. By setting a dial on the box, you can select which of the strain gages (or load cells) is being displayed on the box. Each strain gage is labeled so you can tell which gage corresponds to which channel on the digital readout box. The instrumented truss members correspond to channels 5, 6 and 7, as shown in Figure 2. The load cell gauges correspond to channels 8 and 9. When reading microstrain values, be sure to wait until the readout has settled and is showing a relatively constant value.

The deflection meter should be set up under the central joint as shown in Figure 3, and should be preloaded as described previously.

2. Be sure to wear safety glasses in this experiment, since a failure of the loading mechanism or the truss could be dangerous.

3. The first step is to record the *zero* readings of all instruments at zero load. Because the strain gauges follow a very linear calibration curve, we are interested in *changes* in the readings of the gages, not necessarily the absolute values. Thus, readings can be taken at zero load and these readings subtracted from readings at the other loads to determine the actual reading. Record the microstrain readings from each gage and the deflection meter in Table 1 on the Lab Worksheet.

4. Prior to beginning any loading, the calibration of the two load cells (#8 and #9) needs to be checked. Hang a known weight (~ 30 lb) off of each load cell, one at a time, using the hook apparatus provided in lab. Record the microstrain values for each gauge in Table 2. Check the microstrain reading you obtain against the expected value for that strain gauge based on the calibration data. Note that because we are only hanging 30 lb off of the load cell, this is not really sufficient to check the entire calibration of the gauge, but it is enough to ensure that the gain or scale setting for the gauge readout has not changed since the calibration was performed.

5. Once the initial zero readings have been taken and the calibration check is finished, begin loading by jacking the load apparatus until the two load cells give a microstrain reading corresponding to 50 pounds for each load cell (or a total sum of 100 pounds. Remember the assumption of even load distribution with regard to the analytical model. This assumption is also used in terms of load distribution between the two two-dimensional trusses in the experimental apparatus). Record the raw readings of the two load cells (#8

and #9), the three instrumented members (#5, #6, #7), and the deflection meter in Table 3.
The zero readings you recorded earlier will be used later to correct these values.
Crosscheck that the loads in bars 5-7 are behaving as expected according to your analysis
from step 1.

6 Repeat this procedure by increasing the load on each load cell in 50 lb increments up to a
total of 500 lb for the steel truss, or 250 lbs in 25 lb increments for the aluminum truss. Do
not exceed 500 lb (250 lbs) on each load cell (1000 lb or 500 lb total load) - monitor the
load cells while loading the truss. When complete, slowly remove the loading.

7.   When the load is again zero, **again** take readings (record in Table 4). This allows you to
check to see if the zero values you originally read/set are again achieved. If not, this gives
you some idea as to errors that may have built up within the system.

8. Next, calculate the load applied to each gauge using the data you entered in table 3,
correcting with the zero readings from Table 1. Use the calibration data to convert
microstrain into load. Record these load values in table 5.

This ends the in-lab part of the systems problem. You will then need to make a brief write-
up on the lab:

1. Include your analysis of the truss, and your final answers for the loads in members #5, #6
and #7 as a function of the applied load P on the two dimensional truss.

2. Plot the *ideal* load in truss members #5, #6, and #7 from your analysis of the ideal two-
dimensional truss vs. the applied load, P. Plot the load in each member on a separate
graph, with the applied load along the x-axis going up to the maximum load applied.
Label your axes, including units and use sensible intervals on the axes.   You may do this
using a computer program, or on graph paper. In either case be careful to distinguish
between the experimental data and the results of your model (from M7)

3. Next overlay plots of the experimentally-measured load for the three physical truss
members (#5, #6, #7) as a function of the applied load on each of the ideal plots.
Remember that:

$$P = \frac{Load\ Cell\ 1 + Load\ Cell\ 1}{2}$$

4. Plot the deflection of the truss vs. the applied load.

5. Comment on the analytical and experimental results, particularly paying attention to the ability of the analytical model to capture the behavior actually observed (for load only, not the total deflection of the structure). Try to explain any differences by considering possible sources of error in the measurements as well as inadequacies of the analytical model.

6. Include the attached data worksheets in your write up.

Truss analysis code

In this part of the lab, you will be creating a computerized truss solver to aid in analysis of the truss in the lab. The truss solver should be capable of handling statically determinate trusses, consisting of a pin joint and a roller joint. It should also be able to handle multiple loads applied to a joint, at any angle.

A large portion of the truss solver has already been written, and can be found on the systems problem page of the unified website. The body of the procedure Solve_Forces has not been written yet. You will be writing this procedure. A description of the existing procedures can be found in the next section.

Before writing this procedure, it is crucial to understand how a computer can solve the linear system of equations generated by the method of joints. Look at the truss shown below.
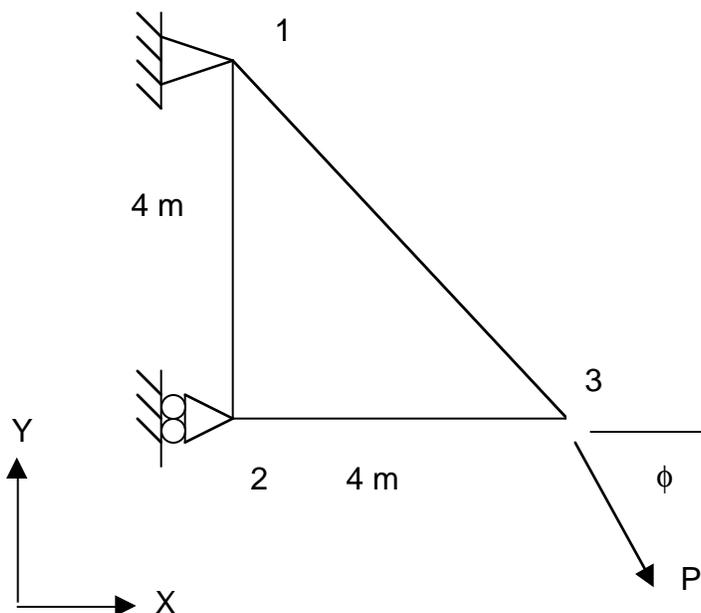
The way to solve this truss learned in class would be to first calculate the reaction forces, and the apply the method of joints to find the bar forces. However, if we leave the reaction

forces as variables, we can apply the method of joints and write the equations at each joint as follows, with the coordinate system defined as shown:

At Joint 1:

$$\cos\theta_{13} * F_{13} + \cos\theta_{12} * F_{12} + R_{1X} = 0$$

$$\sin\theta_{13} * F_{13} + \sin\theta_{12} * F_{12} + R_{1Y} = 0$$

At Joint 2:

$$\cos\theta_{23} * F_{23} + \cos\theta_{21} * F_{21} + R_{2X} = 0$$

$$\sin\theta_{23} * F_{23} + \sin\theta_{21} * F_{21} = 0$$

At Joint 3:

$$\cos\theta_{31} * F_{31} + \cos\theta_{32} * F_{32} + \cos\phi * P = 0$$

$$\sin\theta_{31} * F_{31} + \sin\theta_{32} * F_{32} + \sin\phi * P = 0$$

In these equations, θ is defined as the angle between the bar and the x-axis, and ϕ is the angle that the load makes with the x-axis. The positive direction for the angles is defined as going up from the x-axis (towards the y-axis). The subscripts on the angles are important – they are what determine the sign of the coefficient of the Force. As was shown in class, $F_{XY} = F_{YX}$.

It is now possible to put this linear system of equations into a matrix, which can then be solved easily by the computer. The matrix equation is of the following form:

$$[\text{Coefficient Matrix}]*[\text{Force Vector}] = [\text{Load Vector}]$$

For this example, the equation is:

$$
\begin{bmatrix}
\cos\theta_{13} & \cos\theta_{12} & 0 & 1 & 0 & 0 \\
\sin\theta_{13} & \sin\theta_{12} & 0 & 0 & 1 & 0 \\
0 & \cos\theta_{21} & \cos\theta_{23} & 0 & 0 & 1 \\
0 & \sin\theta_{21} & \sin\theta_{23} & 0 & 0 & 0 \\
\cos\theta_{31} & 0 & \cos\theta_{32} & 0 & 0 & 0 \\
\sin\theta_{31} & 0 & \cos\theta_{32} & 0 & 0 & 0
\end{bmatrix}
*
\begin{bmatrix}
F_{13} \\
F_{12} \\
F_{23} \\
R_{1X} \\
R_{1Y} \\
R_{2X}
\end{bmatrix}
=
\begin{bmatrix}
0 \\
0 \\
0 \\
0 \\
\cos\phi * P \\
\sin\phi * P
\end{bmatrix}
$$

Inverting the coefficient matrix and multiplying it by the load vector will solve the system of equations for the forces.

In your homework, you used the Ada matrix packages for the first time. They will be useful here, as there are functions in those packages to take the inverse of matrices, regardless of their size. To translate the matrix above into Ada code, it is important to notice a few things. First, there are two equations for each joint – one for the x-direction and one for the y-direction. Any force that is represented in the x equation is represented in the y equation. The difference is that the coefficient is cosθ for the x-direction and sinθ for the y-direction. This fact will be useful in setting up the matrix. Second, the reaction forces and the load only appear in certain equations. Third, the angles need to be kept track of so that the signs are maintained.

Using the first fact noted above, we can set up the matrix two rows at a time. Any time we have a bar force, we can set the X-row to the cos of the angle from the joint that we're at to the joint that we're going to (from joint 1 to joint 3, for example) and then we can set the Y-row to the sin of the angle. We can then move to the next column. When we have looped through all of the forces at that joint, we can go on to the next joint by incrementing the X-row counter and the Y-row counter by 2 (this moves us down two rows and to the set of equations for the next joint).

As stated above, it is important to keep track of the angles to get the sign on the forces correct. The pre-written Ada code takes this into account. The force record is as follows:

```
type Force is    record
     Force   : Float;
     Joint1  : Integer;
```

```
    Joint2    : Integer;
    Angle_1_2 : Float;
    Angle_2_1 : Float;
  end record;
```

Angle_1_2 is the angle looking from Joint1 to Joint2, and Angle_2_1 is the angle looking from Joint2 to Joint1. Applying this to the previous example, the record for the force in bar 1-3 would look like:

Force := null; (this has not been set yet)

Joint1 := 1;

Joint2 := 3;

Angle_1_2 := θ_{13}

Angle_2_1 := θ_{31}

Therefore, when setting rows 1 and 2 of the matrix, Angle_1_2 should be used, and when setting rows 4 and 5 Angle_2_1 should be used. The force is what we are solving for, and will be set later in the procedure.

The reaction forces and the load must also be taken into account. If you look at the input procedure for the truss solver, you can see that the joints where the supports and loads are located are specified by the user. Therefore, when looping through the joints to set the forces, it is easy to set the reaction forces in the matrix, and the loads into the laod vectors at the same time. Be careful that you set the loads into the correct rows in the load vectors.

DESCRIPTION OF EXISTING PROCEDURES

Truss_Input:

This procedure takes the input from the user and sets the variables to be used in the rest of the program. A sample input file can be found on the website. It asks the user for the number of joints in in the truss. Then, for each joint, the user inputs the X and Y location – the origin should be set at the lower left corner of the truss. Then, the user inputs the number of bars that connect to that joint, and then the joint numbers of the connecting joints. These connecting joints are stored in an array in the record representing the joint. A 1 represents a connection, while a –1 represents no connection.

For example, look at the array representation shown below:

Index	1	2	3	4	5	6	7	8	9	10
Value	-1	1	1	-1	-1	-1	-1	-1	-1	-1

This is a representaion of a joint that is connected to joint 2 and joint 3 (joint 1 in the previous example, for instance).

The Truss_Input procedure then inputs the total number of bars and the location of the supports, as well as the direction of the roller reaction force, represented by the enumerated type direction (HORIZ or VERT).   Finally, the loads are input, consisting of the joint at which it is applied, the magnitude of the load, and the angle that the load is applied at.

The input is taken from a file. The sample truss input file on the web describes how to set up your input file. The input file must be named "Truss_Input.txt".

A note on File_IO:
It is often convenient to set up input in the form of a file. For situations where input may change often or many different sets of input need to be used, inputting via a file offers more flexibility to the user and is also easier. To do file input in Ada, a file_type variable must be declared. The file_type type is found in Ada.Text_IO. Then the file must be opened, and its mode declared. The possible modes indicate whether the file is used for reading in data (in_file) or for writing out data (out_file). In the same statement, the name of the file is specified. This is the name that the input file must have in order for the data to be read in. Then, get statements can be made from the file using the same get statement with the addition of a file parameter. A sample of the code can be found below, and the process is summarized on page 214 of the Feldman book.

```
        Truss_Input_File : Ada.Text_IO.File_Type;
        Ada.Text_IO.Open( File => Truss_Input_File, Mode => Ada.Text_IO.In_File,
                          Name => "Truss_Input.txt");
        Ada.Text_IO.Get(File => Truss_Input_File, Item => Num_Joints);
```
        This code declares a Truss_Input file as a file_type, then opens the file named
"Truss_Input.txt" as an in_file. The next line reads the first line of text, storing the number found there as Num_Joints.

Force_Set_Up:

This procedure uses the information input in Truss_Input to set up the angles and joints for each force. It sets the values in the array Forces, filling out all the values in each force record, outside of the magnitude of the bar force. It is set up so that bar forces are not repeated (ie F_{12} and F_{21} are not both set).

Truss_Output:

This procedure outputs the results of the program.

Solve_My_Truss:

This procedure is the test program. You do not need to make any changes to this program.

DELIVERABLES:

When you are done with this portion of the lab, you should turn in the following online (zipped into a single file named SP9_Lastname_Firstname.zip):

The new truss.adb file containing your code

The truss.ads and solve_my_truss.adb files

Your input file

Please include the matrix pacakges that you use in your zip file.

You should also turn in a hard copy of your code, and a listing of the results of your program applied to the truss analyzed in M7 and that you experimented with in the experimental part of this systems problem.

WORKSHEET
Truss Laboratory
The Truss

Zero Readings

Table 1

Raw readings at zero load

Load Cell 8	Load Cell 9	Bar 3	Bar 4	Bar 5	Deflection Meter

Be sure to include proper units!

Load Cell Calibration Readings

Table 2

Calibration readings

Known Load	Load Cell 8	Load Cell 9

Be sure to include proper units!

Raw Test Results

Table 3

Raw experimental measurements

Interval	Load Cell 8	Load Cell 9	Bar 5	Bar 6	Bar 7	Deflection Meter
Units -->						
1						
2						
3						
4						
5						
6						
7						
8						
9						
10						

Put proper units at head of column

Final Zero Check

Table 4

Raw readings at zero load

Load Cell 8	Load Cell 9	Bar 5	Bar 6	Bar 7	Deflection Meter

Be sure to include proper units!

Converted Test Results

Use proper calibration curves attached at the end of this Worksheet

Table 5

Load Measurements

Interval	Load Cell 8	Load Cell 9	Bar 5	Bar 6	Bar 7	Deflection Meter
Units -->						
1						
2						
3						
4						
5						
6						
7						
8						
9						
10						
Rezero						

Put proper units at head of column