Name: ____________________

Group: ____________________

# Supercomputing Project

In this project, you will use a supercomputer to perform a simple structural analysis.

## 1 Introduction

Civil Engineering has been vital to the advancement of civilization. Ancient civilizations built huge monuments, such as the pyramids in Egypt, that still stand today. In those days, engineers relied upon tradition and experience to create structures. They had few mathematical tools to help them figure out how to build.

It has only been in the past couple of centuries that engineers began to rely more on experimentation and modeling. They would build small scale models using novel construction techniques to see if they could produce a building that would stand. Eventually, physicists figured out some laws of nature which helped civil engineers to quantify their work.

Today, civil engineers use a combination of tradition, experience, training, and computer modeling to build safe structures. Before they can become licensed engineers, all civil engineers must complete a B.S. in engineering, work for 4 years under the supervision of a licensed engineer, and take two licensing examinations.

For more information on the history of civil engineering, see `http://www.asce.org/history/hp_main.html`.

## 2 Project Overview

We are going to get a taste of what it's like to be a civil engineer. We will use a supercomputer in order to analyze a structure and get a feel for how civil engineers design structures.

First we will draw a structure on graph paper. Then we will use the program GiD to transform this drawing into a format that the computer can understand. Next, we will transfer the file to the supercomputer, where the effect of loading the structure with a large force will be analyzed by a finite element program called FElt. Finally, we will transfer the output file back to our local machines and use GiD to visualize the stress analysis.

### 2.1 Make Sure You Have the Necessary Software

You will need to download and install two programs if they have not already been installed on your computer. First of all, you will need GiD to render your structure in a format that the supercomputer will understand. Also, you will need an SSH client in order to transfer the file to the supercomputer, invoke the computer program, and transfer the output file back to your computer. You can get these programs (as well as this document) from the following URL:

`http://fourier.cse.uiuc.edu/`

### 2.2 Draw a Structure on Graph Paper

In this project, we want to design a section of truss (that framework that supports the tracks) for a roller coaster. Draw a design that you think would hold part of a train of roller coaster cars. It should be at least ten meters tall and five meters wide. Use the graph paper at the end of this packet to make a scale drawing of the truss.

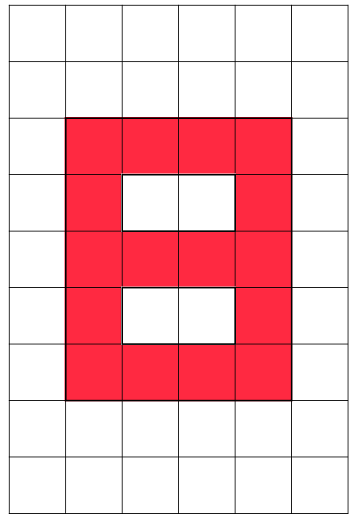

Figure 1: Example structure.

Some suggestions for your truss: Try to create a structure that uses as little material as possible while being structurally strong. Generally, triangle shapes are a lot stronger than rectangles. A rectangle with another support along the diagonal makes a much stronger truss than a rectangle without that extra support. (The example structure in this worksheet is a really lousy truss. You can create something much better!)

Work through the example of creating a GiD drawing first, and then draw your own structure in GiD.

## 2.3 Use GiD to Transform the Drawing

GiD is a commercial program that engineers use to design structures for computer analysis. The basic idea is to draw a picture of the structure, and then divide the structure into smaller parts called finite elements. The supercomputer will then take these pieces and do the analysis on each of them before lumping it all back together. To draw the picture, start the GiD application by double clicking on its icon. When it asks for a password, click cancel.

An important concept to keep in mind is positive versus negative space. Positive space is the area that is part of your structure. Negative space is the holes in your structure. So if we were going to draw the example structure, the red part is the positive space, and the other part is negative space.

Let's suppose that one grid length in Figure 1 equals two meters in real life. In order to make this structure using GiD, we would make the outer rectangle using the GiD drawing tool in the menu *Geometry – Create – Object – Rectangle*. Create the rectangle by entering the coordinates of the lower left side. Let's say that is the origin, so in the Command line at the bottom of the GiD window we type in `0,0` (no spaces) and press enter. Now we enter the second point, the opposite corner, by typing in `8,10` and pressing enter. This will create a blue rectangle with a magenta rectangle in the middle of it. Just ignore the magenta rectangle for now. Press the little button with the magnifying glass and house to center the drawing in the GiD panel.

Next we want to add the negative spaces. We can add them by adding lines, either by clicking in the appropriate places or by typing in coordinates. Alternatively, we can add them by creating more rectangles. Let's do one of each.

The top negative space has two opposing corners at (2,6) and (6,8). We can add this rectangle in the same way as the other one: by selecting the Rectangle drawing tool and typing in the two coordinates. So

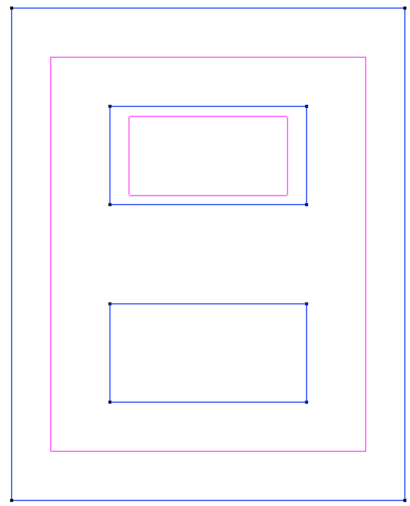

Figure 2: Three shapes, with two magenta rectangles.

select *Geometry – Create – Object – Rectangle*, and type `2,6` (enter) and `6,8` (enter) and that rectangle is created.

The bottom negative space has four corners at (2,2), (2,4), (6,4), and (6,2). We can create this shape using lines. So select the line tool, either the one along the side that has a picture of a straight line diagonally between two points, or by *Geometry – Create – Line*. Type in `2,2` (enter), then `2,4` (enter), `2,6` (enter), `6,2` (enter), and `2,2` (enter). When you type in the last coordinates, a window will pop up asking if you want the lines to be joined at the same point. Select JOIN. Notice that we can start the next line with the last point we entered. If we didn't want to do that, we would just press escape to get out of the tool and then reselect the tool. We had to type in five points because we wanted to go back to the point where we started.

Notice now that we have three shapes. Two of them have magenta rectangles, but one doesn't. The magenta rectangle represents the fact that that shape is considered positive space right now. We don't want that upper inside rectangle to be positive space. So we need to do something about it. We need to delete the magenta shape.

The way we delete the magenta rectangle is to select *Geometry – Delete – Surface*. Then we click on the inner magenta surface and it turns red. We press escape to finish, and the magenta rectangle should be gone.

Now we want to add the holes of the negative spaces to the outer rectangle. We do this using the *Geometry – Edit – Hole NURBS Surface* tool. First we enter which surface we want to make a hole in. Click on the magenta rectangle. Then we need to enter lines that define the hole. Make a box around the four lines of the upper hole. Press escape because those lines completely define the hole. Then we add the next hole in the same manner. After this is done, the drawing should now look something like Figure 3.

Next, we just need to mesh the structure. In order to do that, we select *Meshing – Generate...* . GiD will prompt us for the size of elements. Just press OK. Next it informs us of the size of the mesh and prompts us to press OK to see the mesh. Press OK. The mesh should look something like Figure 4.

Next we need to save our project in the proper format and export the mesh to the finite element program. First just save the project as, for example, newproj.

Then we need to export the mesh in a format that can be read by the finite element program. Choose *Files – Export – GiD mesh*, and save this file in a different location, perhaps on the desktop. Remember that location.

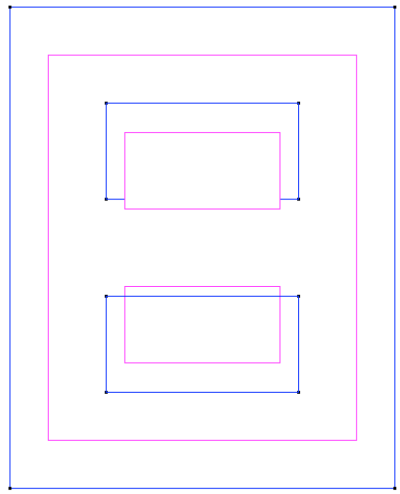

Figure 3: Completed structure.

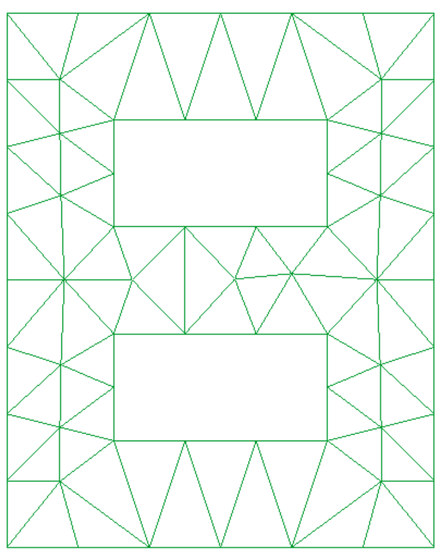

Figure 4: Meshed structure.

## 2.4 Transfer the File to the Supercomputer and Compute

We will use a special program called *SSH* to transfer the file to the supercomputer. SSH can transfer files between different computer systems; also it can be used to type commands into the supercomputer. We will use it for both purposes today.

First, start the SSH application. Click on "Quick Connect", and enter the username and system. Leave the rest of the settings alone. The remote system is called `tgc.trecc.org` and your user name is `hh##`, where the last two characters are numbers between 01 and 40.

This account has been assigned to your group for work on this project only, and only during school hours. You are privileged to have the opportunity to work on a supercomputer. The vast majority of high school students have never had this opportunity, and never will. Treat the supercomputer with respect. The administrators of this machine have your names and will be able to tell if you try to do anything nasty. Misuse of the account could be subject to criminal prosecution. *In summary, use this supercomputer for this class project only.*

You might see a window saying that this is the first time you have connected to this host, and asking whether you want to connect. Choose yes. Next you will be prompted for a password. Enter the password that goes with your username. You should see some new text in the window of the ssh application, and a prompt that looks something like `[hh02@tgc hh02]$`.

We want to upload our input file to the supercomputer. Click on the yellow folder icon to get a file transfer window. In the file transfer window, your local computer is shown in the left window, and the supercomputer in the right window. Find your input file by navigating directories on your local computer. Transfer that file to the supercomputer by clicking on your input file, then clicking the upload button, which is the up arrow icon right below "Help" in the top menu. Alternatively, you can use the menu and select *Operation – Upload*.

After the file is uploaded, go to the ssh window and type in the following command: `ilaunch` *myinputfile.msh* (where *myinputfile.msh* is the name of your input file) and press enter. The computer will then ask for the magnitude of the force you wish to impose on the structure. Enter a positive number greater than or equal to 1000. Then it will prompt you for what type of material you want to have your structure made from. Select a number corresponding to the material you want.

The supercomputer will immediately launch your job. Wait for a second or two and then we will download the output file back to your local computer.

Press F5 to refresh the listing in the file transfer windows. Click on the file *myinputfile.post.res* in the right window. Download that file to your computer by clicing the down arrow icon which is located next to the up arrow you used before. After that file has been transferred, disconnect from the supercomputer by pressing the button with the computer with red slash.

Now, copy and paste the file you downloaded into the directory in which you saved your GiD session at the beginning. If you need help with this, just ask. The downloaded file has to be in the proper location in order to proceed to the next step.

## 2.5 Use GiD to Visualize Results

Return to the GiD window and select *Files – Postprocess*. This will load the results into GiD. Now you will be able to see graphical representations of the effects of a heavy load on your structure.

Select from the menu *View Results – Contour Fill – Node Displacements – X-Node Displacements*. A color graph will show up on your structure, with red representing the most displacement and blue the least displacement. The red parts moved the most to the right, and the blue parts moved the most to the left. Similarly, look at the graph from *View Results – Contour Fill – Node Dosplacements – Y-Node Displacements*. The blue parts moved the most downward, and the red parts moved the least in the downward direction. Think about trying to squish your structure from the top, using a gigantic hand. Do these results make sense to you?

*Strain* is the tension force on the structure. Look at the graph from *View Results – Contour Fill – Element Stresses – |Element Stresses|*. Where does the most strain occur on your structure? This is the place where it is most likely to fail first. Circle these areas of your structure on your drawing.