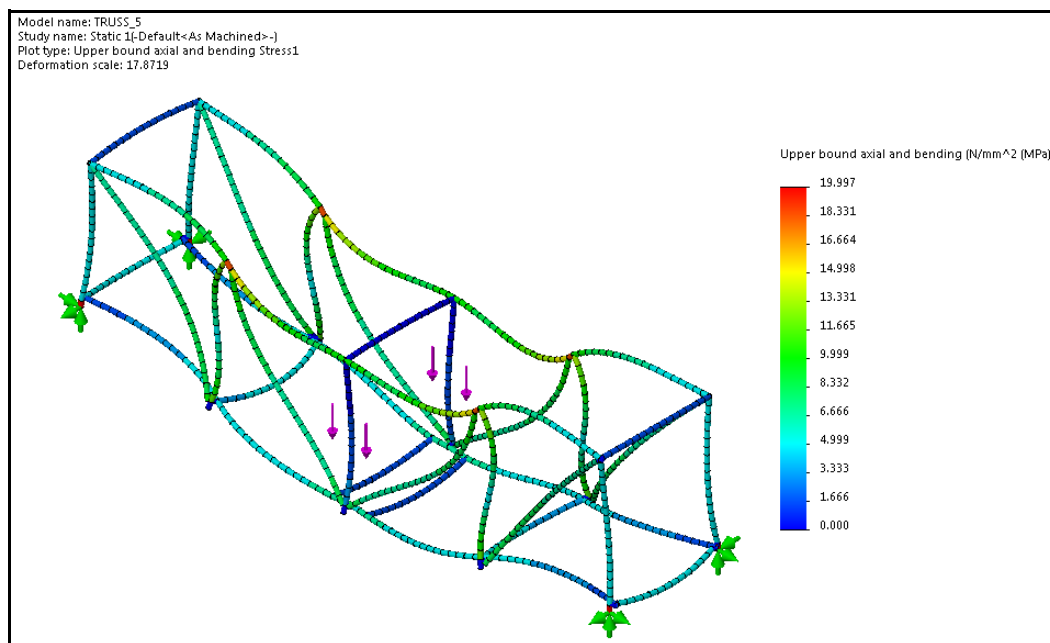


Bridge Design Project



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Introduction

Goals of This Lesson

- ❑ Describe the relationship between Parts, Assemblies and Drawings.
- ❑ Identify the principal components of the SOLIDWORKS user interface.
- ❑ Download and extract the required companion files.

Using This Book

The *Bridge Design Project* helps you learn the principles of structural analysis using SOLIDWORKS and SOLIDWORKS Simulation as an integral part of a creative and iterative design process.

For this project, you will “learn by doing” as you complete a structural analysis.

What is SOLIDWORKS Software?

SOLIDWORKS is design automation software. In SOLIDWORKS, you sketch ideas and experiment with different designs to create 3D models using the easy to learn Windows® graphical user interface.

SOLIDWORKS is used by students, designers, engineers and other professionals to produce single and complex parts, assemblies and drawings.

Prerequisites

Before you begin the *Bridge Design Project* you should complete the following tutorials that are integrated in the SOLIDWORKS software:

- ❑ Lesson 1 - Parts
- ❑ Lesson 2 - Assemblies
- ❑ Lesson 3 - Drawings

You can access the tutorials by clicking **Help, SOLIDWORKS Tutorials, Getting Started**. The tutorial resizes the SOLIDWORKS window and runs beside it.

As an alternative, you can complete the following lessons from *CAD Student Guide*:

- ❑ Lesson 1: Using the Interface
- ❑ Lesson 2: Basic Functionality
- ❑ Lesson 3: The 40-Minute Running Start
- ❑ Lesson 4: Assembly Basics
- ❑ Lesson 6: Drawing Basics

Conventions Used in This Book

This manual uses the following typographical conventions:

Convention	Meaning
Bold Arial	SOLIDWORKS commands and options appear in this style. For example, Insert, Boss means choose the Boss option from the Insert menu.
Courier New	Feature names and file names appear in this style. For example, Sketch1.
17 Do this step.	The steps in the lessons are numbered in Bold Arial.

Before You Begin

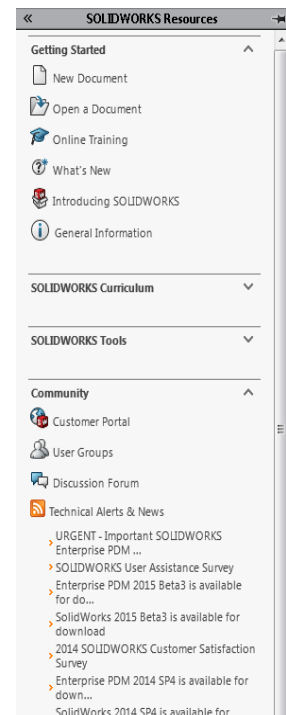
If you have not done so already, copy the companion files for the lessons onto your computer before you begin this project.

1 Start SOLIDWORKS.

Using the **Start** menu, start the **SOLIDWORKS** application.

2 SOLIDWORKS resources.

Click the **SOLIDWORKS Resources** tab  and click **Student Curriculum**.



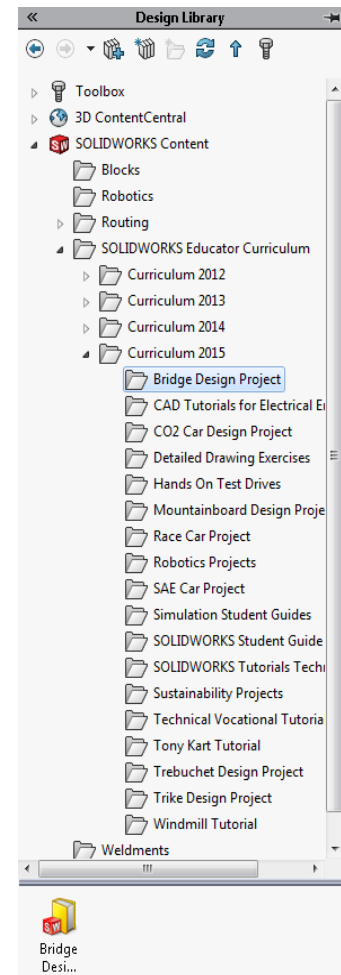
3 SOLIDWORKS Content.

Expand the *SOLIDWORKS Educator Curriculum* folder.

Expand the proper *Curriculum <year>* folder.

Click the *Bridge Design Project* folder.

The lower pane will display an icon representing a Zip file that contains the companion files for this project.



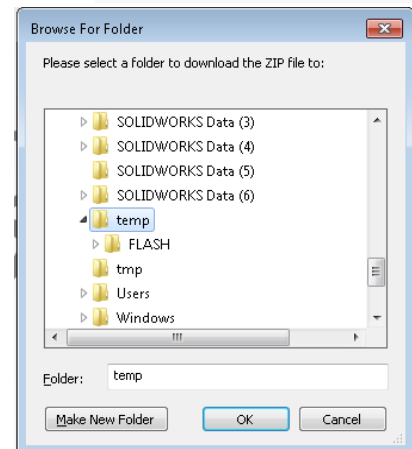
4 Download the Zip file.

Press **Ctrl** and click the *Bridge Design Project - English* icon.

You will be prompted for a folder in which to save the Zip file.

Ask your teacher where you should save the Zip file. Usually the *C:\Temp* folder is a good location.

Click **OK**.

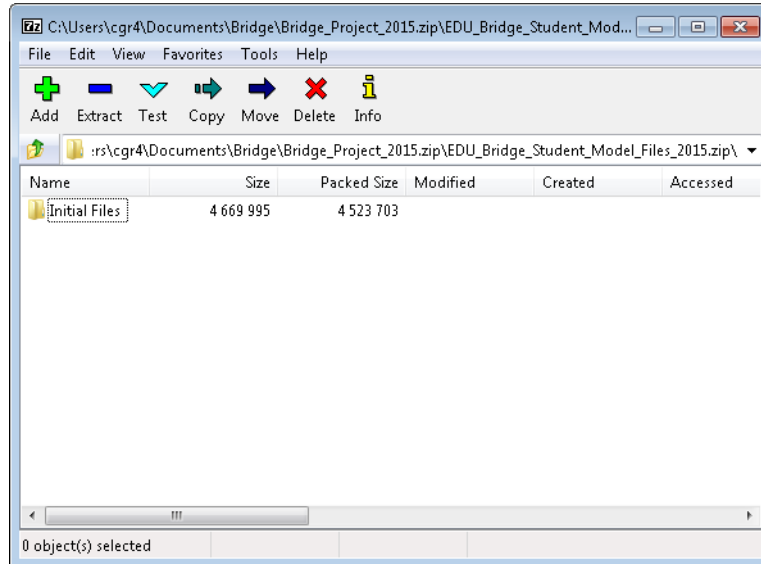


Tip: Remember where you saved it.

5 Open the Zip file.

Browse to the folder where you saved the Zip file in step 4.

Double-click the *Bridge Design Project.zip* file.



6 Click Extract.



Click **Extract** and Browse to the location where you want to save the files. The system will automatically create a folder named *Bridge_Design_Project_ENG* in the location you specify. For example, you might want to save it in *My Documents*. Check with your teacher about where to save the files.

You now have a folder named *Bridge Design Project* on your disk. The data in this folder will be used in the exercises.

Tip: Remember where you saved it.

Analyzing a Structure Using SOLIDWORKS and SOLIDWORKS Simulation

During this session, you will learn how to analyze a structure using SOLIDWORKS and SOLIDWORKS Simulation. You may also build the structure using balsa wood (see Building the Structure on page 81).

Once you have had a chance to see how easy it is to use SOLIDWORKS solid modeling software, you will use an assembly to check whether components fit properly.

You will then make a drawing of one of the components, complete with a cut list. If a printer is available, you can print out a hardcopy of your drawing.

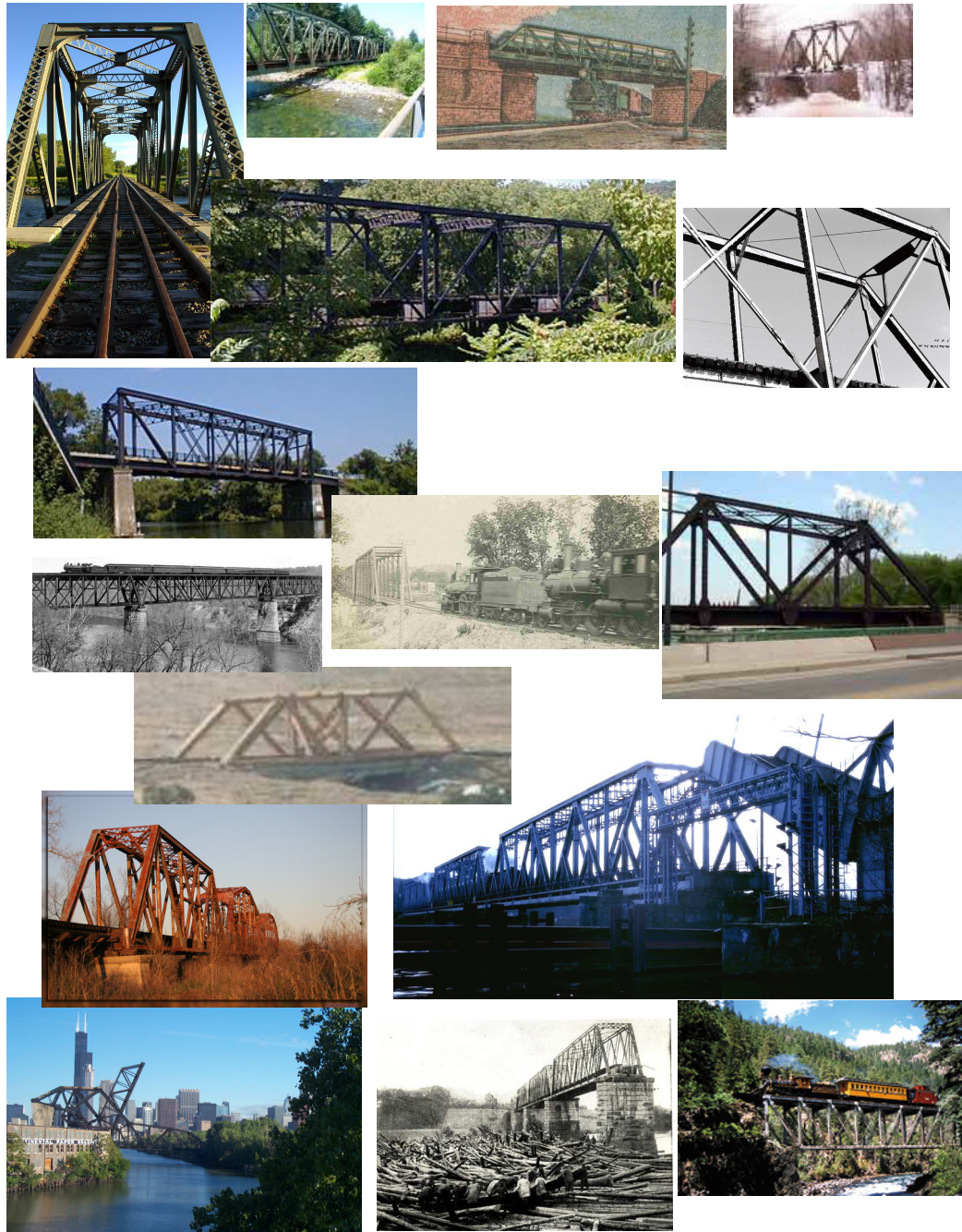
Lesson 1: Structure Design

Goals of This Lesson

- ☐ Define a structure.
- ☐ Describe several types of trusses.
- ☐ Understand what beams are.
- ☐ Understand what factors provide strength in a beam.
- ☐ Calculate a moment of inertia.
- ☐ Understand the importance of triangular bracing in a structure.

What is a Structure?

Structures are frames commonly used bridges for railroads, automobile and foot traffic. Examples of these structures can be seen across the country and the world.



Structure Designs

Structure designs are meant to be simple structures that are efficient, meaning that they are easy to build and accomplish their goals with the minimum amount of materials. There are many different structure designs, the differences are based on the load that the structure is required to support and the span that it must cross. The structure design may be repeated over several spans in the same bridge.

Trusses

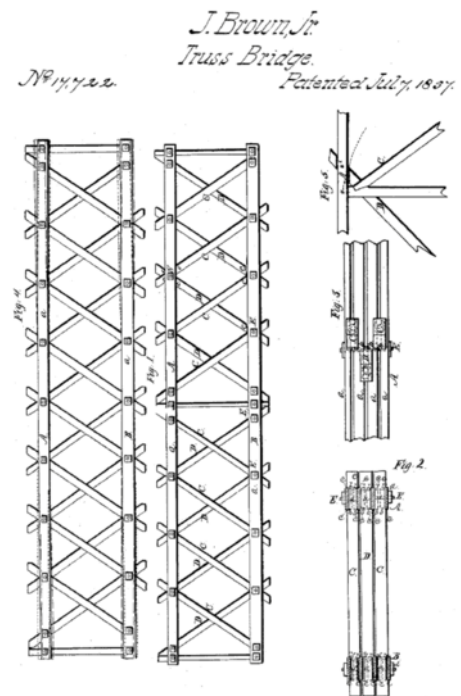
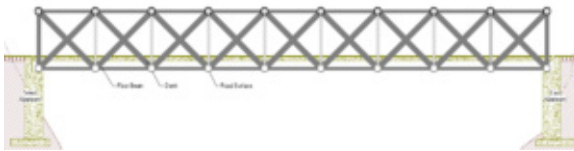
Trusses are specific types of structures commonly used on railroad bridges. They usually consist of a road or rail surface (deck), two walls and sometimes bracing on the top. You will be analyzing a truss design.



Search on **truss** for more information.

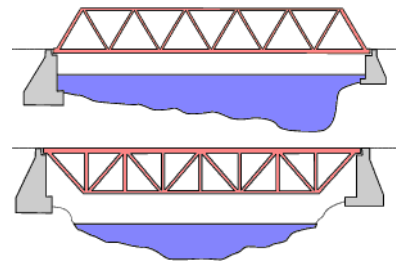
Brown Truss

The **Brown Truss** (patent shown here) was used in the design of covered bridges. This truss is a “box” truss (named for its boxy shape) that was so efficient that it could be constructed using only the (diagonal) cross bracing beams to support it.



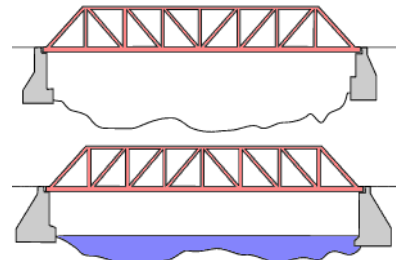
Warren Truss

The **Warren Truss** is another simple and economical design. It can be reversed and used with or without the vertical bracing depending on the load it needs to carry.



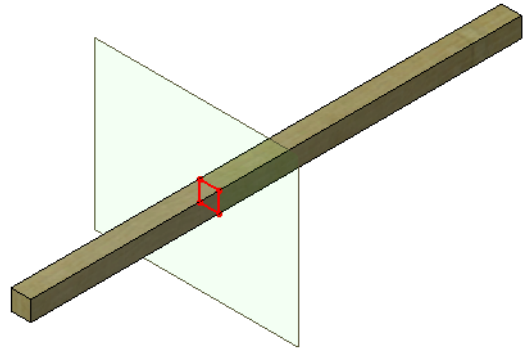
Pratt and Howe Trusses

The Pratt Truss and Howe Truss are very similar. Like the reversed Warren Truss shown above, they both have vertical and cross bracing. The difference is the direction of the cross bracing.



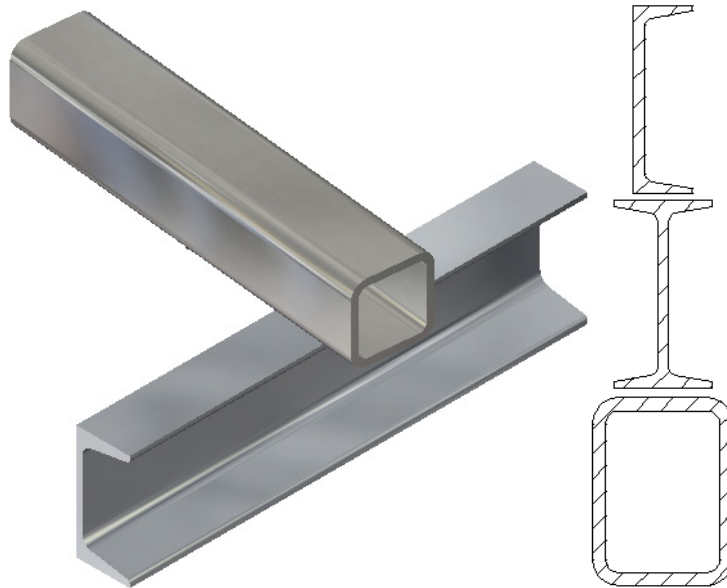
Beams

A **Beam** is an object that has the same cross section along its whole length. In this case, the cross section is square. Structures like trusses are composed of beams.



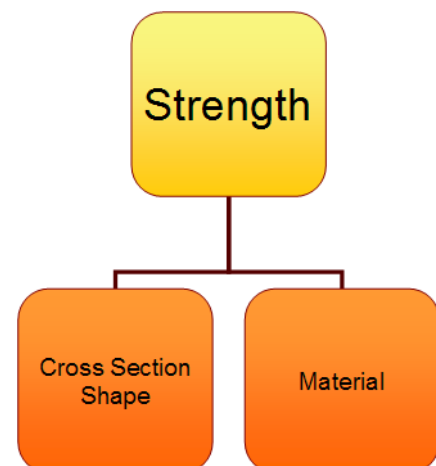
Steel Beams

Steel beams use standard shapes like channels, I-beams and tubes.



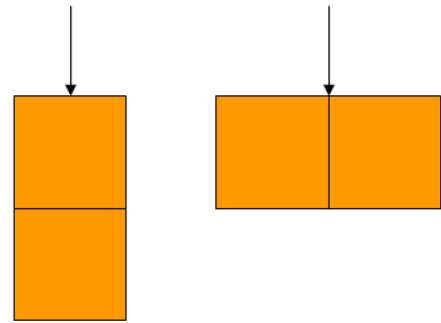
Strength

The strength of a beam depends on two factors, the **Cross Section Shape** and the **Material**.



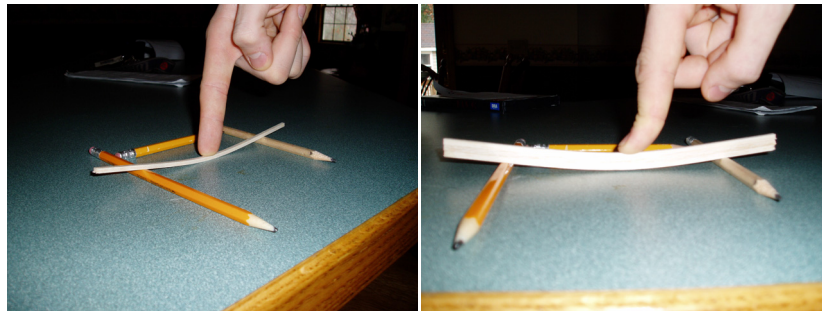
Cross Section Shape

Stacking two square beams creates a “deeper” section. The deeper the section (left) the stronger the beam. Wider sections (right) help a little but not that much.



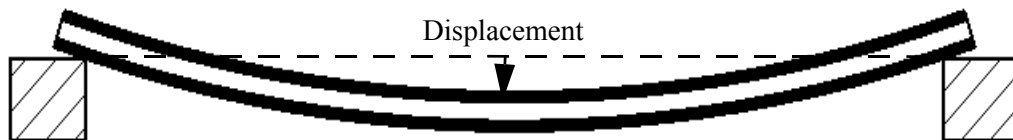
Try it!

Notice the difference in resistance between 1 balsa wood beam and 3 stacked beams when you try to press down. Use pencils for support and distance.



Displacement

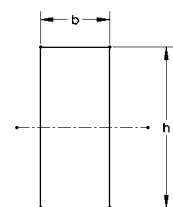
One of the results that we will be searching for in the structural analysis is the largest **Displacement**. It is the distance that the beam moved from the start when it an *external force* was applied to it. The displacement will help us determine the capacity of the structure.



Area Moment of Inertia

The reason that deeper beams are stronger is because of the **Area Moment of Inertia**. This is a formula calculated using the width (b) and height (h) dimensions of the cross section. It is a measure of the strength of the beam section alone, not the material.

The Area Moment of Inertia is used in calculations as resistance of a beam to bending. The higher the value, the more resistance against bending.







Calculating the Area Moment of Inertia

Using the formula below, you can calculate this value for several arrangements of rectangular cross sections.

$$AreaMomentofInertia = \frac{b \times h^3}{12}$$

Try some calculations

Try some calculations using the formula above and the values shown in the table below. The values are based on the cross section of a balsa wood beam, **3.175mm** (1/8") square.

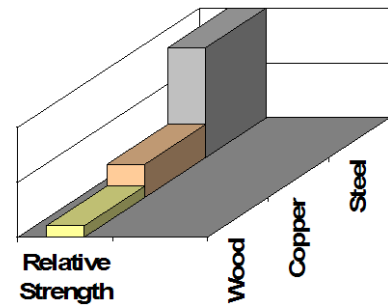
Number of square sections	Arrangement of square sections	b	h	Area Moment of Inertia
1		3.175mm	3.175mm	_____
2 Stacked		3.175mm	2 X 3.175mm	_____
2 Side by Side		2 X 3.175mm	3.175mm	_____
3 Stacked		3.175mm	3 X 3.175mm	_____

Questions

- 1 Which arrangement has the largest value? _____
- 2 Is the 2 side by side as strong as the 2 stacked arrangement? _____
- 3 Which arrangement is the weakest? _____?

Material

The material of the beam is another critical factor in determining the strength of the beam. Take three materials as an example: Wood, Copper, and Steel. The relative strength of each is shown in a chart at the right. In general, steel is stronger than copper, which in turn is stronger than wood. Keep in mind that there are a wide range of values within every material type and there are several types of *Material Properties* such as *Young's modulus* and *Poisson's ratio* that are used to define a material.



Note: Metals are manufactured products and due to the way they are created, they have equal strength in each direction. Materials like this are called *isotropic* materials.



Search on **material properties** for more information.

Wood as a Material

Wood is an especially difficult material to predict because it has a grain within it. The grain causes the strength to be different in each direction and it is not really an isotropic material. The porosity of Balsa wood makes it very susceptible to moisture which can cause large variations in the property values.

The values that we are using are estimates. If you choose to build and test a structure your results will be relative but the values may vary.

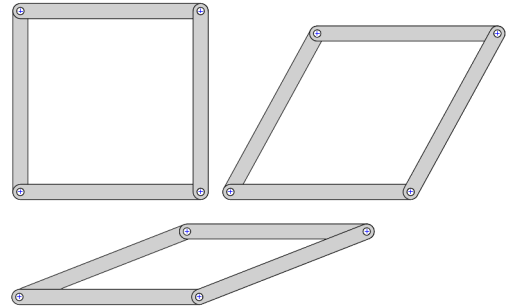
Truss Walls

The side walls of a truss are much more than just a fence to prevent objects from falling off. The walls usually contain bracing in the vertical and diagonal directions. When a truss contains both vertical and diagonal bracing, it is generally more stable.

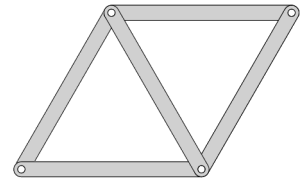
Triangles

Many structures, especially truss designs, contain triangles. Why are triangles so important? One reason is for stability. Stability is achieved by using cross braces to form triangles. Triangular shapes create stability in the truss.

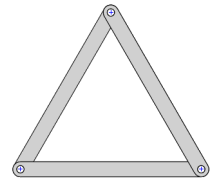
Consider a collection of members connected in a square shape by bolts or pins. Holding the bottom still, push on the top or side. It can form a square but can also be easily pushed into a flattened parallelogram.



Adding a 5th member diagonally makes a big difference. The shape is now locked in that position. The addition has broken the parallelogram into two triangles.

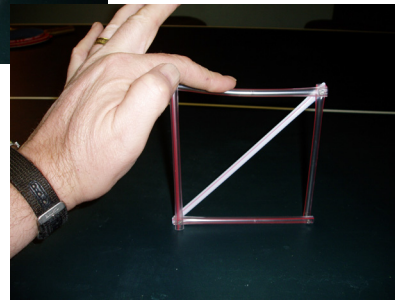
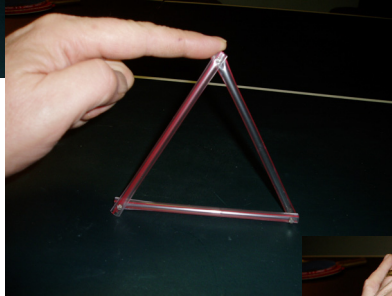
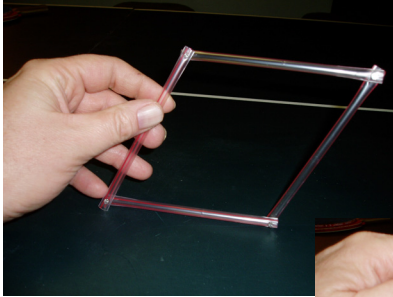


Using the same members and fasteners, create a triangle. This time fewer members are used but stability is achieved.



Try it!

You can simulate this process using something as flexible as a drinking straw. Use small pins to connect them together.



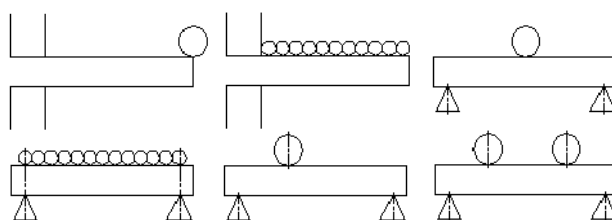
Lesson 2: Using the Beam Calculator

Goals of This Lesson

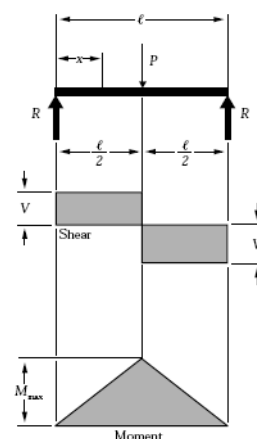
- ❑ Start SOLIDWORKS.
- ❑ Add-in the SOLIDWORKS Simulation software.
- ❑ Open an existing SOLIDWORKS part.
- ❑ Understand a simply supported beam.
- ❑ Assign a material.
- ❑ Calculate section properties.
- ❑ Use the measure tool.
- ❑ Use the beam calculator to calculate a displacement.

Using Beam Calculations

Before you perform any kind of an analysis, it is a good idea to have some idea on what results to expect. Although you will not know how much weight the structure can withstand, you can make an educated guess on one or more of the results that you will get. This is where beam calculations, simple formulas for beams, can be used. Below are some of the beam calculations available.



Note: Hand-calculated type beam calculations typically include formulas and look like this.



Order of Magnitude

Will the displacement (see Displacement on page 5) be close to 1mm? 10mm? The difference is **10** times greater than the previous one and are increasing by what is called an **Order of Magnitude**. An initial calculation can give you an idea of the order or magnitude of the results. This will help you determine whether the analysis has been done correctly.

Questions

- 1 What is the next value after 1mm and 10mm using an increasing order of magnitude? _____
- 2 What values are missing in this set? 5mm, _____, 500mm

Starting SOLIDWORKS and Opening a Part


- 1 Start the SOLIDWORKS application.

From the **Start** menu, click **Programs, SOLIDWORKS, SOLIDWORKS**.

Adding in SOLIDWORKS Simulation

SOLIDWORKS Simulation software is included with **SOLIDWORKS Education Edition**. To use it, it must be activated using **Tools, Add-Ins**. Click both **Active Add-ins** and **Start Up** for SOLIDWORKS Simulation, SOLIDWORKS Toolbox Library, and SOLIDWORKS Toolbox Utilities and click **✓**.

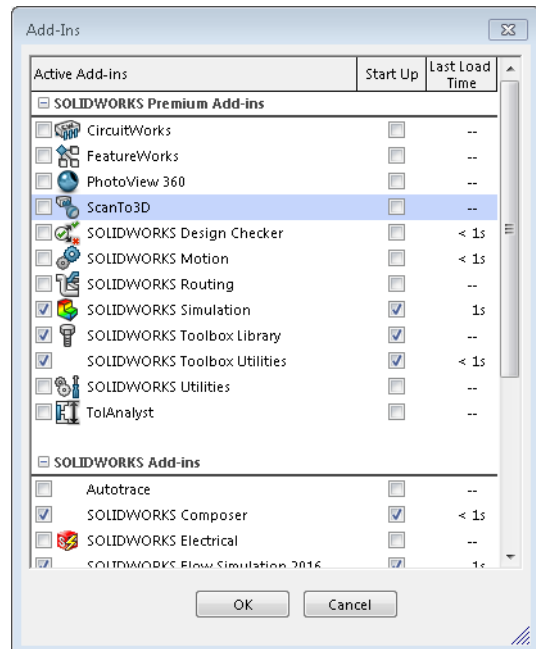
Where to Find It

- Menu Bar: **Options, Add-ins** 
- Menu: **Tools, Add-ins**

- 2 Add-in selections.

Click **Tools, Add-ins** and make sure that **Active Add-ins** and **Start Up** for **SOLIDWORKS Simulation**, **SOLIDWORKS Toolbox Library**, and **SOLIDWORKS Toolbox Utilities** are checked.

Click **✓**.




Warning!

If **SOLIDWORKS Simulation**, **SOLIDWORKS Toolbox Library**, and **SOLIDWORKS Toolbox Utilities** are not added in the project cannot be completed.

Opening a Part

Existing SOLIDWORKS files can be opened using the **Open** tool.

Where to Find It

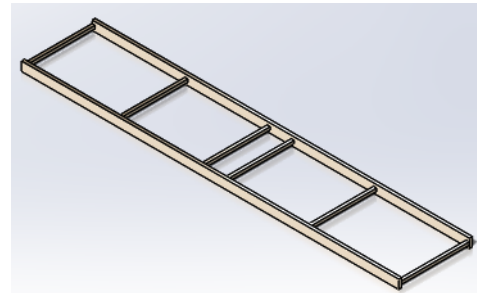
- Menu Bar: **Open** 
- Menu: **File, Open**
- Keyboard Shortcut: **Ctrl+O**

3 Open the part file.

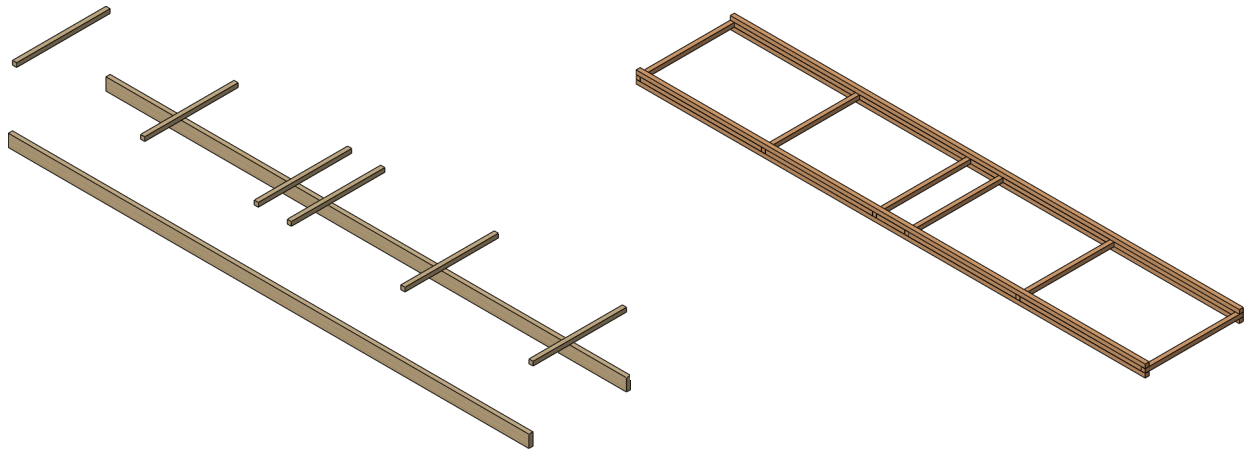
Click **Open** .

From the **Open** window, browse to the *Bridge Design Project\Student\Lesson 2* folder.

Select *TRUSS_1.sldprt* and click **Open**.

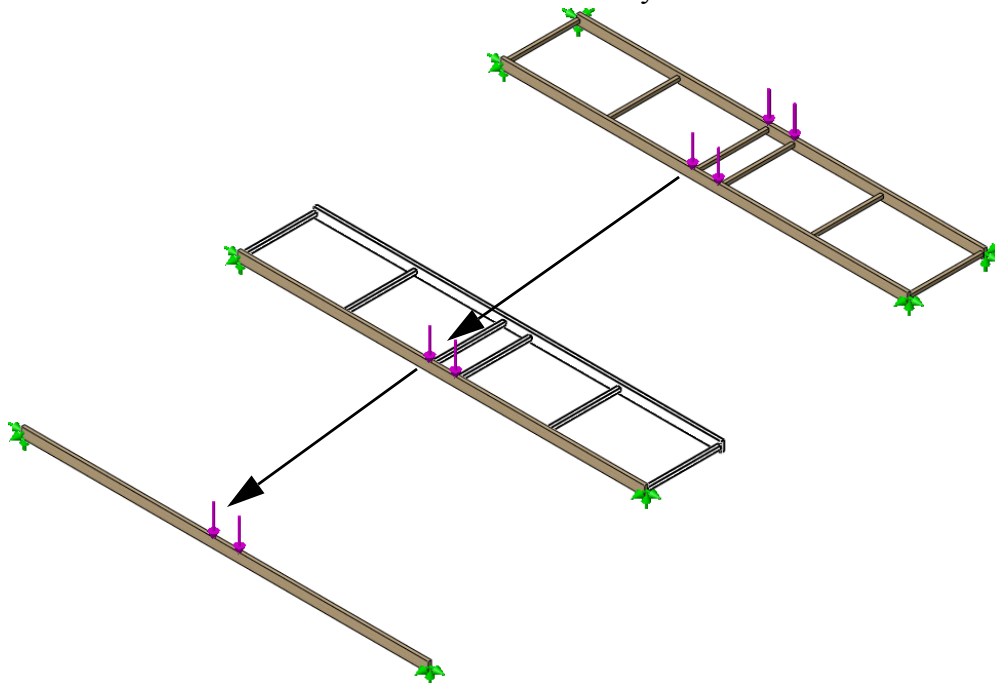
**The Model Geometry**

This model is made up of a series of *beams* that are placed against each other. The beams represent balsa wood sticks. In your project, the beams are combined by gluing them together. In a real structure, the beams would be welded or bolted together.



Simplifying the Analysis

The model appears as two parallel beams connected by smaller beams in several places. If we take just half of the model (just the large beam) and apply 1/2 the loads, we should get some idea of what the values will be in the real analysis.



The Simply Supported Beam

This type of beam calculation is often referred to as a “simply supported beam” where the contact points are not completely fixed and a load is applied. There are two key definitions you will need to know: fixtures and external loads.

Fixtures

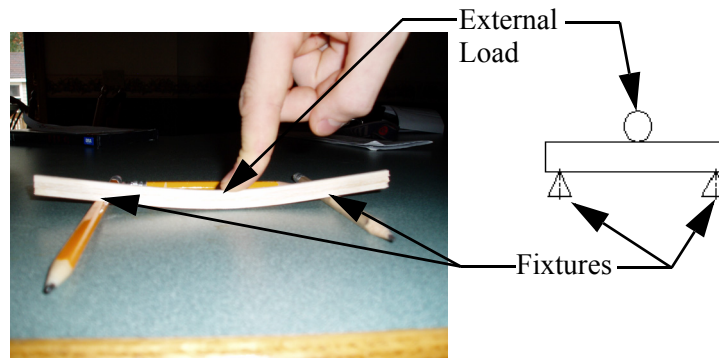
Fixtures are used to limit movement of certain points in the model. These are usually points of contact. They are also called constraints or restraints.

External Loads

External loads or forces can be used to add **Force** or **Gravity** loads to the structure. Adding a force requires a location on the structure, a value (in Newtons) and a direction.

Theoretical Model

This is the theoretical model (right) of the beam supported by the pencils in the previous lesson.

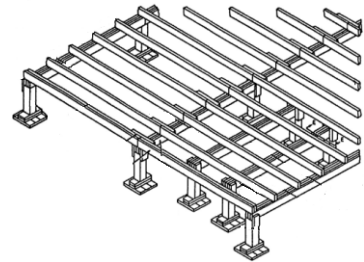


Why are Simply Supported Beams Important?

Although the theoretical model may seem overly simple, it serves as a practical approximation for many real structures. There are examples of the simply supported beam in use in many places.

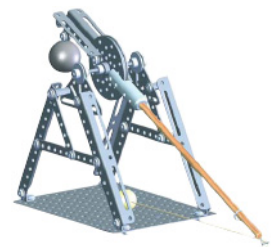
Structures

Wood and steel frame spans for homes and buildings are designed using simply supported beams.



Trebuchet

The trebuchet arm rotates on an axis between the frames. The axle is a simply supported beam.



Mountainboard

If you were standing in the middle of a mountainboard, you would be the external load and the wheels would be the fixtures. The structure can be approximated with a simply supported beam.

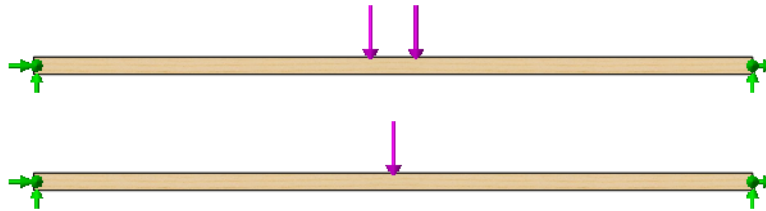


Note: This example is a “simplified analysis” that takes a 3 dimensional problem and simplify it to a 2 dimensional problem. A full simulation would still be needed.

Conservative Assumptions

Engineers often use “conservative assumptions” to make the analysis worse than reality for the structure. Doing this adds an extra level safety to the design and makes it stronger than it has to be. Here are some assumptions that will be made:

- 1 Using the ends of the structure as fixtures is worse than using the actual points of contact.
- 2 Using a single external load at the center is worse than two external loads near the center.



Required Data for the Beam Calculation

There are several pieces of data that are required to use this beam calculation. They include:

Data	Where to Find It?	What is it?
Modulus of Elasticity	Material Properties	Stiffness of material
Moment of Inertia	Section Properties	Resistance to bending
Length	Geometry	Length to cross span
Load	(given)	External load

Common Units

Common metric units are used in this project. Some of the common units used in the SI and IPS (inch, pounds, seconds) units systems are:

Data	SI Units	IPS Units
Modulus of Elasticity	Pa, MPa,	psi
Moment of Inertia	mm ⁴ , cm ⁴ , m ⁴	in ⁴
Length	mm, cm, m	in, ft
Load	N, kN	lb

Note: We will use the **SI** unit system in this analysis. The SI system of units is also known as the International System of Units. It uses metric units such as meters, millimeters and Newtons.



Search on **International System of Units** for more information.

Collect the Data

The required data will be collected using several different tools in upcoming steps. You will calculate the missing values in the chart below.

Note: As an initial guess, we will assume that the total weight load on the full structure is **40N**. We will use half of that, $40\text{N}/2 = \mathbf{20\text{N}}$, for the beam calculation.

Data	Value	Units
Modulus of Elasticity (pressure)	???	Pa (Pascals)
Moment of Inertia (length ⁴)	???	cm ⁴
Length	???	mm
Load (force)	20	N (Newtons)

Assign a Material

The first step is to assign a **Material** to the beams of the model. We want to make the structure out of balsa wood.

Where to Find It

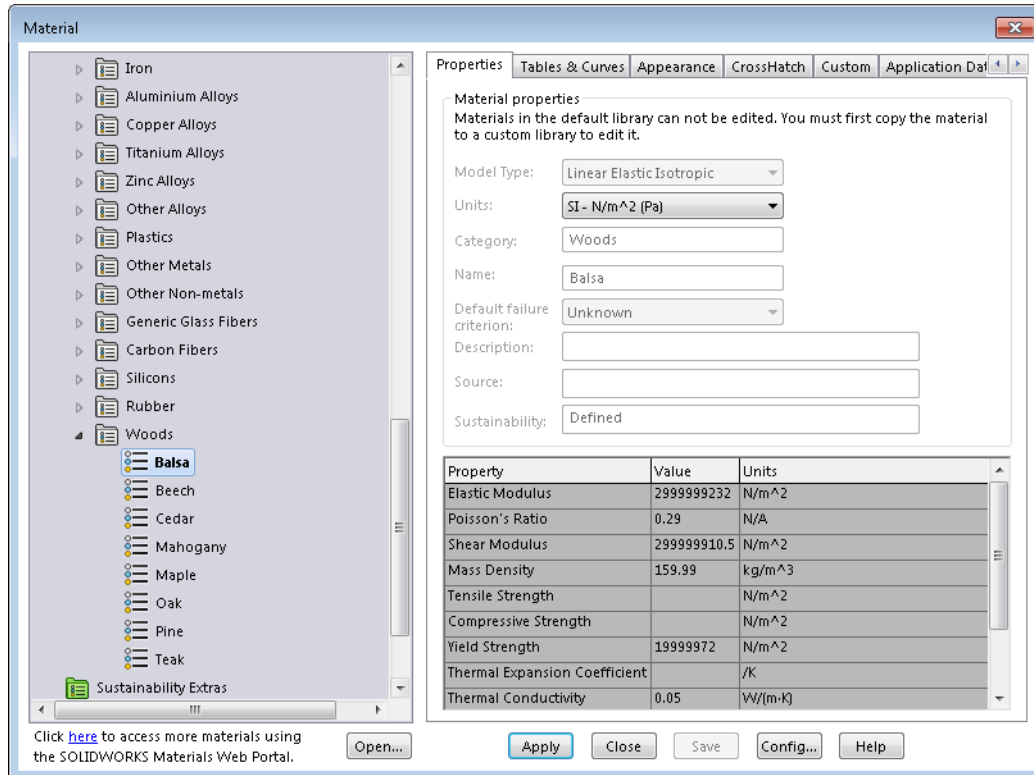
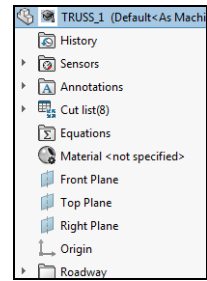
- Menu: **Edit, Appearance, Material**
- Shortcut Menu: Right-click the *Material* feature click **Edit Material**

4 Material.

Right-click the *Material* feature and select **Edit Material**. Expand the *SOLIDWORKS Materials* and *Woods* folders on the left and click *Balsa*.

Under **Units**, select **SI - N/m² (Pa)**.

Click **Apply** then click **Close**.



Note: The material used, *Balsa*, is chosen to make the analysis useful to those who will actually design, construct and test the structure. Balsa wood is a common material for student building projects.

The value of the **Elastic Modulus** or **Modulus of Elasticity** = 2999999232 N/m^2 .

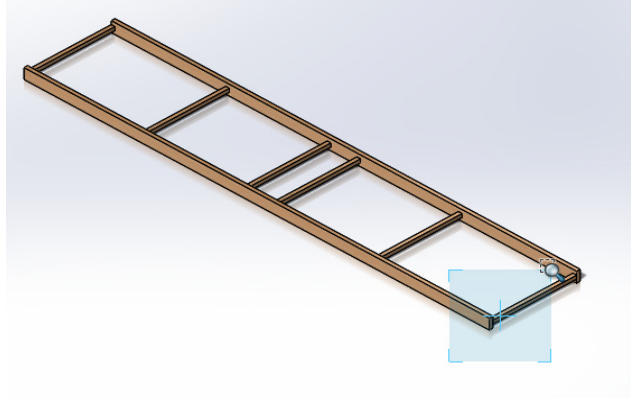
*You will learn more about materials, construction and testing in later lessons.

Section Properties

The section properties are based on the cross section of the beam.

5 Zoom to area.

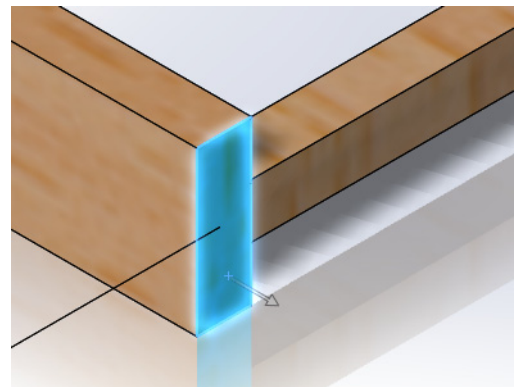
Click **View**, **Modify**, **Zoom To Area** and drag a window, upper left to lower right, around the corner of the structure as shown.



Note: Press the **Esc** key to turn off the zoom tool.

6 Face selection.

Select the face as shown.



Where to Find It

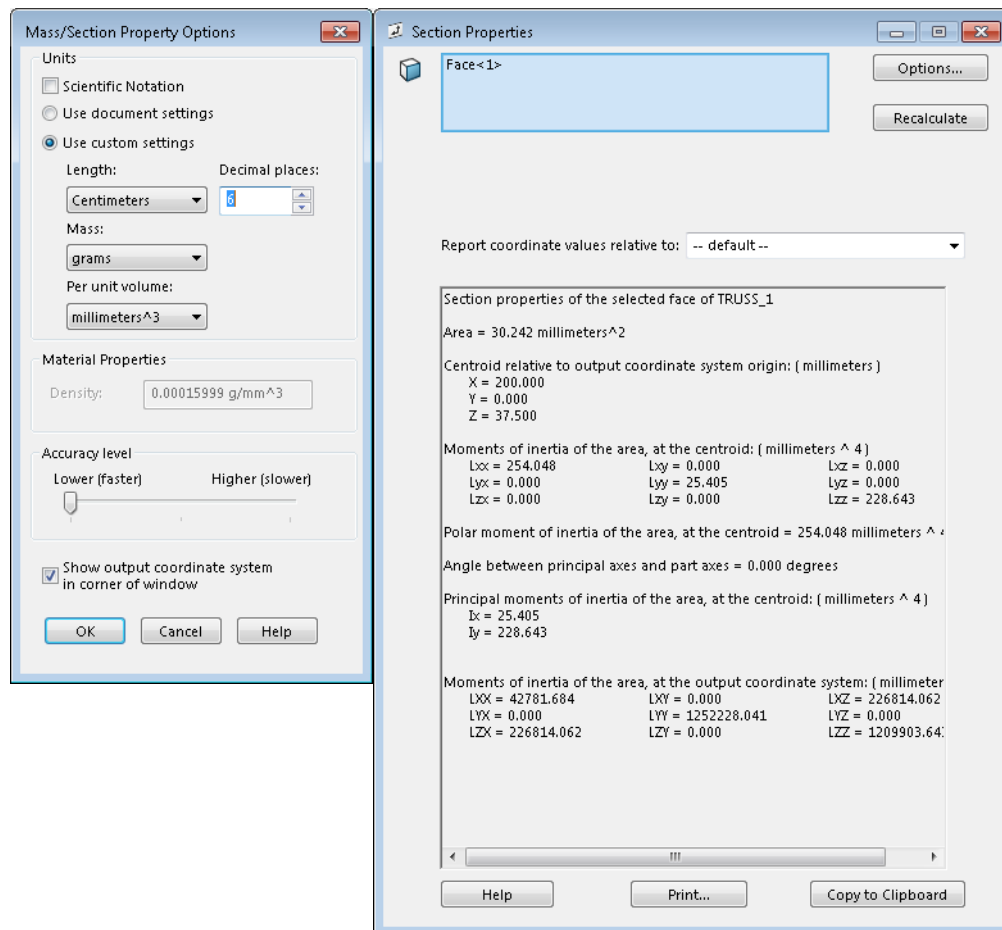
- ❑ CommandManager: **Evaluate > Section Properties**
- ❑ Menu: **Tools, Section Properties**

7 Section properties.

Click **Tools, Section Properties**. Click **Options** and **Use custom settings**. Select **Centimeters** and **6** decimal places as shown.

Click **OK** and **Recalculate**.

Moments of inertia of the area, at the centroid: (centimeters ^ 4) $L_{xx} = 0.025405$. Click **Close**.




8 Zoom.

Click **View, Modify, Zoom To Fit** or click the **f** key to return to the full view.

Using Measure

Measure can be used to measure distances or angles using model geometry.

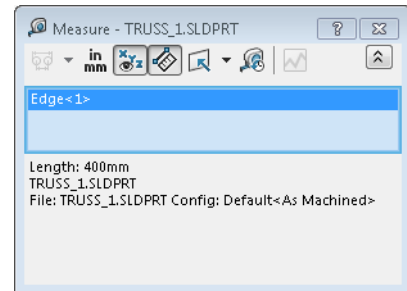
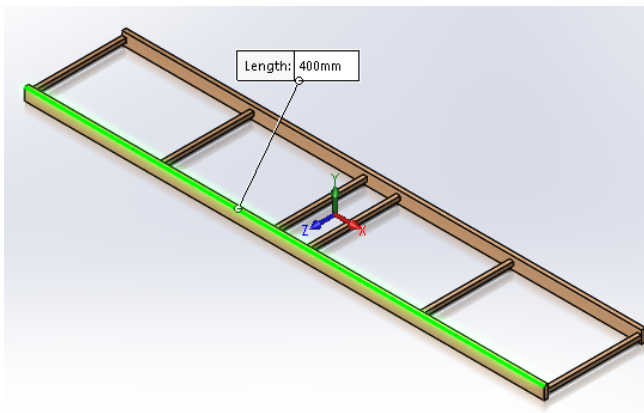
Where to Find It

- CommandManager: **Evaluate > Measure** 
- Menu: **Tools, Measure**

9 Measure.

Click **Tools, Measure**. Select an edge of the beam as shown. The length of the beam is displayed.

Length: 400mm.



10 Close.

Click the “x” in the upper right hand corner of the dialog to close it.

Beam Calculator

The beam calculator uses the input to determine the largest displacement or deflection of the beam.

Where to Find It

- Menu: **Tools, Toolbox, Beam Calculator** 

11 Start beam calculator.

Click **Beam Calculator**.

12 Settings.

Clear any values in the **Deflection** field (the **Solve** button will not be available until that field is cleared). Use the scroll bars to access **Supported at both ends, load in middle**. Click **Y local axis**, **Metric** and **Deflection**.

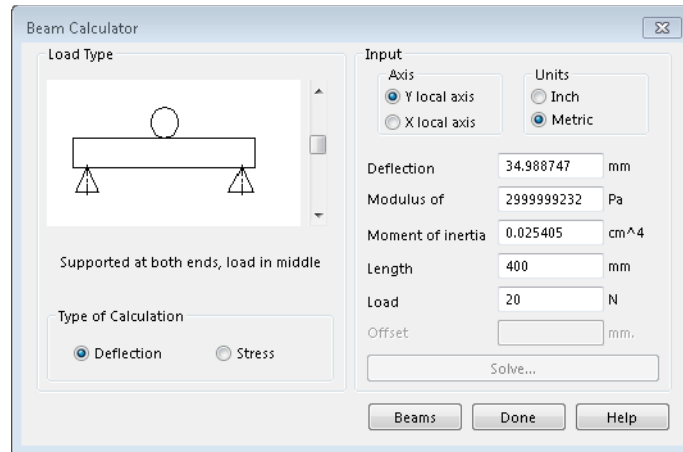
13 Type in values.

Type or copy and paste the values listed below into the dialog:

Data	Value	Units
Modulus of Elasticity (pressure)	2999999232	Pa (Pascals)
Moment of Inertia (length^4)	0.025405	cm^4
Length	400	mm
Load (force)	20	N (Newtons)

14 Solve.

Click **Solve**. The displacement is about **35mm** at the load. Click **Done**.



Questions

- 1 Is this displacement more or less than an inch? _____
- 2 Convert the displacement to inches: $35\text{mm}/25.4 =$ _____ in

15 Close the part.

Click **File, Close** to close the part.

At the *Save changes to TRUSS_1?* message, click **Don't Save**.

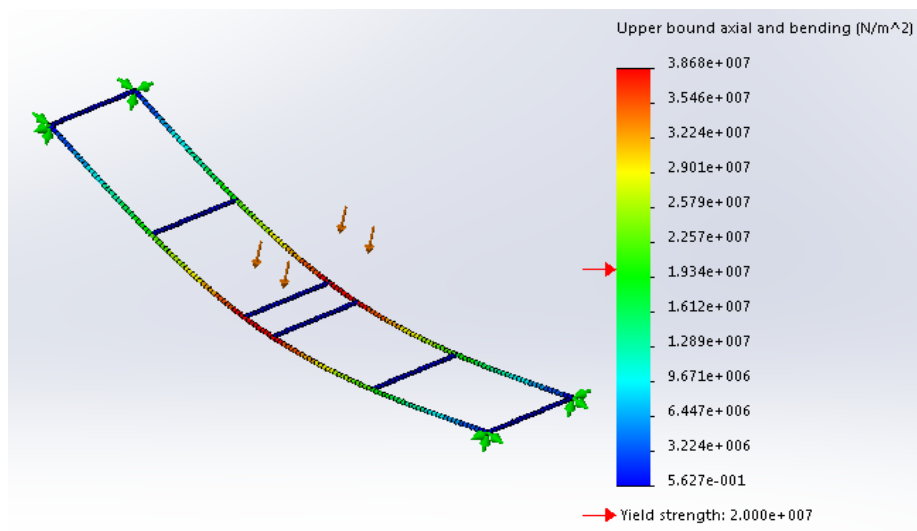
Lesson 3: Analyzing the Structure

Goals of This Lesson

- ❑ Understand what SOLIDWORKS Simulation does.
- ❑ Describe the stages of a Structural Analysis.
- ❑ Understand the environment of the analysis including fixtures and loads.
- ❑ Use SOLIDWORKS Simulation.
- ❑ View the results of an analysis.

Analysis of the Structure

During this lesson, you will use SOLIDWORKS Simulation to analyze the beam structure.



What is SOLIDWORKS Simulation?

SOLIDWORKS Simulation is a structural analysis tool for designers that is added into SOLIDWORKS. With this software you can analyze the solid model directly. You can also easily set up units, material type, external loads and more by using a study. You can make changes to the solid model and update the structural analysis results.

There are several steps to the analysis:

- 1 Create a design in SOLIDWORKS.

SOLIDWORKS Simulation can analyze parts and assemblies.

- 2 Create a new static study in SOLIDWORKS Simulation. SOLIDWORKS Simulation projects will contain all the settings and results of a problem and each project that is associated to the model. This includes: adding fixtures, adding external loads and meshing the model.
- 3 Run the analysis. This is sometimes called solving.
- 4 Viewing the SOLIDWORKS Simulation results which includes plots, reports and eDrawings.

Structural Analysis

Structural Analysis is an Engineering process that uses Physics and Mathematics to predict how a structure will behave under external loads such as weights and pressures. Buildings, bridges, aircraft, ships and automobiles are among the many products that require structural analysis.

Through structural analysis we can determine *Stresses*, *Factor of Safety* and *Displacements*.

Stresses: The external loads applied to a structure create internal forces and stresses that may cause the structure to fail or break.

Factor of Safety: The factor of safety (FOS) is a ratio of the actual stress divided by the maximum stress the material can handle.

$$\frac{\text{Maximum Stress under Loading}}{\text{Maximum Stress of the Material}} = FOS$$

If the **FOS > 1**, the structure is safe. If the **FOS < 1**, the structure is considered unsafe.

Displacements: As mentioned in a previous lesson, the external loads applied to a structure can force the structure to move from its unloaded position. The displacement is the distance a point moves from its original position.

Structural analysis is used in many fields of the manufacturing industry:

❑ Buildings and Bridges

Floors, walls and foundation.

❑ Aircraft

Aircraft fuselage, wings and landing gear.

❑ Ships

Hulls, bulkheads and superstructure.

❑ Automobiles

Chassis, body and crash testing.

Why Do Design Analysis?

After building your design in SOLIDWORKS, you may need to answer questions like:

- ☐ Does the truss cover the required span?
- ☐ What is the most efficient design for the truss?
- ☐ What is the maximum load that the truss can handle?

In the absence of analysis tools, expensive prototype-test design cycles take place to ensure that the product's performance meets customer expectations. Design analysis makes it possible to perform design cycles quickly and inexpensively on computer models instead. Even when manufacturing costs are not important considerations, design analysis provides significant product quality benefits, enabling engineers to detect design problems far sooner than the time it takes to build a prototype. Design analysis also facilitates the study of many design options and aids in developing optimized designs. Quick and inexpensive analysis often reveals non-intuitive solutions and benefits engineers by allowing them to better understand the product's behavior.

Structural Analysis Stages

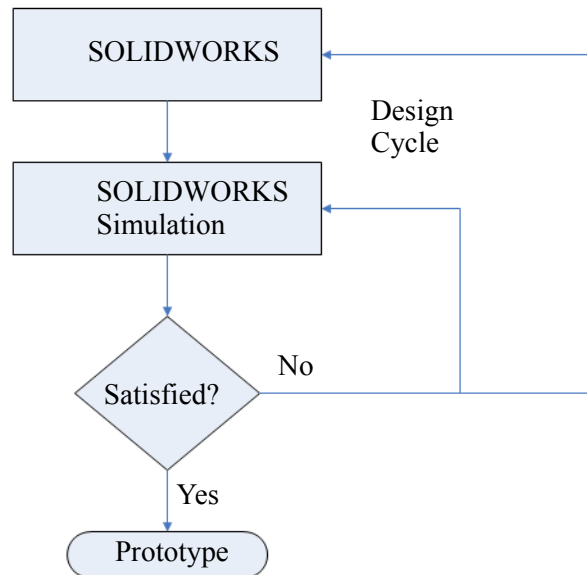
SOLIDWORKS Simulation walks you through several stages of structural analysis. This is what is happening behind the scenes:

- ☐ **Pre-Processing-** In this stage, you add the required information about the structure and the environment. This includes materials, fixtures and external loads applied to the structure.
- ☐ **Analysis-** The model is broken down into tiny pieces called elements using a process called meshing. In this project, the elements are **Beam Elements**. This information is then used to create a finite element model and is solved. This includes the **Analyze** page of the SOLIDWORKS Simulation wizard.
- ☐ **Post-Processing-** The results are presented to you in a graphic form so you can identify the problem areas. This includes the **Optimize** and **Results** pages of the SOLIDWORKS Simulation wizard.

Once all the stages are complete, you can save all the analysis information with the model. When the analysis information is saved, future changes will be faster.

Design Cycle

The **Design Cycle** is used to make a change to the model or the pre-processing information. Model changes would be size changes such as the length of beams. Changes to the pre-processing information would include changes to the material, fixtures or loading. Either change forces the model to be re-analyzed, cycling until the best solution is reached.



Changes in the Model

The SOLIDWORKS part is now very simple, but sides and braces will be added and you will see why they are important aspects of the structure. Let's open it and take a look at the model and what it represents.

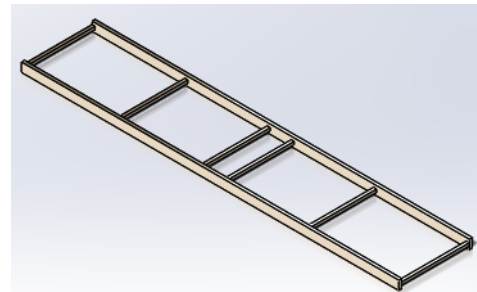
1 Open the part file again.

Click **Open** .

From the **Open** window, browse to the *Bridge Design Project\Student\Lesson 2* folder.

Select *TRUSS_1.sldprt* and click **Open**.

This is the same part that was used in the previous lesson.



Create a Study

In order to perform a structural analysis, a new study must be created.

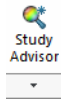
SOLIDWORKS Simulation uses a **Study** to store and organize all the data associated with a structural analysis.

The study is also used to specify the type of analysis that you are running. Many types are available. They include:

- ☐ **Static**
- ☐ **Frequency**
- ☐ **Buckling**
- ☐ **Thermal**
- ☐ **Drop test**
- ☐ **Fatigue**
- ☐ **Nonlinear**
- ☐ **Linear Dynamic**
- ☐ **Pressure Vessel Design**

In this project we will be using a **Static** analysis. This type of study is used to predict where a structure will fail due to stress.

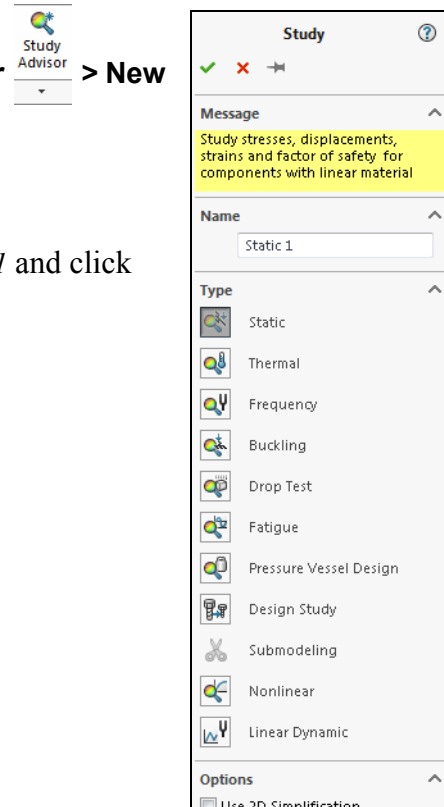
Where to Find It

- CommandManager: **Simulation > Study Advisor**  **> New**
- Menu: **Simulation, Study**

2 Create a new study.

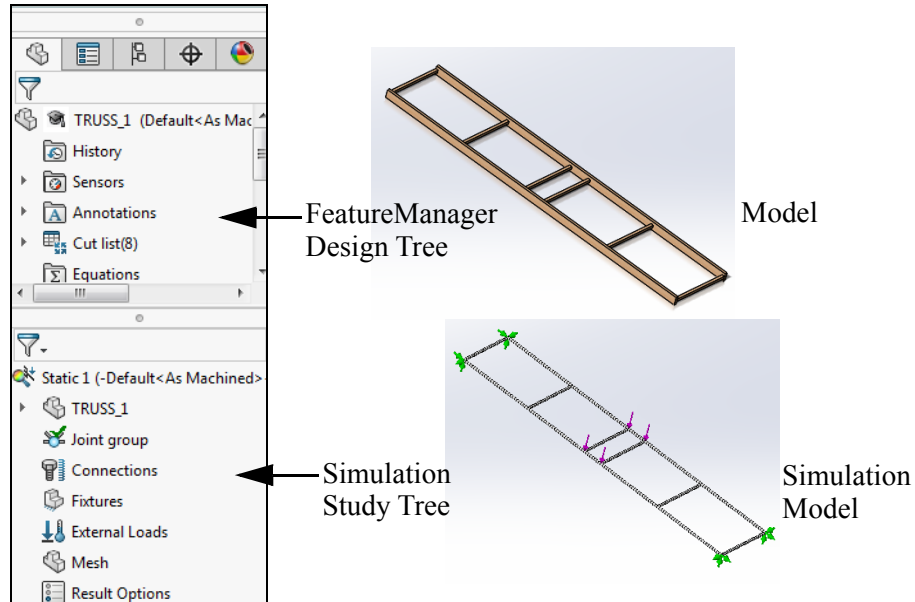
Click **Simulation, Study**. Use the default name *Static 1* and click **Static**.

Click .



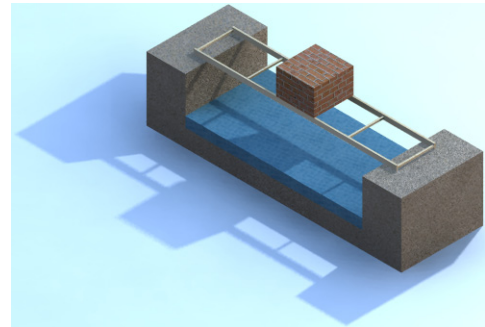
FeatureManager Design Tree and Simulation Study Tree

The **FeatureManager Design Tree** appears above the **Simulation Study Tree** on the left side of the screen. The upper tree lists the features of the model geometry while the lower tree lists the features of the analysis or simulation model.



The Environment

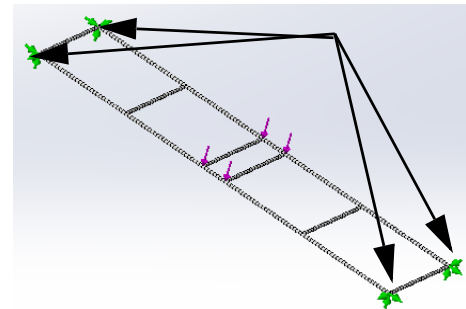
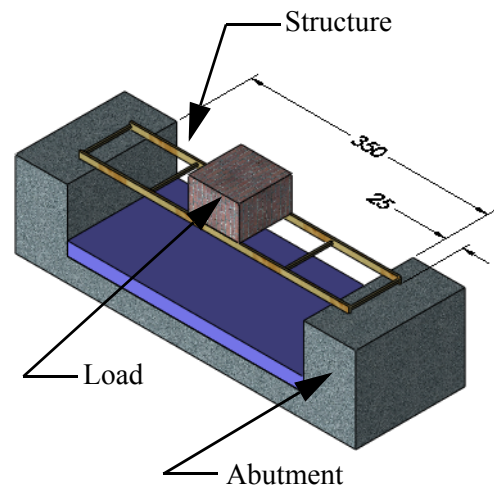
The environment describes how the structure is used. In this case, the model represents a structure crossing a river. From knowing the placement of the structure and the external loads that must cross it, we can determine two critical items required for SOLIDWORKS Simulation: the **Fixtures** and the **External Loads**.



Fixtures

The **Fixtures** are the areas of the structure that will be fixed or limited in movement. We define the span as the crossing distance that is not supported, **350mm** in this case. On each end, there is **25mm** of overlap where the structure ends are supported by the abutment or shore. The span is always less than the full length of the structure.

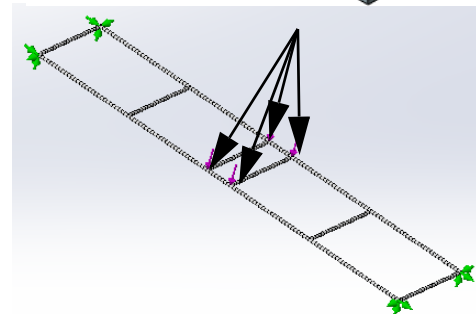
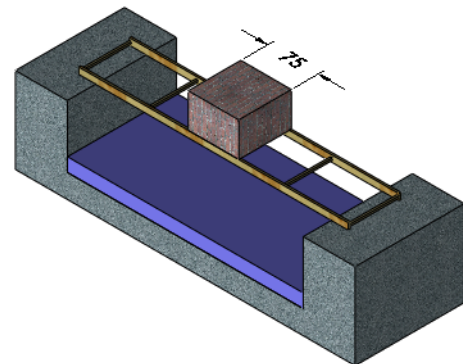
The fixtures are defined at the ends of the model in four places.



External Loads

The model must have **External Loads** that impose forces onto the structure. Let's say a rectangular stack of bricks is sitting in the center of the span, crossing the structure. Assume that the total weight of the bricks is **40N**.

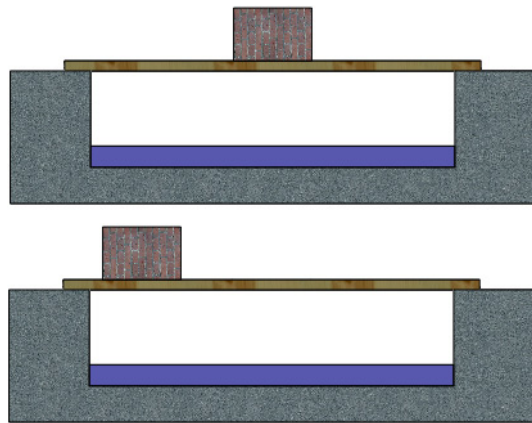
There are four loading points, one for each point where the beams connect near the center of the span. This means that the load on each point is $40\text{N}/4 = 10\text{N}$ (about 2.25lbs).



Why is the load in the center?

When using a structural analysis model, engineers like to perform what is called a “worst case” analysis. This is the situation where the structure is most likely to break due to the conditions of the environment.

Placing the load at the center of the span is the worst case for a truss structure like this.



How much do you think it will hold?

The structure is fairly weak at this point, but you will strengthen it as you go through this guide. What is the maximum force it can withstand? Take a guess.

Force = _____ N

Note: If you are thinking in terms of pounds (lb), start thinking in metric terms. Convert pounds to newtons (N) using this formula:

$$1 \text{ lb} = 4.4482 \text{ N}$$

Unit Settings

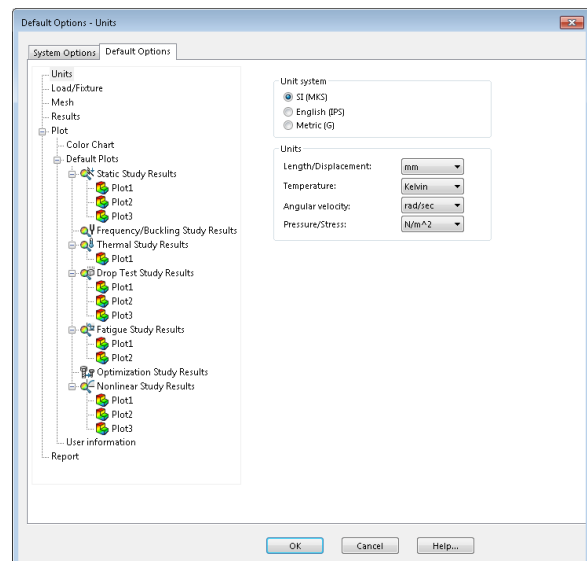
The **Options** can be set to create consistent results throughout the analysis. In this example, mm and MPa will be selected for use.

Where to Find It

- Menu: **Simulation, Options**

3 Set the units.

Click **Simulation, Options** and click the **Default Options** tab. Under **Units**, select **mm** for **Length/Displacement** and **N/mm² (MPa)** for **Pressure/Stress**. Click **OK**.



Pre-Processing

The first stage of the structural analysis is the pre-processing, gathering all the required information and applying it to the simulation model. The information that we will supply or create includes:

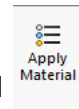
- ❑ **Material** - The material of the beams.
- ❑ **Fixtures** - Positions that cannot move freely.
- ❑ **External Loads** - Forces that are applied to the model.
- ❑ **Mesh** - A simulation model, based on the model, that breaks the beams up into small pieces called elements.

Material

The Material is a required value that sets the material properties and appearance of the model geometry. In this case it will be applied to all the beams at once.

Where to Find It

- CommandManager: **Simulation > Apply Material**
- Menu: **Simulation, Material, Apply Material to All**
- Simulation Study Tree: Right-click the part name and select **Apply/Edit Material**

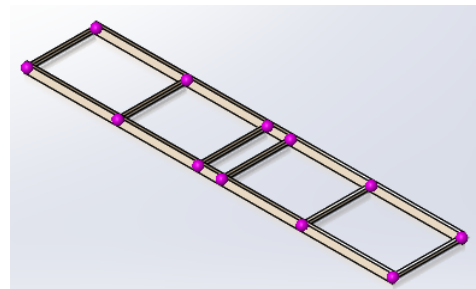


4 Set the material.

Click **Simulation, Material, Apply Material to All**. Expand the *Woods* folder and select *Balsa*. Click **Apply** and **Close**.

What are Joints?

Joints are generated automatically where the centerlines of beams meet. These joints will be used to locate the fixtures and external loads that follow.

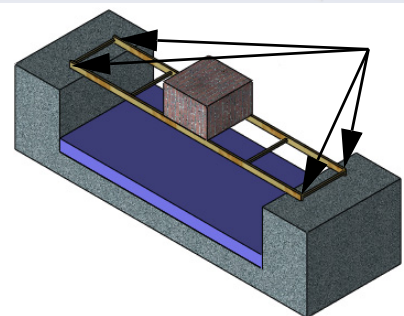


Fixtures

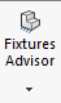
Fixtures are used to limit movement of certain points in the model. The points where the ends of the structure sit on the abutment will be assigned fixtures.

What type of fixtures?

In this project, the bridge will be placed on the abutment so that it crosses the span. The bridge will contact the abutment but it will not be glued or attached in any way.

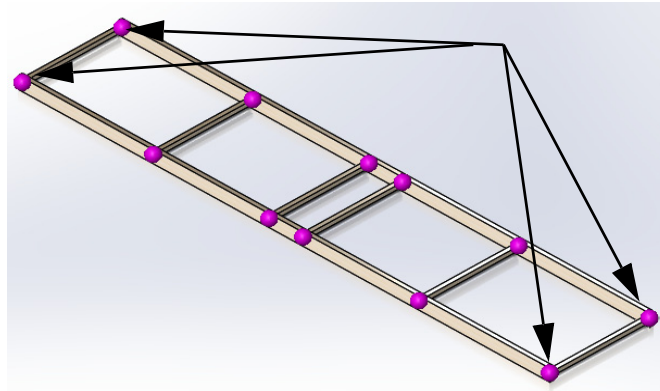


Where to Find It

- CommandManager: **Simulation > Fixtures Advisor**  **> Fixed Geometry**
- Menu: **Simulation, Loads/Fixture, Fixtures**
- Simulation Study Tree: Right-click **Fixtures** and select **Fixed Geometry**


5 Add fixtures.

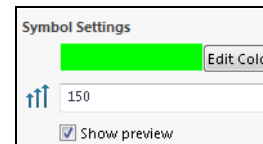
Click **Simulation, Loads/Fixture, Fixtures**. Click **Immovable (No translation)**  and select the joints as shown.



Note: To correct errors, right-click in the box where the selections are listed and select **Clear selections**. When the box is emptied, try selecting again.

6 Size of symbols.

Expand the **Symbol Settings** section and increase the **Symbol Size** to **150**. This makes the symbols larger and more visible. Click .

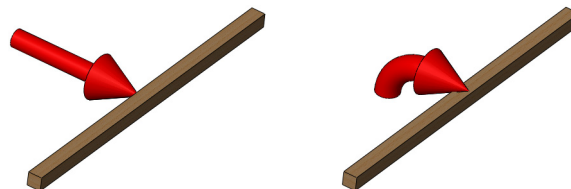


External Forces

The total force on the structure will be divided equally into four **5N** forces placed near the center of the structure.

Forces

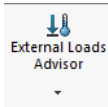
Forces have direction and a value (magnitude). They can be a direct *force* like hanging a weight or a *moment* that twists or bends like turning a doorknob.



Gravity

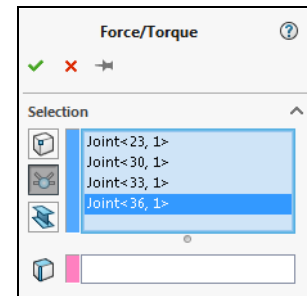
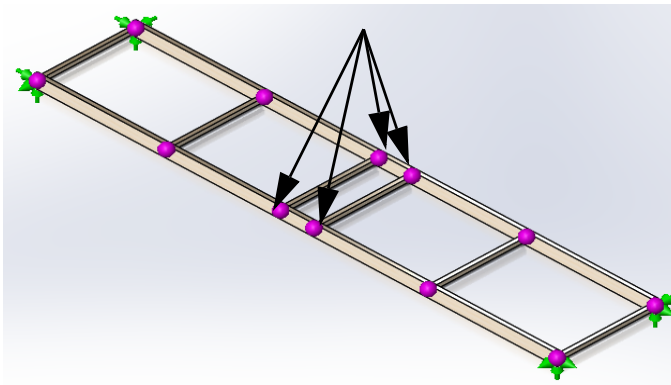
Gravity uses the weight of the structure as a load. It is not significant in this project and will not be considered.

Where to Find It

- CommandManager: **Simulation > External Loads Advisor**  **> Force**
- Menu: **Simulation, Loads/Fixture, Force**
- Simulation Study Tree: Right-click **External Loads** and select **Force**

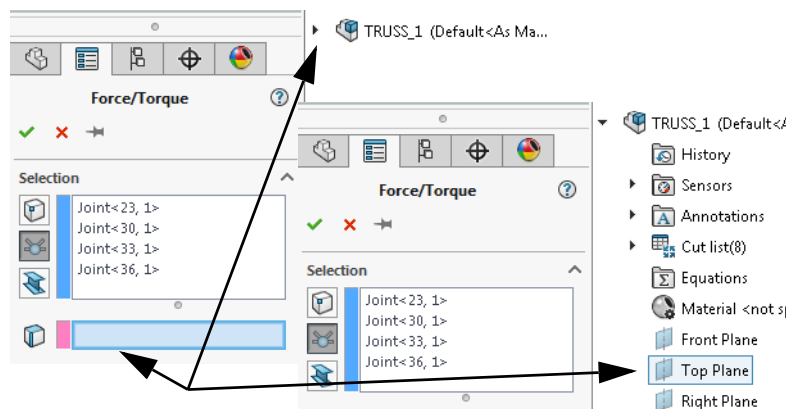
7 Add forces.

Click **Simulation, Loads/Fixture, Force**. Click **Joints**  and select the visible joints as shown.



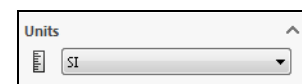
8 Set direction.

Click in the **Direction** field and expand the Flyout FeatureManager Design Tree. Click the feature **Top Plane**.




9 Set units.

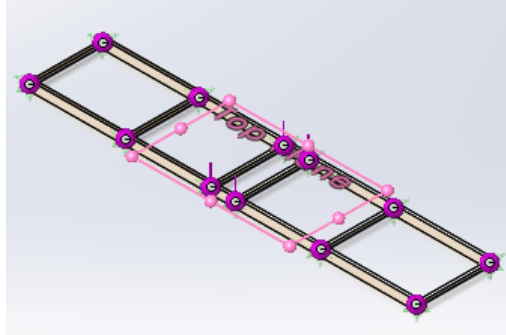
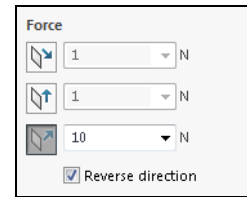
Make sure that **Units** are set to **SI**.



10 Assign force.


Click **Normal to Plane**  and set the value to **10N** as shown. Click **Reverse direction** to get the arrows pointing down.

Click .



Tip: The **Symbol Settings** options can be used like those in fixtures to increase or reduce the size of the symbol. These have been set to **150**.

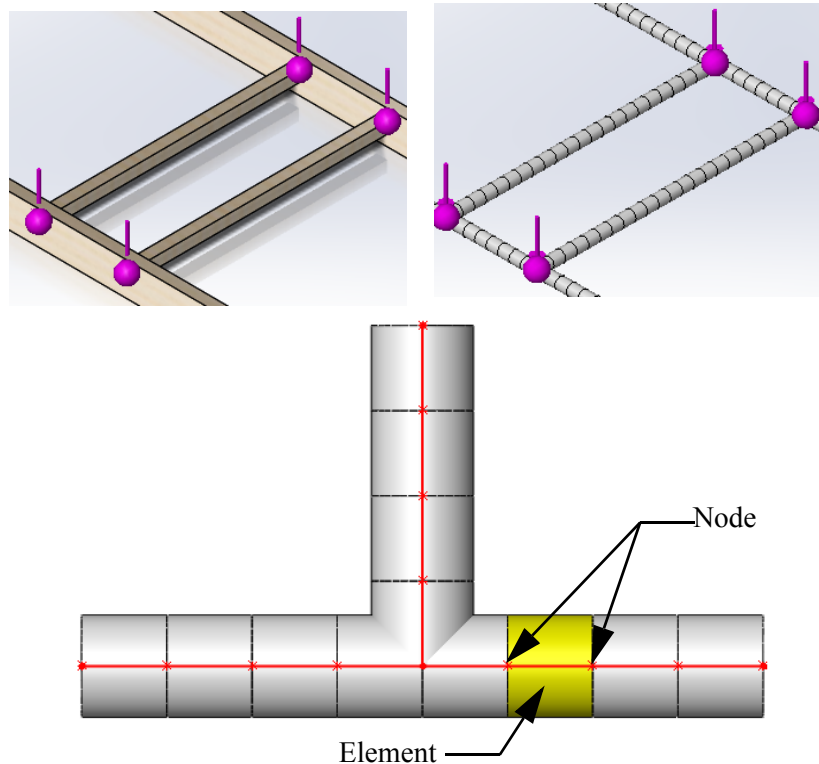
11 Save.

Click **Save**  to save the model and simulation data.

Tip: It is a good idea to save periodically and prevent unintentional loss of data.

Meshing the Model

The mesh must be created to generate the small pieces used in the analysis. The analysis model is made up of a series of connected nodes and elements.

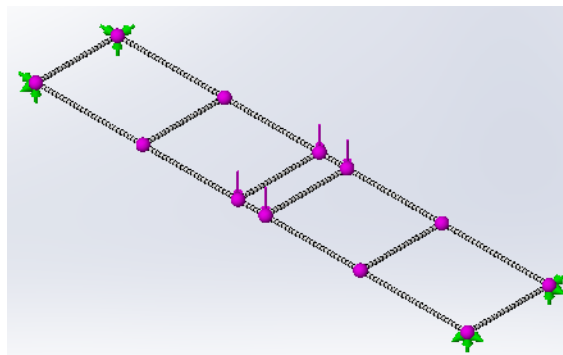


Where to Find It

- CommandManager: **Simulation > Run**  **> Create Mesh** 
- Menu: **Simulation, Mesh, Create**
- Simulation Study Tree: Right-click **Mesh** and select **Create Mesh**

12 Meshing.

Click **Simulation, Mesh, Create**. A mesh is created using the geometry of the model.



Note: This step is automatically included within **Simulation, Run** but is shown here to highlight the mesh.

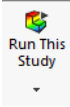

Analysis

The analysis portion is the easy part. SOLIDWORKS Simulation takes your input and does the work to find the results. You will use the default settings so that the results will be faster.

Expectations

In the previous lesson, beam calculations were used to determine a rough displacement based on a simplified analysis of a simply supported beam. That analysis determined that the displacement was approximately **35mm**. We expect that the displacement we get from the simulation analysis falls in the same order of magnitude; between **3.5mm** and **350mm**; hopefully close to the **35mm** result.

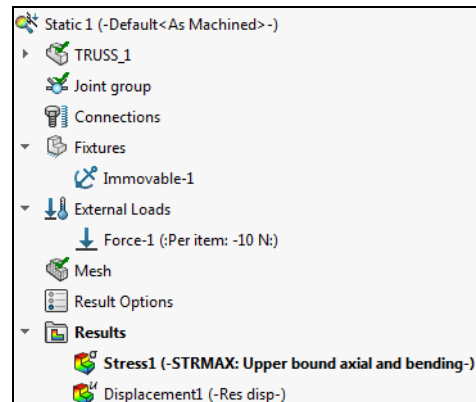
Where to Find It

- CommandManager: **Simulation > Run This Study**  **> Run This Study** 
- Menu: **Simulation, Run, Run**
- Simulation Study Tree: Right-click the study name and select **Run**

13 Run.

Click **Simulation, Run**. When the run is complete, you will see two features in the *Results* folder of the Simulation Study Tree.

The simulation is ready for post-processing.



Some Terminology

While the analysis is running, let's look at some terminology that will help with interpretation of the results.

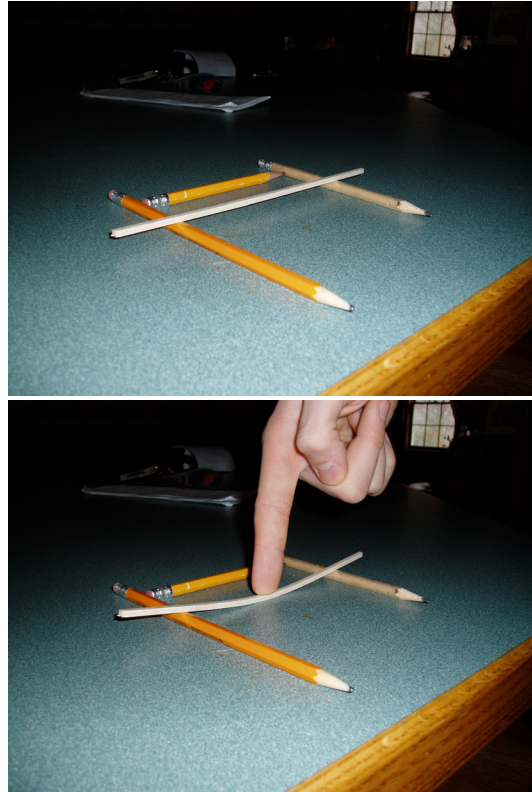
Bending and Displacement

Bending is caused by a load that is applied to a beam. The load causes the beam to bend and move in the direction of the load.

The **Displacement** is the movement of the beam from its original position. The “worst case” displacement occurs when the load is at the center of the beam.

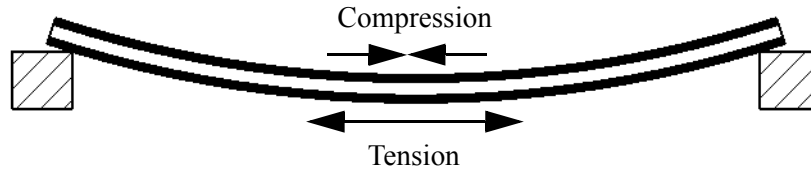
You can see displacement if it is large enough, but it is usually very small.

Is there a place in your house where the floor creaks when you walk over it? The creaking is caused by the displacement of the floor beam bending under a load- your weight!



Tension and Compression

While the beam bends, the top portion of the beam (the face where the load is applied) compresses (pushing together) while the opposite face sees tension (pulling apart).



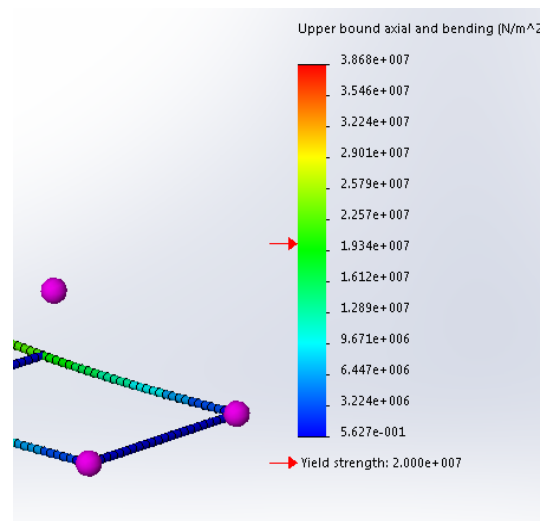
Search on **tension and compression** for more information.

Stresses

Stress is a quantity measured by force per unit area inside a structure that is caused by external loads applied outside of the structure. You cannot see stress but it can cause your structure to break.

Common units are Newtons per meter squared, Pascals and pounds per square inch (psi).

Stress can cause the beam to break under a load. SOLIDWORKS Simulation provides maps that show areas of high and low stress on the structure.



Yield Strength

How much can the beam take before it breaks? We use the **Yield Strength** as the limit of the beam's strength based on the stresses that the beam sees. Both the material and beam section contribute to the strength.

Note: In metals, the material will often bend under load but will return to its original shape when the load is removed. The yield strength is the point where the material bends and stays bent after the load is removed. This is called a Plastic Deformation.

Factor of Safety

The **Factor of Safety** (FOS) is a quick way to see the results of the analysis. It is defined as the ratio of the highest stress and the yield stress of the material. If the **FOS > 1**, the structure is safe. If the **FOS < 1**, the structure is unsafe.

Note: Engineers generally design for a FOS of more than 2. Structures are generally "over designed" for safety and reliability.



Search on **stress (physics)**, **yield strength** or **factor of safety** for more information.

Post-Processing

Once the analysis is complete, post-processing can begin. Post-processing produces two plots in the *Results* folder of the Simulation Study Tree that can be viewed and modified. These plots will help you understand and modify the bridge structure.

As post-processing begins, two plots are posted in the *Results* folder:

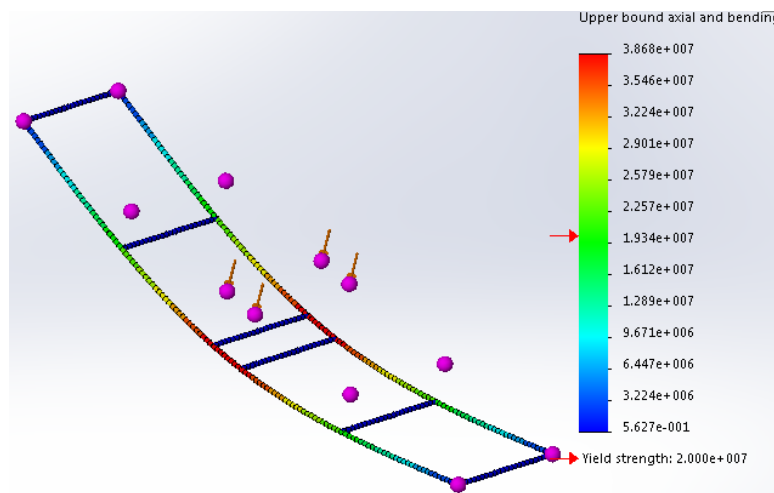
Stress 1 (-STRMAX-High axial and bending) and

Displacement1 (-Res disp-).

The stress plot is selected and viewed automatically.

14 Stress distribution.

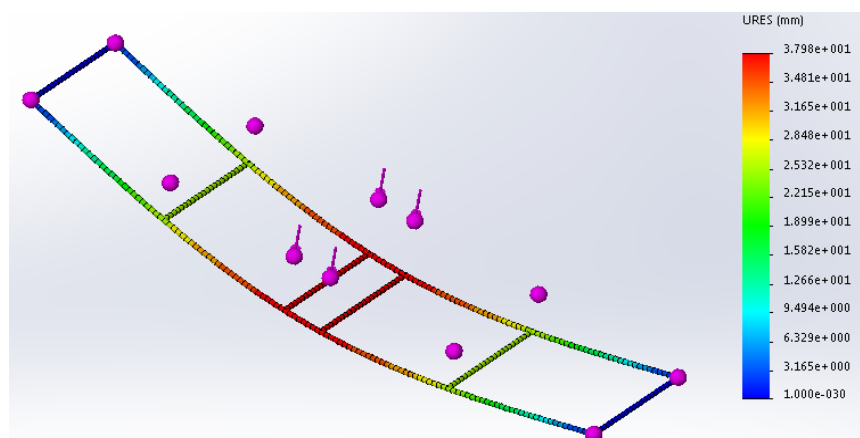
The display shows the model with displacement. The **Stress Distribution** is represented by the colors on the displaced model. The chart shows the distribution; warm colors for higher stresses, cool colors for lower stresses.



Note: The joints ● can be hidden. Right-click *Joint group* and select **Hide** or **Show**.

15 Displacement.

Double-click the *Displacement1 (-Res disp-)* plot to view it.



Interpreting the Results

The stress and displacement plots are helpful because they tell us the actual values and where they are highest. What is a MPa? Lets get some sense of what the results mean. Here are results so far (*yours may vary*):

Stress	Displacement
38.683MPa (Megapascals)	3.798e+001mm

Numbers

The displacement is shown in scientific notation. (your results may have a different combination of formats).

3.798e+001 means 3.798×10^1 or $3.798 \times 10^1 = 3.798 \times 10 = \underline{\hspace{1cm}}$ mm

What is that in inches? Divide the result above by 25.4 = in

Units

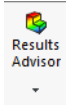

Understanding units is important in interpreting the results. Length units like mm or inches are familiar. Stress may not be. Stress units are those of pressure, measuring force/area. You may have seen psi (pounds per square inch) when you pump up a bicycle tire. Here is a tire pressure in common units:

60 psi = 4.136854×10^5 Pa = 0.4136854 MPa ($1 \text{ MPa} = 1 \text{ N/mm}^2 = 1,000,000 \text{ Pa}$)

Creating a New Plot

What we need to know is: how much stress can the structure withstand? The best solution is to create a **Factor of Safety** plot. It is a three step process.

Where to Find It

- CommandManager: **Simulation > Results Advisor**  **> New Plot > Factor of Safety** 
- Simulation Study Tree: Right-click on the *Results* folder and click **Define Factor of Safety Plot**

16 Factor of Safety Plot.

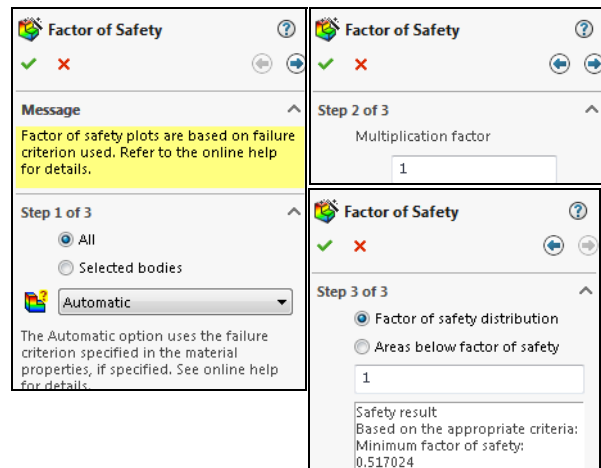
Right-click on the *Results* folder in the Simulation Study Tree and select **Define Factor of Safety Plot**.

Keep the default settings and click **Next** .

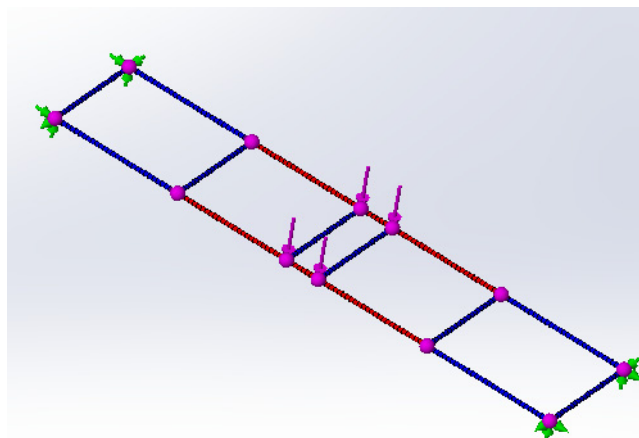
Keep the **Multiplication factor** at 1 and click **Next** .

Click **Areas below factor of safety**.

Click ✓.



Note: The current factor of safety is listed as 0.517024, or about 0.5, in the dialog box. This is less than the minimum value of at least 1. The plot appears with everything colored blue or red.



What does the Factor of Safety Plot Tell Us?

The areas below factor of safety are shown as red on the plot. If a FOS of **1** is the limit, that means that the loading is too heavy for the structure to support.

The load must be reduced.

Iterating Changes

Since the structure cannot support the load, the next step is to find out what load the structure can support. To do this, we will change the load, re-analyzing the structure until we can get the FOS to about 1. This is called *iterating*.

Determine the Load

Before we iterate a change and decrease the load, we need to decide how much of an decrease is required. The current information tells us that the FOS is about **0.5** for a load of $4 \times 10\text{N} = 40\text{N}$.

If we multiply the FOS times the total load, the result should produce a FOS of about 1.

$\text{FOS} \times \text{Total Load} = 0.5 \times 40\text{N} = 20\text{N}$ or **5N** per face

Using iteration, we will reanalyze the model to see if this formula can be validated.

Editing Simulation Data


Simulation data, such as an external load, can be edited to reflect the new value. The results will not update until the analysis has been rerun.

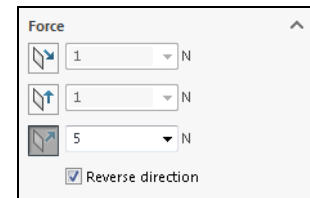
Where to Find It

- Shortcut Menu: Right-click the fixture or load feature and select **Edit Definition**

17 Edit external load.

Right-click the feature

Force-1 (:Per item: -10 N:) and select **Edit Definition**. Set the load to **5N** and click .



18 Rerun.

Click **Simulation, Run** to rerun the analysis.

19 Factor of Safety.

Double-click the result *Factor of Safety1 (-Automatic-)*. The FOS is blue meaning greater than 1.

20 Close the part.

Click **File, Close** and click **Save** to save changes.

Conclusion

From the analysis, it is obvious that the structure was inadequate to support the initial load. Using SOLIDWORKS Simulation, we were able to iterate and find the highest load that the structure could hold.

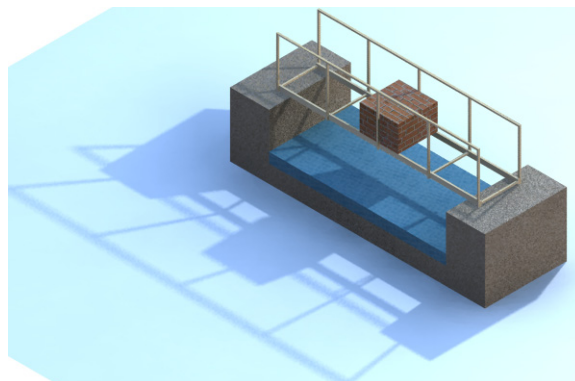
Lesson 4: Making Design Changes

Goals of This Lesson

- ❑ Understand the importance of cross bracing.
- ❑ Find the maximum load.
- ❑ View displacement plots.
- ❑ Edit plots and charts to enhance viewing.
- ❑ Calculate the strength to weight ratio.

Adding to the Design

Based on the analysis of the structure using SOLIDWORKS Simulation, we can conclude that the structure needs strengthening. This version has added side walls that strengthen the design and allow it carry higher loads.



Open the Model

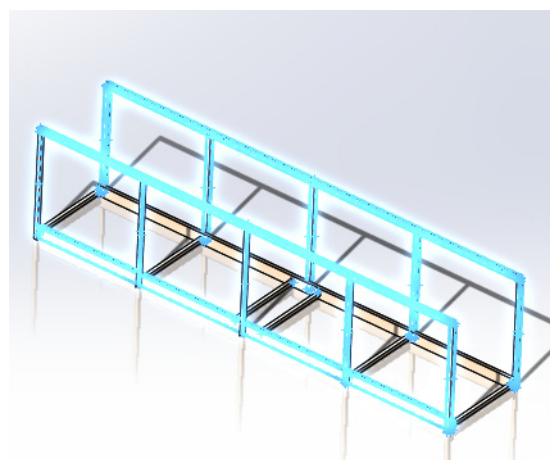
1 Open the part file.

Click **Open** .

From the **Open** window, browse to the *Bridge Design Project\Student\Lesson 4* folder.

Select *TRUSS_2.sldprt* and click **Open**.

This version has sides made up of horizontal and vertical members.

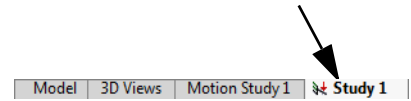


Existing Study

This part is the same as the previous one with the addition of the walls. It also has a study *Study 1* that uses the same values as the previous part.

2 Access an existing study.

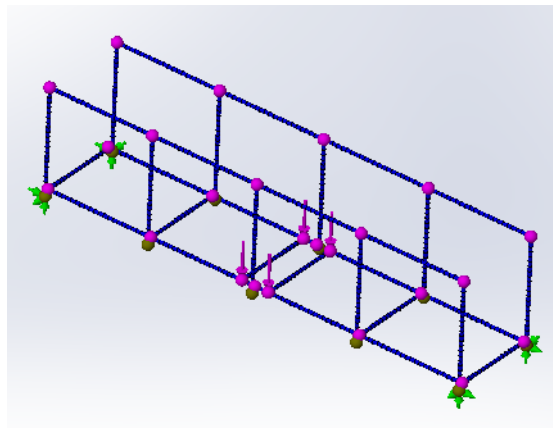
Click the *Study 1* tab located on the lower left portion of the screen. The Simulation Study Tree appears. The analysis has fixtures, external loads and mesh.



3 Run the analysis.

Click **Simulation, Run, Run**.

The simulation is ready for post-processing. Note that the factor of safety plot is *not* created automatically.

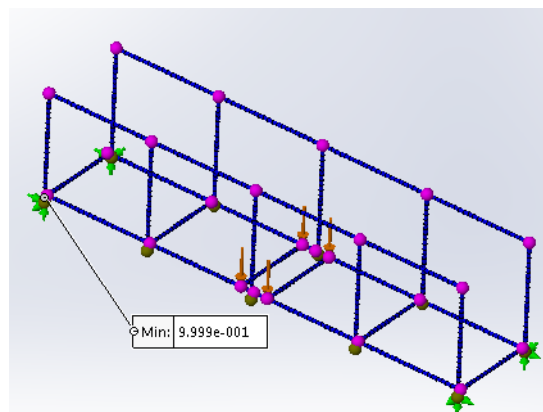


4 Factor of Safety Plot.

Right-click on the *Results* folder in the Simulation Study Tree and select **Define Factor of Safety Plot**. Use the same procedure as Creating a New Plot on page 40.

5 Labels.

Right-click the result *Factor of Safety1 (-FOS-)* and select **Chart Options**. Click **Show min annotation** and click ☒.




The results show that the FOS has decreased compared to a similar initial load in the previous lesson.

Change the Load

Let's iterate the external loads to see how much this version of the structure can take, again using a FOS of **1** as the target. In the previous lesson we learned that multiplying the total load by the factor of safety produced the maximum allowable load.

$$20\text{N} \times 0.684 = 13.68\text{N} \text{ each load is } 13.68\text{N}/4 = \mathbf{3.42\text{N}}$$

6 Edit external load.

Right-click the feature *Force-1 (:Per item: -5 N:)* and select **Edit Definition**. Set the load to **3.42N** and click .

7 Rerun.

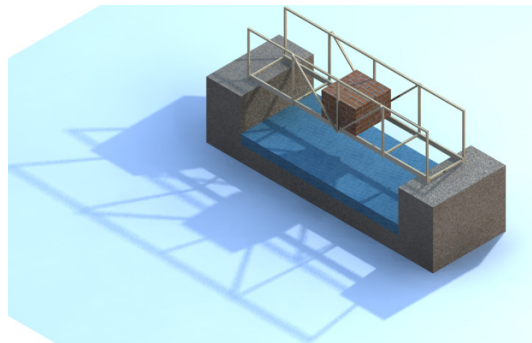
Click **Simulation, Run** to rerun the analysis. The minimum Factor of Safety should again be close to **1**.

8 Close the part.

Click **File, Close** and click **Save** to save changes.

Cross Bracing

In a previous lesson, the value of triangles and cross bracing was discussed (Triangles on page 8). We will look at a structure with some cross bracing to see how it changes the results. As before, the loading remains at the previous setting (**3.42N** in four places) and everything is the same with the exception of the added bracing.



Open the Model

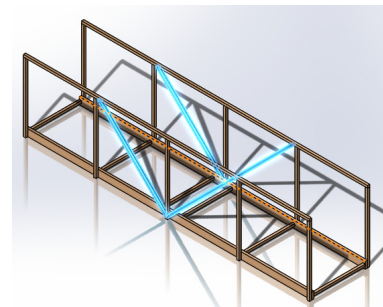
1 Open the part file.

Click **Open** .

From the **Open** window, browse to the *Lesson 4* folder.

Select *TRUSS_3.sldprt* and click **Open**.

This version is similar to the previous one with the addition of some cross bracing in the center section.

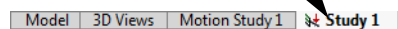


Existing Study

This part is the same as the previous one with the addition of some cross bracing. It also has a study *Study 1* that uses the same values as the previous part.

2 Access an existing study.

Click the *Study 1* tab located on the lower left portion of the screen. The Simulation Study Tree appears. The analysis has fixtures, external loads and mesh.



3 Run the analysis.

Click **Simulation, Run**.

The simulation is ready for post-processing. Create the FOS plot. The value is greater than 1 (add labels using the procedure in step 5 on page 44).

4 Edit external load.

Right-click the *Force-1* feature and select **Edit Definition**. Set the load to **5.4N** and click .

5 Rerun.

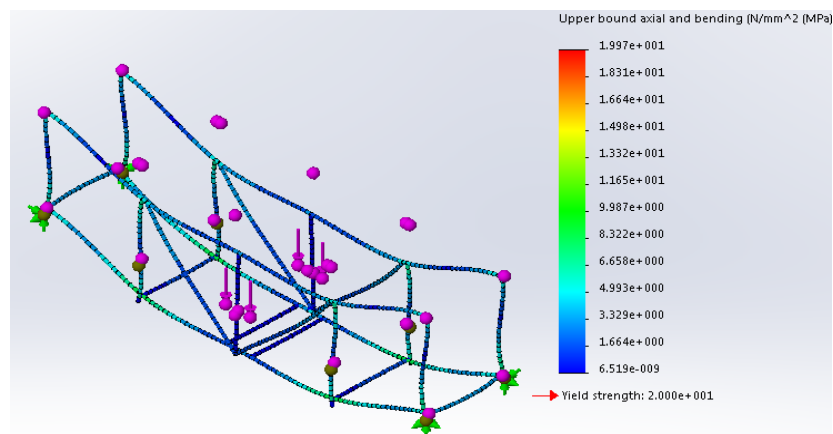
Click **Simulation, Run, Run** to rerun the analysis. The minimum FOS should again be close to 1.

What did the Cross Bracing do?

Cross bracing creates triangles that “stiffen” the frame and help it resist bending and twisting. To see how effective it is, we’ll look at the results.

6 Stress plot.

Double-click the *Stress1 (-STRMAX: Upper bound axial and bending-)* plot to see the stress plot.




Working with Plots

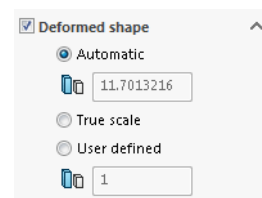
There are many options that can be used to make plots easier to read and understand. We will look at some options to change the appearance.

Deformation Plot Factor



The deformed shape of the stress plot may use an exaggerated displacement, which can be very large. To exaggerate the displacement, you can set the deformed shape to an **Automatic** or **User defined** value of your choice.

7 Deformed shape.



Right-click the *Stress1 (-STRMAX: Upper bound axial and bending-)* plot and select **Edit Definition**. Make sure that **Deformed Shape** and **Automatic** are selected. Click .

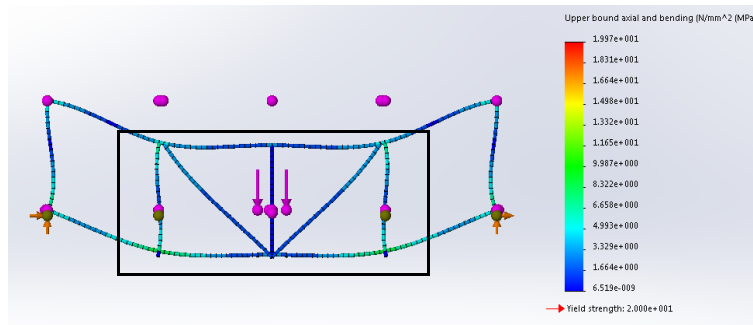


Where to Find It

- Heads-up View Toolbar: **View Orientation** , **Front** 
- Keyboard Shortcut: **Ctrl+1**

8 Front view.

Click **Front**  from the **View Orientation**  icon and look at the stress distribution of the model from the front. The center, cross braced, section of the model retains its shape better than the end portions due to the strength added by the bracing.



Superimposing the Model

The **Settings** options allow you to superimpose the undeformed shape and change the appearance of the chart to have show distinct color changes.

Where to Find It

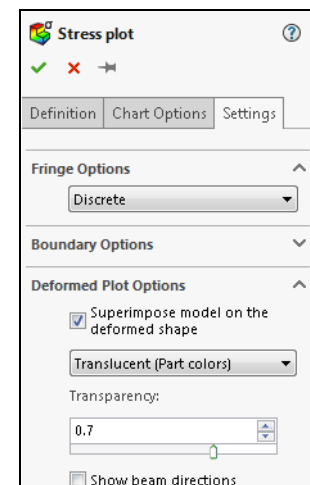
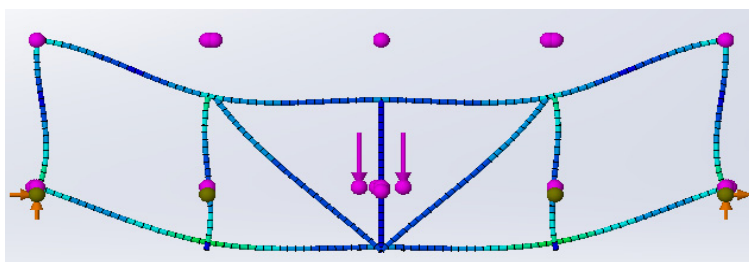
- Shortcut Menu: Right-click a plot and click **Settings**

9 Settings.

Right-click the *Stress1 (-STRMAX: Upper bound axial and bending-)* plot and click **Settings**.

Under **Fringe Options**, select **Discrete**. Under **Deformed Plot Options**, click **Superimpose model on the deformed shape** and set the **Transparency** to **0.7**.

Click .



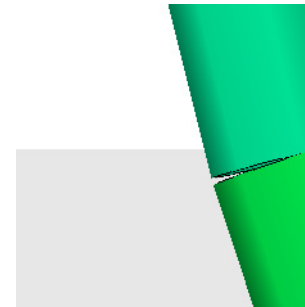
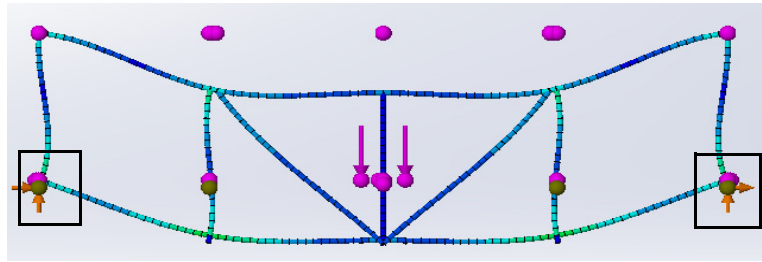
Note: The title and color charts can be moved by drag and drop.

The Weakest Link

Are you familiar with expression “the weakest link”? The literal meaning is the most vulnerable portion of a chain, the link that is most likely to break.

If you look closely at the lower left section of the image, you will see the label of the highest stress value. This is the weakest link, a *high stress* area.

There should be a similar high stress area (red) on the right side near the fixture. Zooming in will show it.

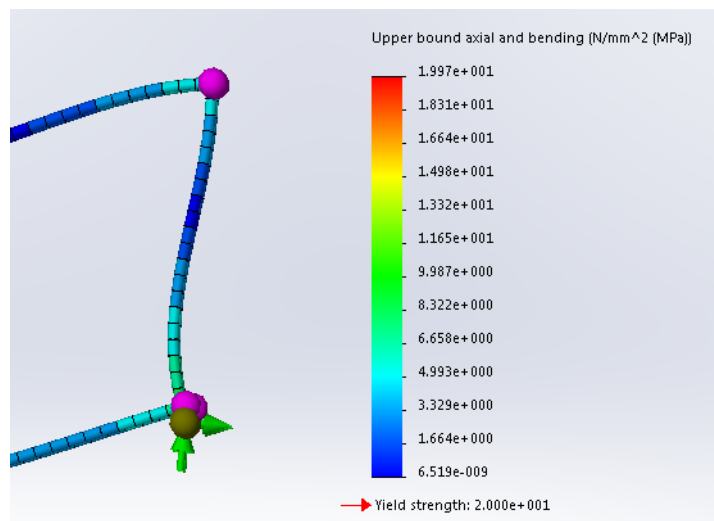


Stress Distribution Colors

The stress distribution always includes a color chart that allows you to match the colors with real stress values. The highest stress is a red/orange/yellow, the lowest is shades of blue.

SOLIDWORKS Simulation is used to identify the “weakest links” in the model so they can be repaired.



Keep in mind that the highest stress may not cause the structure to fail. Follow the **Yield strength** arrow, that is the failure point.



Using a Probe

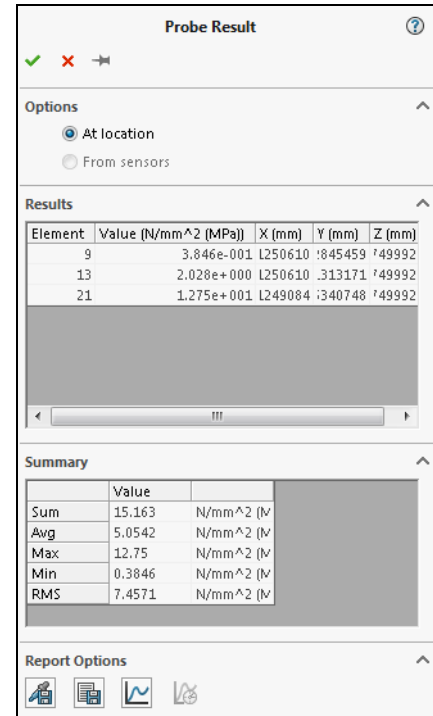
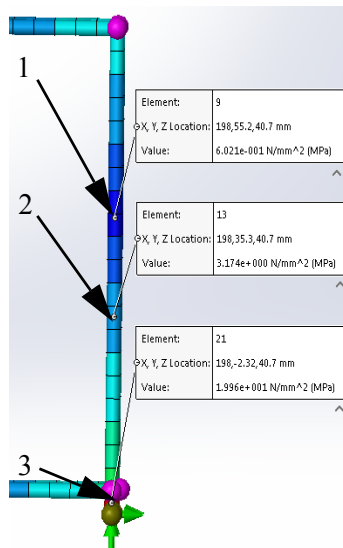
Probes allow you to get deeper information from a plot by selecting elements directly. The element will receive a label that displays the exact value, per the type, of that element. Plots can also be generated from the probe data.

Where to Find It

- CommandManager: **Simulation > Plot Tools**  > **Probe** 
- Menu: Click **Simulation, Result Tools, Probe**


10 Add a probe.


Click **Simulation, Result Tools, Probe**. Select the elements from top to bottom, in order, as shown. The values show that the stress value increases steeply from the first to the last elements selected.

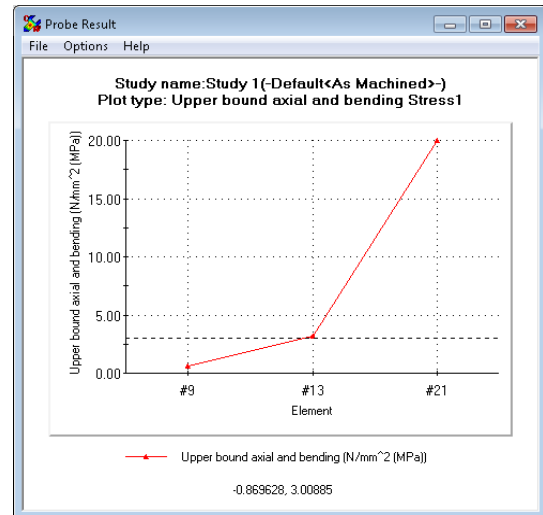


Note: Make selections similar to those shown here. The labels that you see may have slightly different element numbers and values.

11 Plot.


Click **Plot**  to create the **Probe Result**. The change in the stress value across those few elements is dramatic as demonstrated by the plot.

Click the “x” to close the **Probe Result** dialog and click  to close the **Probe Result** Property Manager.

**12 Isometric.**

Click **Isometric**  from the **View Orientation**  icon.

13 Deformed shape.

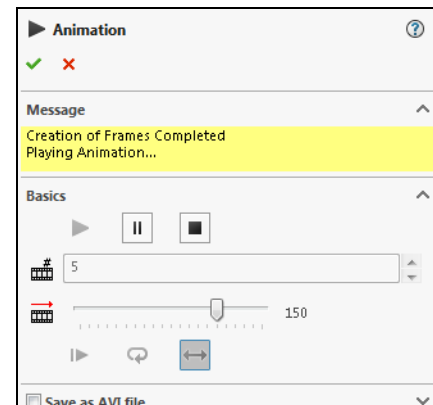
Right-click the *Stress1 (-STRMAX: Upper bound axial and bending-)* plot and select **Edit Definition**. Click **Deformed Shape** and **Automatic**. Click .

14 Animate.

Click **Simulation, Result Tools, Animate**. Move the **Speed** slider to the value **10** as shown.

Click .

Tip: The **Frames** slider can be used to create a smoother animation by increasing the number of frames.

**Adjusting the Number Format**

The values that accompany the charts use a number format based on the size. For instance, if the numbers are very small or very large, a scientific notation is used. You can change the number format to make the charts easier to read. Here is the same number in three different number formats.

Scientific	Floating	General
3.727e+000	3.727	3.73

15 Displacement.

Double-click the *Displacement1 (-Res disp-)* plot. Displacement numbers tend to be small, and in this chart they run between 0 and about 3 mm. They are in scientific notation but would be easier to read in a decimal format.

Where to Find It

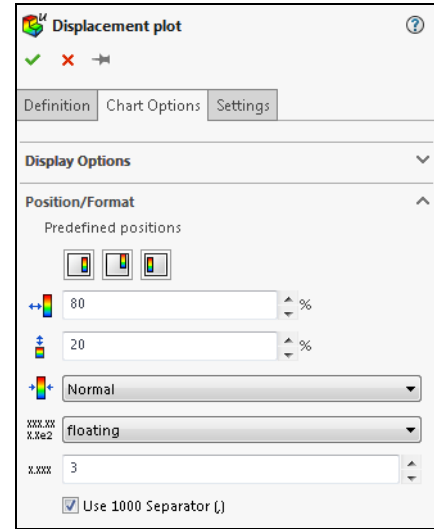
- Shortcut Menu: Right-click a plot and click **Chart Options**

16 Chart options.

Right-click the *Displacement1 (-Res disp-)* plot and click **Chart Options**. Under **Position/Format**, select **Number Format floating**.

The numbers appear as easier to read floating format numbers.

Click .



Solution

Now that the weak areas have been identified, they can be addressed. What do you think that the best solution to this problem is?

- 1 Decrease the load to increase the FOS to a value greater than 1.
- 2 Add cross bracing to the unbraced sections.

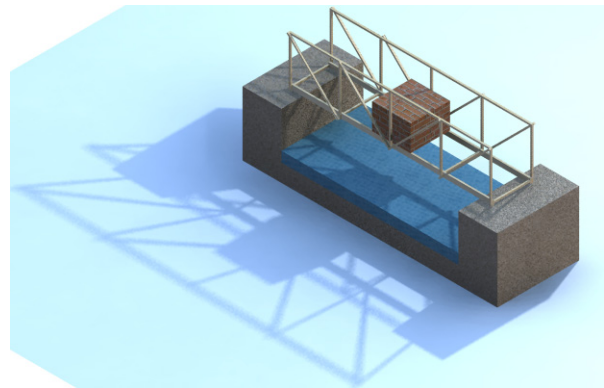
We will choose item 2 and then maximize the load on the structure.

17 Close the part.

Click **File, Close** and click **Save** to save changes.

Finishing the Bracing

To complete the cross bracing, members have been added in the outer sections. Let's see what this does to the structure.



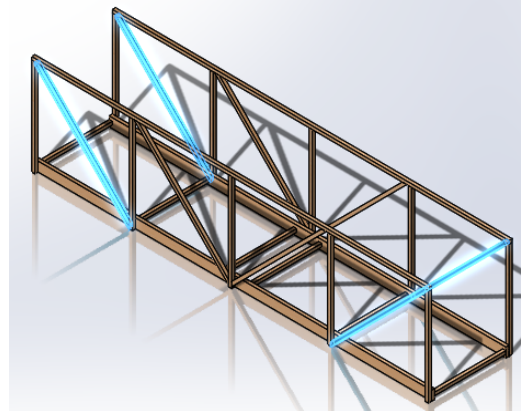
1 Open the part file.

Click **Open** .

From the **Open** window, browse to the *Lesson 4* folder.

Select *TRUSS_4.sldprt*, and click **Open**.

This version is similar to the previous one with full cross bracing.



2 Rerun.

Open the existing study *Study1* and rerun the analysis.

Compare Stresses

The added bracing seems to have been very effective. How can we tell? The maximum stress has been reduced.

Would you expect the FOS value to increase or decrease? _____

3 Factor of safety plot.

Create a factor of safety plot and check the value of the FOS.

4 Maximize the external load.

Again, we will maximize the load for a factor of safety of **1**.

$$5.64 \times 5.4\text{N} = \underline{\hspace{2cm}} \text{ N}$$

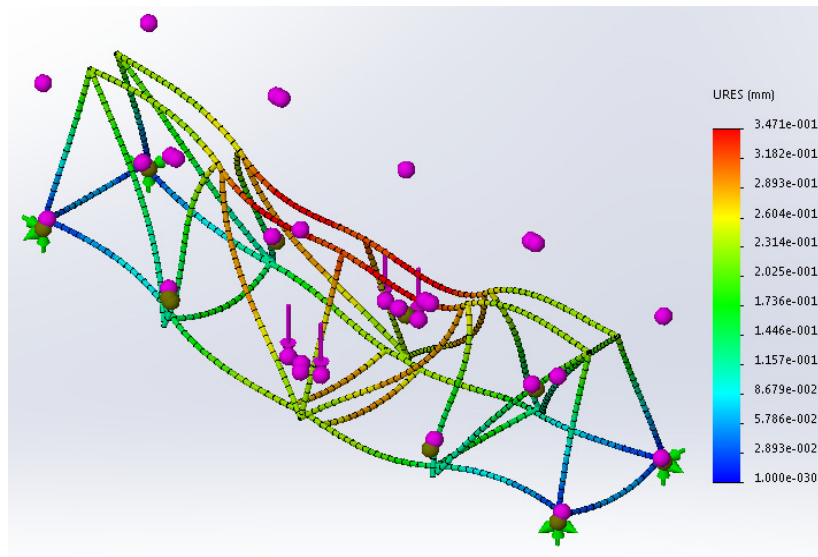
Edit the *Force-1* feature external load and set it **30.46 N**.

5 Rerun.

Click **Simulation, Run** to rerun the analysis. The minimum FOS should again be close to **1**.

6 Displacement.

Double-click the *Displacement1 (-Res disp-)* plot. Animate the plot.



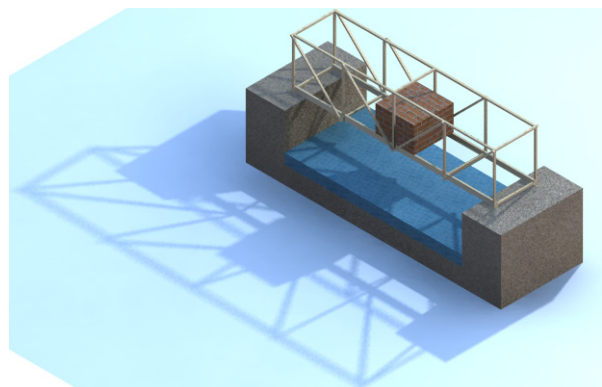
The displacements are smaller but you may notice that the model seems oddly shaped. The upper portions of the walls are bending inward. Some additional bracing is required.

7 Close the part.

Click **File, Close** and click **Save** to save changes.

Top Beams

To complete the structure, members have been added to the top of walls, connecting them. Let's see what this does to the structure.



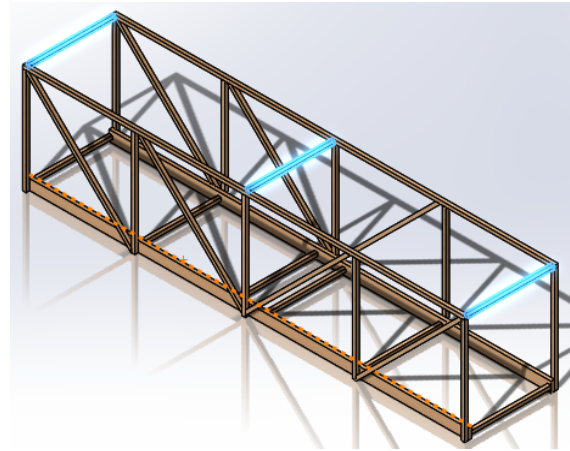
1 Open the part file.

Click **Open** .

From the **Open** window, browse to the *Lesson 4* folder.

Select *TRUSS_5.sldprt*, and click **Open**.

This version is similar to the previous one with three top braces added.



2 Open existing study.

Open the existing study *Study 1*.


3 Analysis and edits.

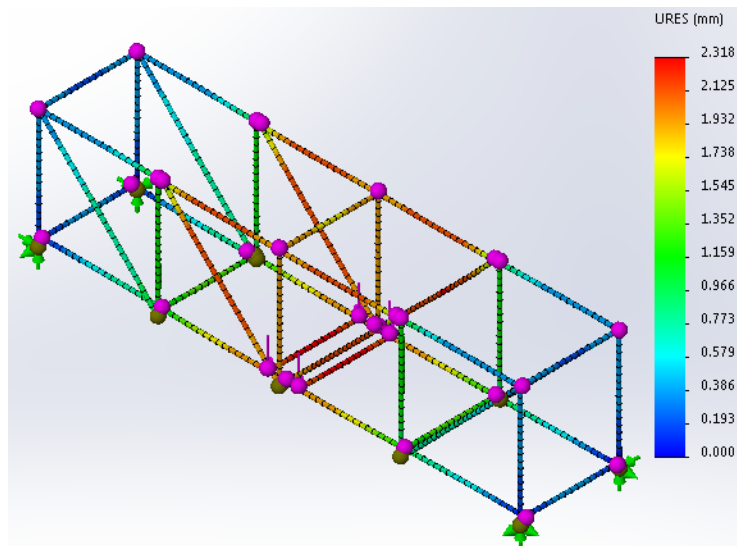
Run the analysis and create a FOS plot. The FOS falls above **1**.

To bring the FOS closer to 1, change the load to 37.95 and rerun.

4 Displacement.

Although the additional bracing did very little to change the maximum load, it reduces the maximum displacement.

Right-click the displacement result and select **Edit Definition**. Set the **Deformed Shape** to **True scale** and click . Also right-click **Chart Options** and select **Floating** as the **Number Format**.




Strength to Weight Ratio

This is just one of many structures that can be designed to support a load. If there were three different structures that could hold three different loads, how could you determine which design was the most efficient? The **Strength to Weight Ratio** (maximum load/structure weight) can be used.

What does our Structure Weigh?

Using SOLIDWORKS, finding mass properties is easy. They have been calculated for the model automatically.

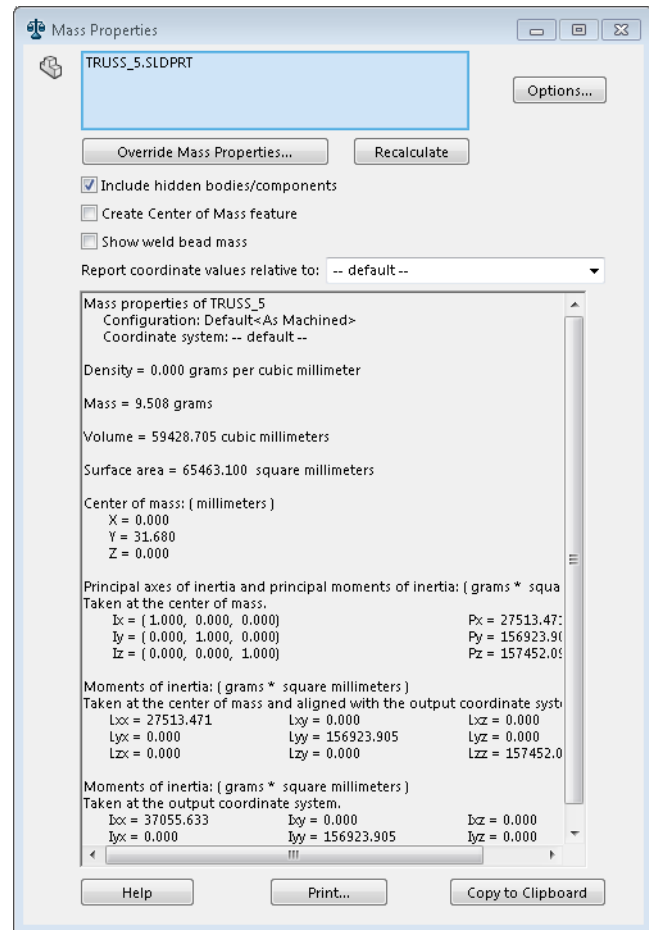
Where to Find It

- CommandManager: **Evaluate > Mass Properties** 
- Menu: **Tools, Evaluate, Mass Properties**

5 Mass properties.

Click **Tools, Evaluate, Mass Properties** to list the mass properties of the part. The key information is the line for **Mass**. This is the total weight of the structure in grams.

Click **Close**.



Note: Conversion of grams to newtons: 1 gram is about 0.01 newtons.

Efficiency Comparison

Use the information in the chart below to calculate the **Max Load Capacity** and **Efficiency** for each iteration in the design. Which design is the most efficient?

Structure	Max Load	Weight of Structure	Efficiency (Max Load/ Weight)
TRUSS_1	20N	4.566 g = _____ N	_____
TRUSS_2	13.68N	7.418 g = _____ N	_____
TRUSS_3	21.6N	8.266 g = _____ N	_____
TRUSS_4	121.84N	9.130 g = _____ N	_____
TRUSS_5	151.8N	9.508 g = _____ N	_____

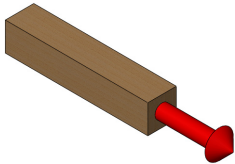
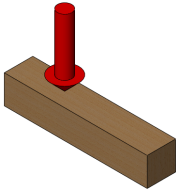
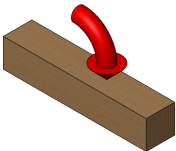
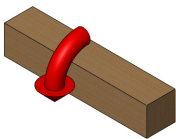
Which iteration of the structure proved to be the most efficient? _____

6 Close the part.

Click **File, Close** and click **Save** to save changes.

More to Explore

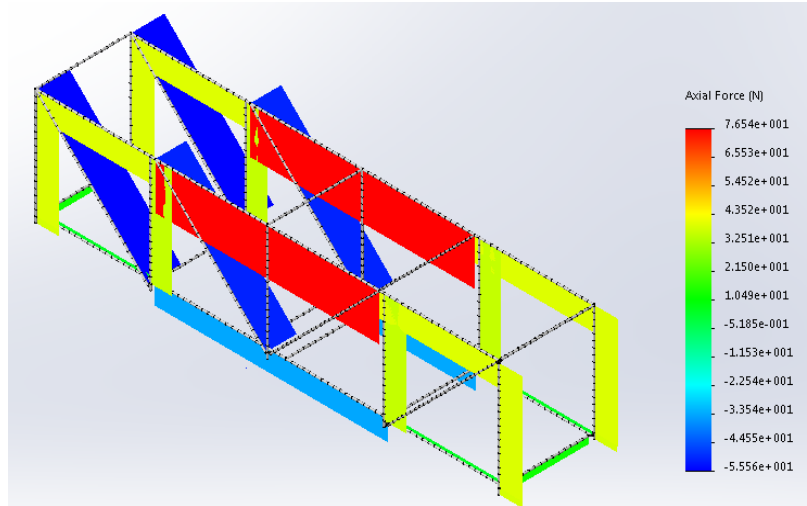
Each simulation can have multiple plots to display the results in different ways, but the beam analysis has a unique type, the **Beam Diagram**. This plot can be used to display several quantities directly on the beams. The forces and shears are displayed in Newtons (**N**), the moments and torque in Newton-Meters (**N-m**).

Beam Force Type	Force Direction
Axial Force	
Shear Force (directional)	
Moment (directional)	
Torque	

A beam diagram can be added to the results by right-clicking the Results folder and selecting **Define Beam Diagram**. One of the above types must also be selected.

Reading the Plot

As an example, look at a plot using **Axial Force**. The axial force in the angled brace members is blue, meaning that the value is between **-44N** and **-55N**. The braces are in tension, because their axial force values are negative.



Note: The axial forces in the vertical member nearest the external loads are very small because the braces absorb most of the load.

Lesson 5: Using an Assembly

Goals for This Lesson

- ❑ Open an assembly.
- ❑ Move components in the assembly.
- ❑ Check interferences between assembly components.
- ❑ Make a change to a part while in the assembly.


Testing an Assembly

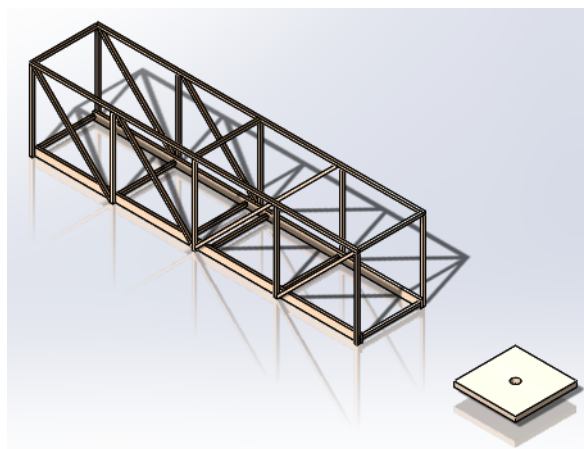
Assemblies are SOLIDWORKS files that contain multiple parts. We can use an assembly to test whether a test block, representing a vehicle, can be moved through the structure.

Testing using the Test Block

If you were to build and test this structure, it would have to meet certain criteria for length, width and height. One of the criteria would be a test to see if a wooden block of a certain size and length could fit through.

1 Open the assembly file.

Click **Open** . From the **Open** window, browse to the *Bridge Design Project\Student\Lesson 5* folder. Select *Test_Block_Assembly.sldasm*, and click **Open**. The assembly includes a copy of the previous structure and a representation of a wood block.

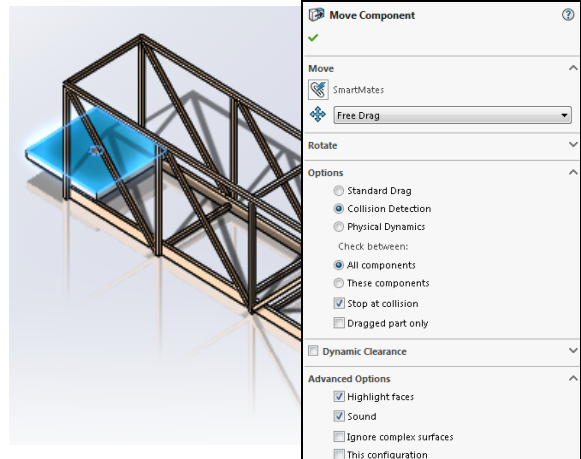


Where to Find It

- CommandManager: **Assembly** > **Move Component**  > **Move Component** 
- Menu: **Tools, Component, Move**

2 Move component.

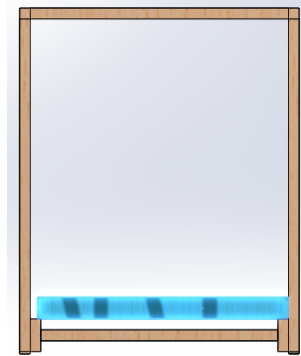
Select the *Load_Plate_75* component and click **Tools, Component, Move**. In the dialog, click **Collision Detection, All components, Stop at collision, Highlight faces** and **Sound**. Select and drag the *Load_Plate_75* through the structure. It should move smoothly through and back to the starting position outside the structure.



3 Fit.

The block fits through the structure. In fact, there is a larger clearance than is needed. To get the most efficient structure, we want to limit the width of the structure so that the block fits with a very small clearance.

Click ✓.



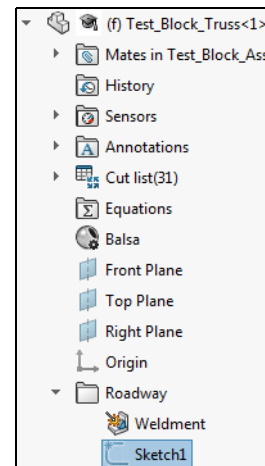
Changing the Model

Changes made to a model affect the assembly and the analysis.



4 Expand features.

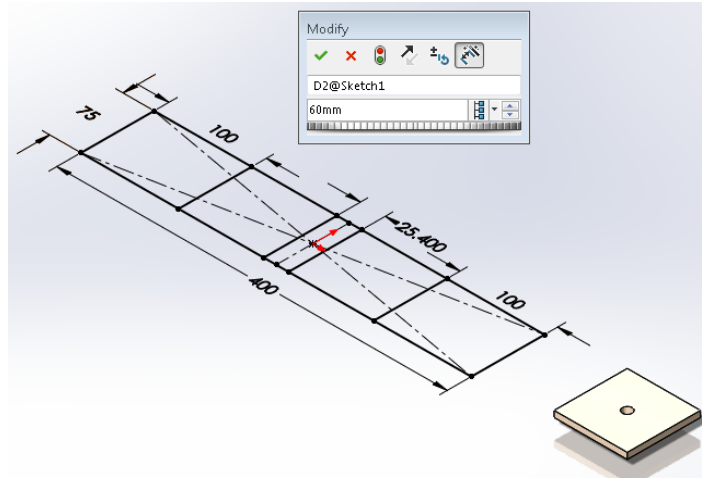
In the FeatureManager, double-click the *Test_Block_Truss* component and then the *Roadway* folder to expand them.

Double-click the *Sketch1* feature.



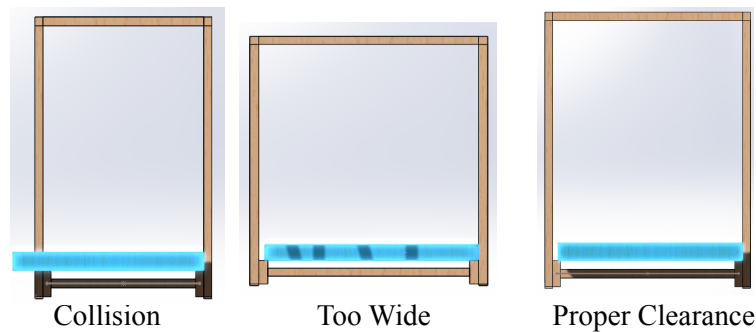
5 Change dimension.

Double-click the **75** dimension and change it to **60**. Click **Rebuild**  and . The structure part changes size.



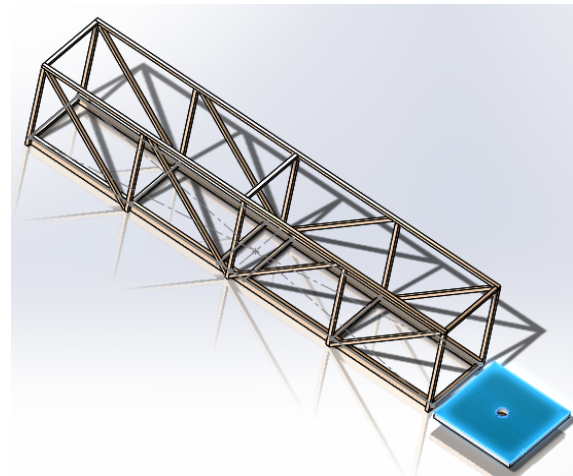
Collision Detection

Clearances are small distances between parts designed to allow them to fit together properly. If any part is too small or too large, the assembly will not fit together properly.



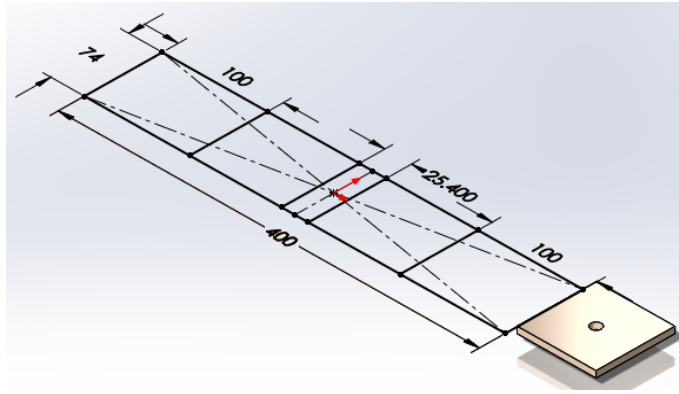
6 Move.

Using the same **Move** procedure as before (step **2**), try to move the block through the structure. It collides with the structure.



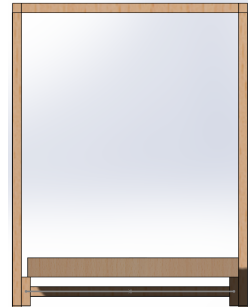
7 Increase width.

Using the same dimension change procedure as before (step 5), change the dimension to **74mm**.




8 Correct size.

This size provides a small clearance and allows the block to slide through.



9 Open the part.

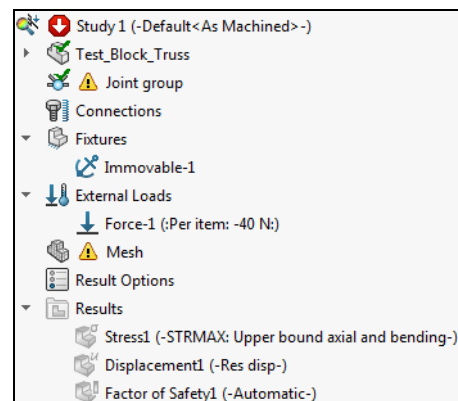
Right-click on the *Test_Block_Truss* in the FeatureManager and select **Open Part** . The structure part opens in its own window.

Updating the Analysis

The model has changed and has actually become narrower. The model change will cause several errors in the joints which in turn cause errors in the fixtures, loads and mesh.

10 Warnings and errors.

Click *Study 1*. There are warning and error markers on several features.

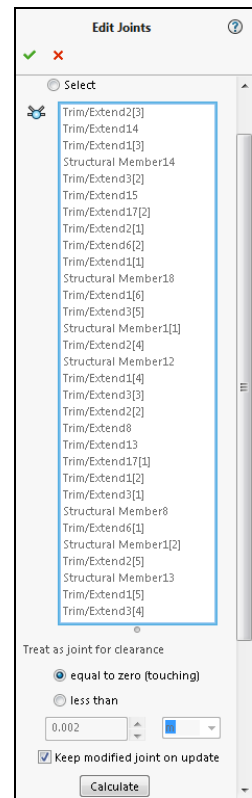


11 Joint group.

Right-click the *Joint Group* and select **Edit**.

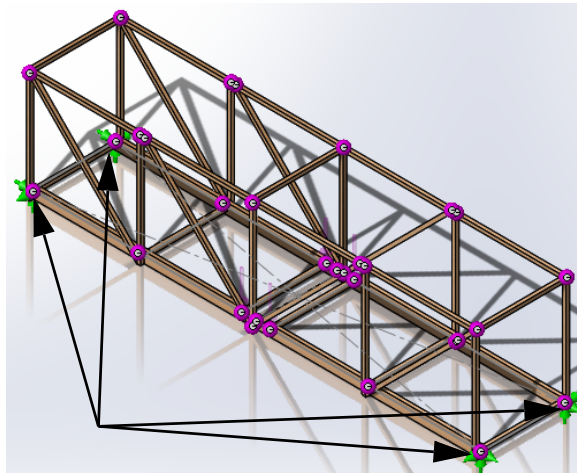
Click **Calculate** and ✓.

The message says *Joints are recalculated*. *Calculated joints may look same, but the order may be different*. *Re-definition of fixture/load/connection may be required*. Click **OK**.



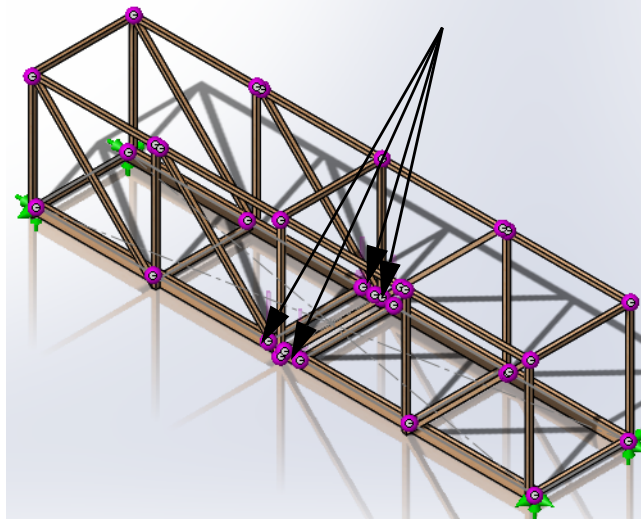
12 Fixture.

Right-click the fixture *Immovable-1* and select **Edit Definition**. Check to see that the same four (green) joints are selected and click ✓.



13 Load.

Right-click the external load *Force-1* (:Per item: -37.95 N:) and select **Edit Definition**. Check to see that the four joints as shown are selected and click ✓.



14 Mesh and run.

Right-click the *Mesh* feature and select **Mesh and Run**. The changes are insignificant. Click **File, Close** and save all changes.

Lesson 6: Making Drawings of the Structure

Goals of This Lesson

- ❑ Add a drawing view of the part.
- ❑ Create a weldment cut list table.
- ❑ Add balloons to a drawing view.

Drawings

SOLIDWORKS allows you to easily create drawings of parts and assemblies. These drawings are fully associative with the parts and assemblies they reference. If you change a dimension on the finished drawing, that change propagates back to the model. Likewise, if you change the model, the drawing updates automatically.

Drawings communicate three things about the objects they represent:

- ❑ **Shape** – *Views* communicate the shape of an object.
- ❑ **Size** – *Dimensions* communicate the size of an object.
- ❑ **Other information** – *Notes* communicate nongraphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs and so forth.

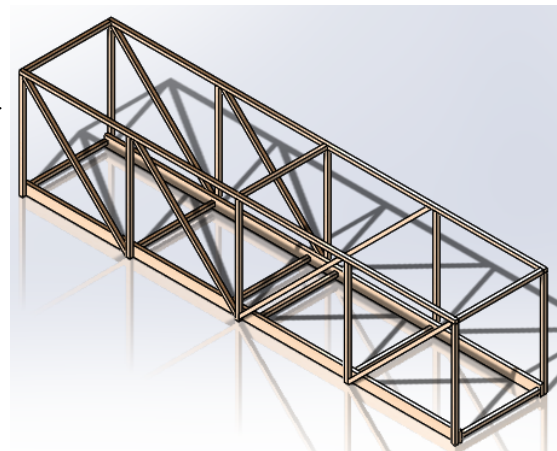
Creating a Drawing and Views

Once the model has been completed, a drawing can be made using that part. In this example, a blank drawing sheet has been associated to the part.

1 Open the part file **Drawings**.

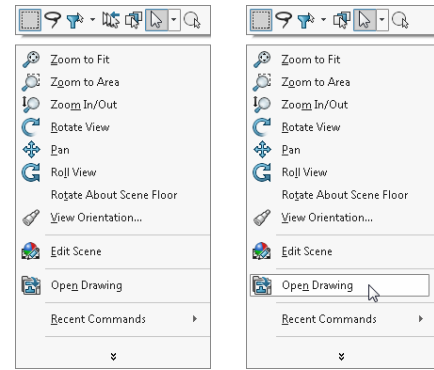
From the **Open** window, browse to the *Bridge Design Project\Student\Lesson 6* folder.

The part is a completed model of the structure.





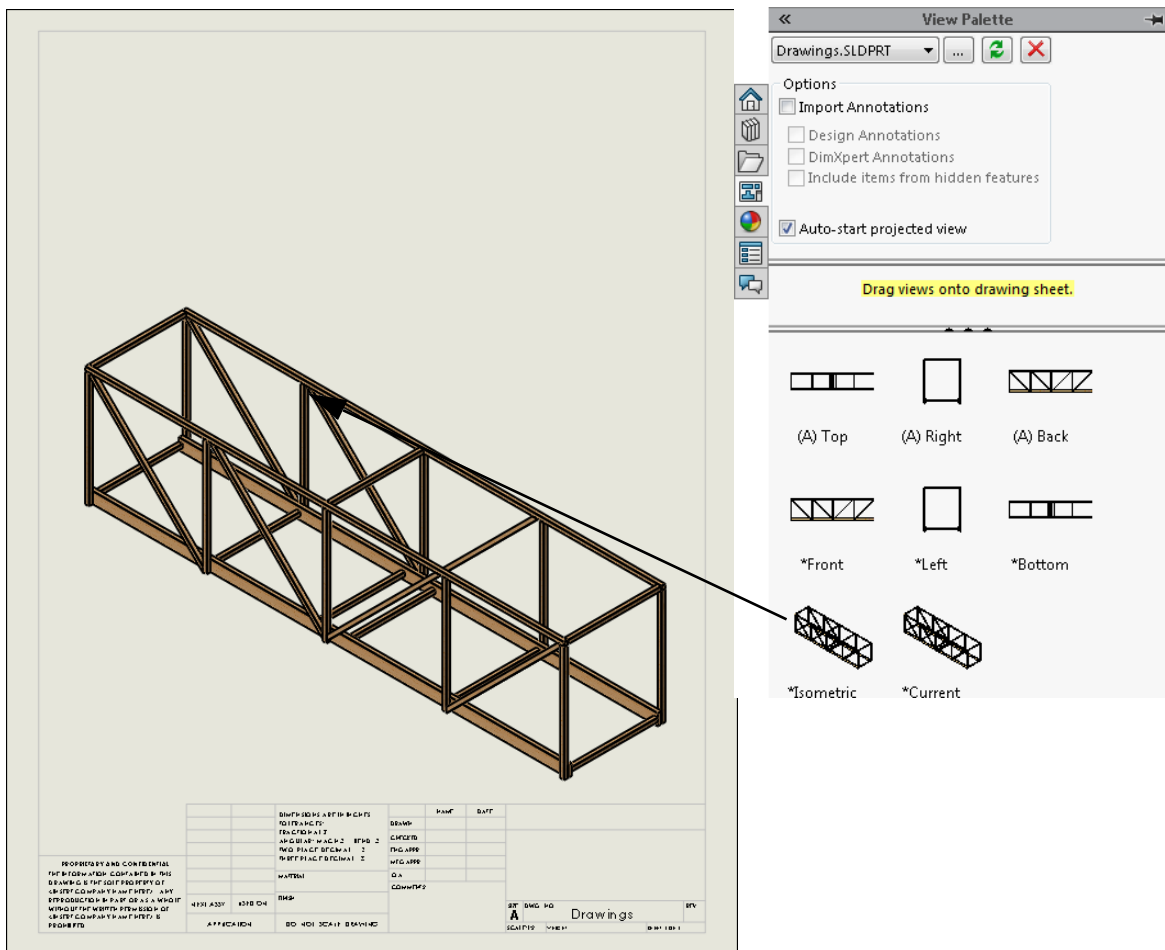
2 Open drawing file.

This part has an associated drawing file. It has no drawing views or notes, but it contains many of the setting that we need. To open it, right-click in the graphics area, then click **Open Drawing**.



3 Expand View Palette.

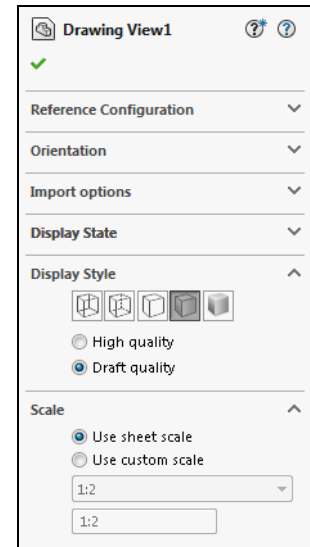
Click on the **View Palette**  to expand it. The view palette contains views of the current part. Click **Refresh**  and clear **Import Annotations**. Drag and drop an **Isometric* view from the View Palette to the drawing sheet.



4 Drawing view properties.

Select **Shaded With Edges**  from **Display Style**. Click **Use sheet scale**.

Click  to complete the view.



What is a Weldment Cut List Table?

The **Weldment Cut List Table** is a listing of the members or beams in the part. They are sorted into groups by length, and include an item number, quantity, description and length. All this information is extracted from the part.

Where to Find It


- CommandManager: **Annotation > Tables**  > **Weldment Cut List** 
- Menu: **Insert, Tables, Weldment Cut List**

5 Weldment Cut List.

Click **Insert, Tables, Weldment Cut List** and select the drawing view.

Select the file *Bridge_Weldments.sldwldtbt* as the **Table Template**.

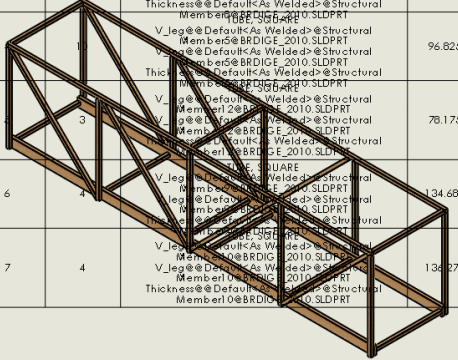
The file is stored in the folder *Bridge Design Project\Student\Lesson 6*, the same folder as the part.

Click  and move the cursor onto the drawing.



6 Place table.

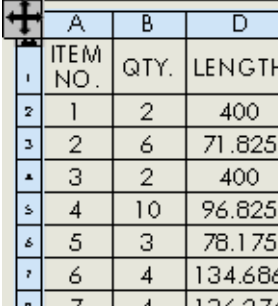
Move to the upper left corner of the drawing and click to place the table.



ITEM NO.	QTY.	DESCRIPTION	LENGTH
1	2	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member1@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member1@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member1@BRIDGE_2010.SLDPR	400
2	6	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member3@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member3@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member3@BRIDGE_2010.SLDPR	71.825
3	2	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member5@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member5@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member5@BRIDGE_2010.SLDPR	400
4	10	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member5@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member5@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member5@BRIDGE_2010.SLDPR	96.825
5	3	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member2@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member2@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member2@BRIDGE_2010.SLDPR	78.175
6	4	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member6@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member6@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member6@BRIDGE_2010.SLDPR	134.686
7	4	TUBE, SQUARE V_leg@@DefaultAs Welded@Structural Member0@BRIDGE_2010.SLDPR V_leg@@DefaultAs Welded@Structural Member0@BRIDGE_2010.SLDPR Thickness@@DefaultAs Welded@Structural Member0@BRIDGE_2010.SLDPR	136.274

7 Resize columns.

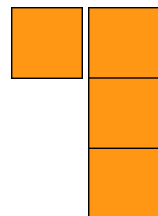
Drag the column and row borders of the table to resize them.
Each column and row border can be resized.



	A	B	D
ITEM NO.	QTY.	LENGTH	
1	2	400	
2	6	71.825	
3	2	400	
4	10	96.825	
5	3	78.175	
6	4	134.686	
7	4	136.274	

Why are there two Items of the Same Length?

A different beam than what corresponds to the top member of the walls is used to represent the stack of three beams that makes up the bottom of the bridge. So although items 1 and 3 are the same length, they are considered different beams.



Tip: In the building section the actual sizes of the individual beams will be listed.

Balloons

Balloons label the members in the part and relate them to the cut list numbers on the weldment cut list.

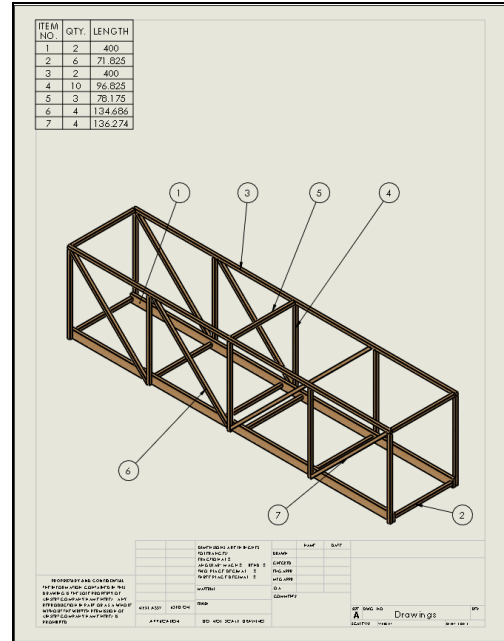
Where to Find It

- CommandManager: **Annotation > Balloon** ⓘ
- Menu: **Insert, Annotation, Balloon**

8 Balloons.

Click **Balloon**. Click on the member and then click to place the text. Repeat the process to add some balloons.

Click ✓.

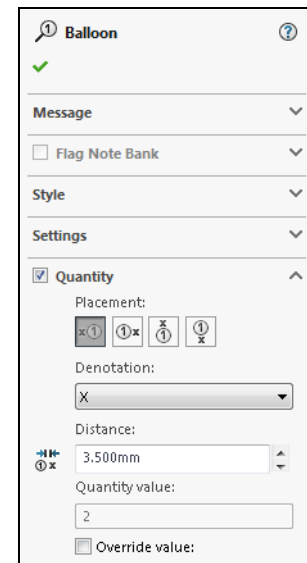
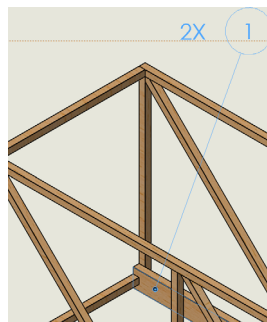


Note: You can move balloons by dragging the text.

9 Balloon quantities.

The quantity of items in a balloon can be set. Click a balloon and click **Quantity**.

Select a **Placement** and click ✓.



10 Close the drawing and part.

Click **File, Close** and save all the files.

Lesson 7: Reports and SOLIDWORKS eDrawings

Goals of This Lesson

- Create an HTML report.
- Load the SOLIDWORKS eDrawings add-in.
- Describe a SOLIDWORKS eDrawings file.
- Create SOLIDWORKS eDrawings of SOLIDWORKS Simulation data.
- Save the SOLIDWORKS eDrawings file as an HTML file.

Reports and SOLIDWORKS eDrawings

There are many ways to generate data from the structural analysis. A **Report** is useful for printing and viewing text and static data. Use **SOLIDWORKS eDrawings** to view, share and manipulate the analysis result plots without having to open the part.

1 Open the part file *Reports&eDrawings*.


From the **Open** window, browse to the *Bridge Design Project\Student\Lesson 7* folder.

Open the part file *Reports&eDrawings*. Click *Static 1* and run the analysis.

Creating a Report

Using SOLIDWORKS Simulation, you can create a printable report that captures all the important data.

Where to Find It

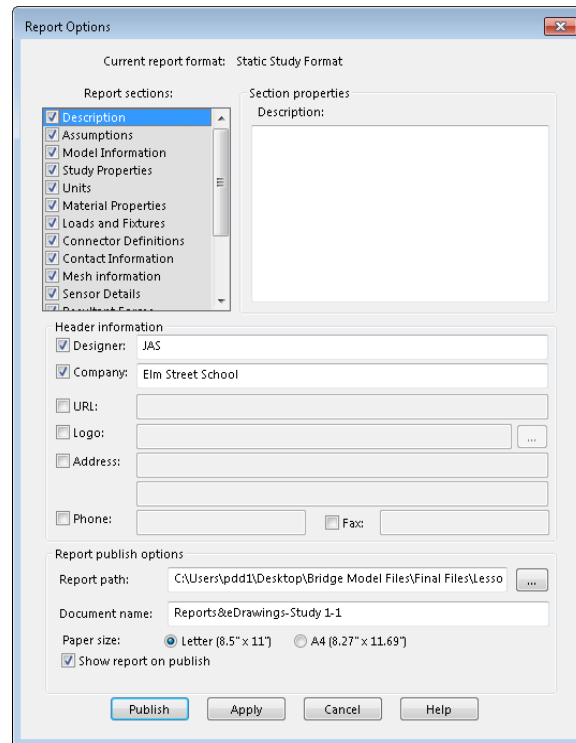
- CommandManager: **Simulation > Report** 
- Menu: **Simulation, Report**

2 Report.

Click **Simulation, Report**.

3 Dialog.

Click **Designer** and **Company**. In **Designer**, add your initials. In **Company**, add the name of your school.

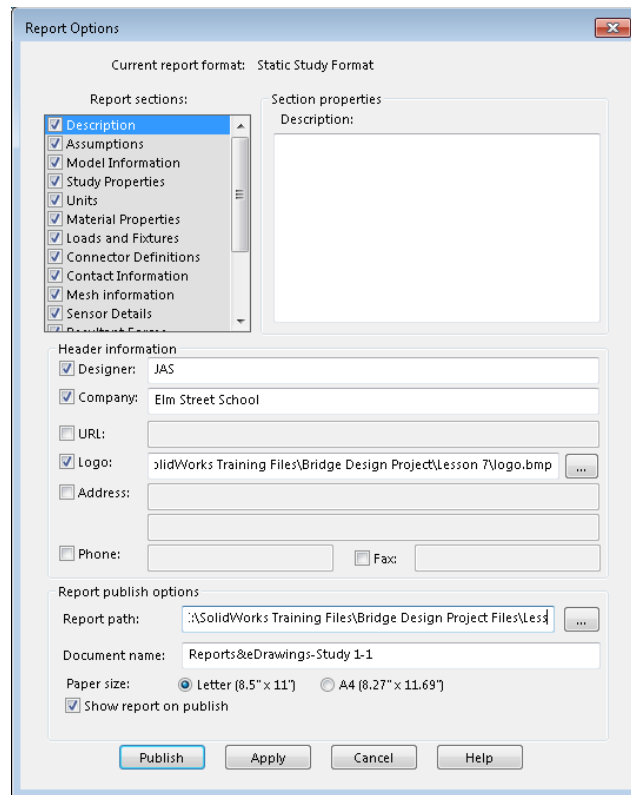


4 Logo.

Click **Logo**. Click the **Browse** button and select the file *logo.bmp* from the *Lesson 7* folder.

5 Description.


Click **Description** and type *This is the structural analysis of a balsa wood truss* into the comments section and click **Publish**.



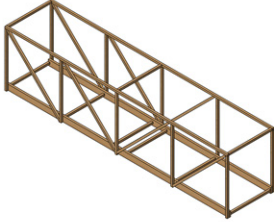
Tip: The **Report path** can be set to receive the report and associated data.

6 Report.

The report, complete with data and images, appears in a new window as soon as it is generated. Close the window or print the report.



Elm Street School



Description
No Data

Simulation of Reports&eDrawings

Date: Thursday, September 03, 2015
Designer: JAS
Study name: Study 1
Analysis type: Static

Table of Contents

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Units	9
Material Properties	10
Loads and Fixtures.....	12
Connector Definitions	12
Contact Information.....	12
Mesh information	13
Sensor Details.....	14
Resultant Forces	15

The results are placed in the same folder as the part by default. They can be printed and opened independently of SOLIDWORKS or SOLIDWORKS Simulation.

Note:Do not close the part.

SOLIDWORKS eDrawings for Sharing Information

eDrawings® is an email-enabled communications tool designed to dramatically improve sharing and interpreting 2D mechanical drawings. eDrawings are small enough to email, are self-viewing and significantly easier to understand than 2D paper drawings.

Advantages of eDrawings

- ❑ Recipients do not need to have the SOLIDWORKS application to view the file.
- ❑ View parts, assemblies and drawings outside of SOLIDWORKS.
- ❑ Files are compact enough to email.
- ❑ Creating an eDrawing is quick and simple.
- ❑ Publish an eDrawing from any SOLIDWORKS file.
- ❑ You can create eDrawings from other CAD systems, too.

Viewing eDrawings

You can view eDrawings in a very dynamic and interactive way. Unlike static 2D drawings, eDrawings can be animated and viewed dynamically from all angles. The ability to interact with eDrawings easily — in an interactive manner — makes it a very effective design collaboration tool.

eDrawings Professional gives you the ability to perform mark-up and annotation of eDrawings which further enhances design collaboration.

Viewing eDrawing Animations

Animation automatically demonstrates how drawing views relate to each other and to the physical design. With the click of a button, eDrawings “animates” all views contained in each sheet of your drawing, morphing from one view to another.

The animation continuously shows you the eDrawing from different views. This dynamic motion is similar to turning a part around in your hand as you inspect it.

Creating a SOLIDWORKS eDrawing

eDrawings are an easy way to share data, especially the image data that is generated by SOLIDWORKS Simulation.

7 Plot.

Double-click the *Displacement1 (-Res disp-)* plot to activate it. This is the plot that will be saved into the eDrawing.

8 Save.

Click **Simulation, Result Tools, Save As**. Save the data using the *eDrawings Files (*.analysis.eprt)* file type. Click **Save**.

The default name is of the form:

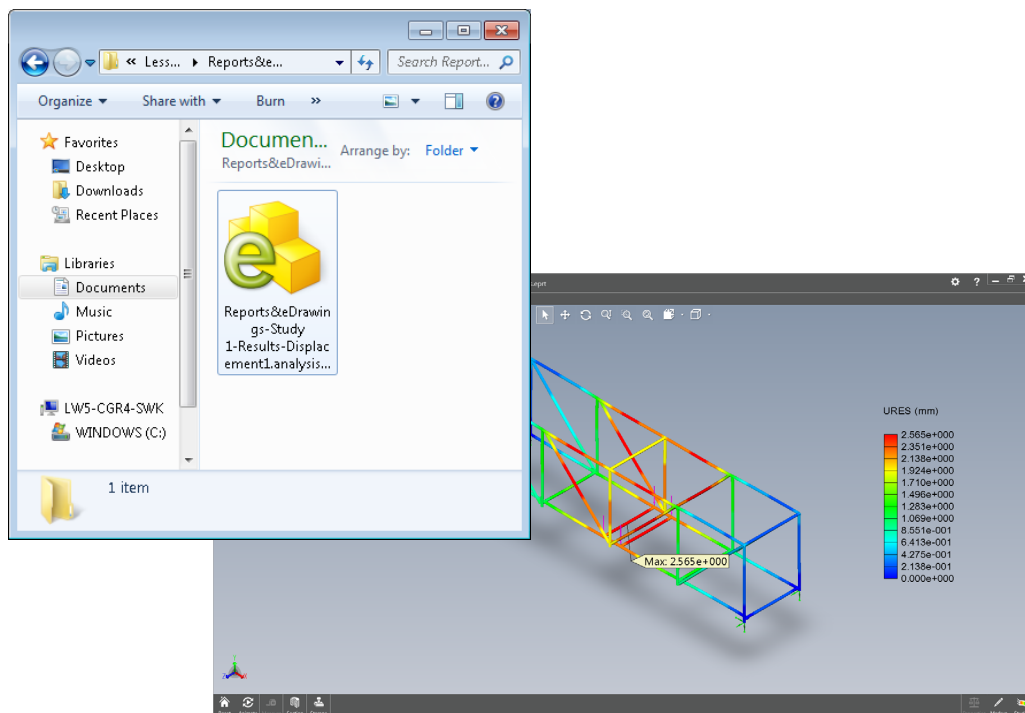
part name-study name-results-plot type. In this case it is

Reports&eDrawings-Static 1-Results-Displacement1.analysis.eprt

It is stored in the same folder that was created by the report.

9 Open the eDrawing.





Double-click the eDrawings file in the folder. The eDrawings window appears.

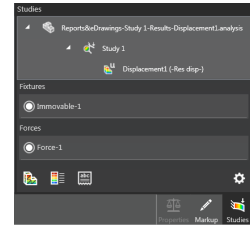


Note: If eDrawings has not been used before, it may ask permission to load.

The eDrawings User Interface.

10 Settings.


Click the **Studies** tab , and the options **Show Mesh** , **Show Legend** , and **Show Title** .

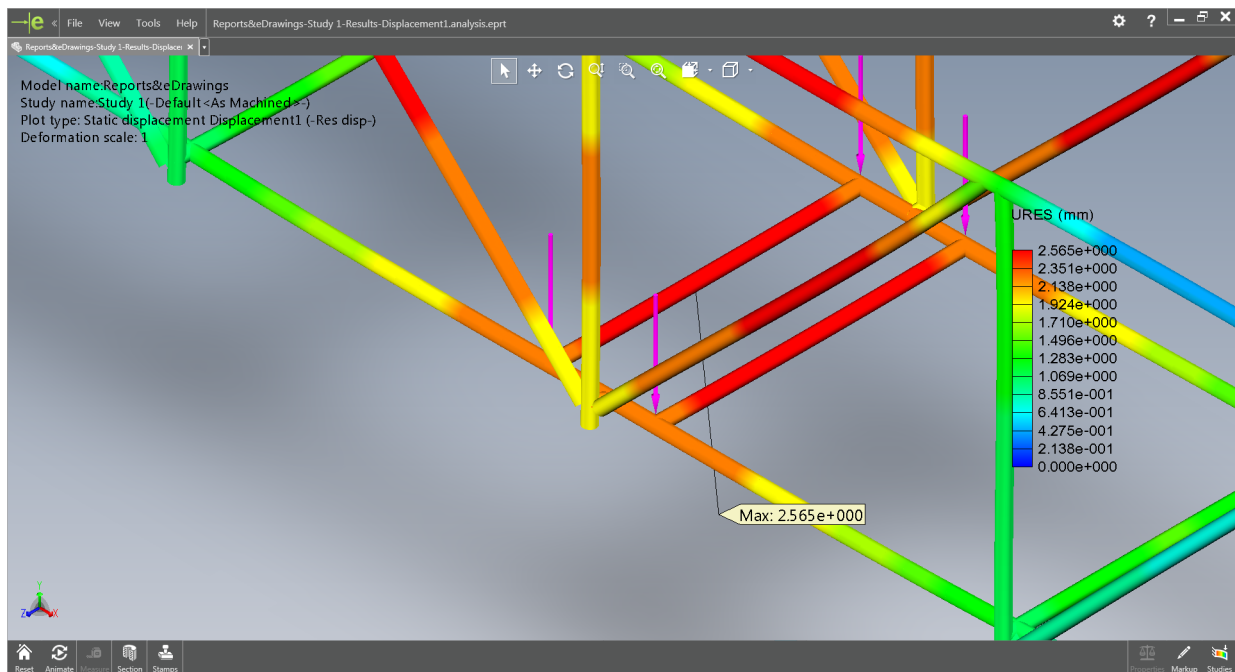


eDrawings Functions

You can animate, zoom, scroll, and rotate the image using various tools.


11 Move components.

Click **Zoom to Area**  and drag a window around the center section of the structure.




Playing an eDrawings Animation


12 Play the animation.

Click **Play** . This starts a continuous forward play loop showing each view in sequence. The animation sequence is system controlled; you cannot set the sequence.


13 Stop the animation.

Click **Stop**  to stop the animation.

14 Reset the view.

To return the animation to the start of the sequence, click **Reset** .

Saving eDrawings

Click **File, Save** , or press **Ctrl+S** to save the file currently open in the eDrawings Viewer. You can save files as the following file types:

- ☐ eDrawings files (*.eprt, *.easm, or *.edrw)
- ☐ eDrawings Zip files
 - The eDrawings Zip file contains the eDrawings Viewer and the eDrawings file. You can unzip the eDrawings Zip file and run the eDrawings executable file to extract the embedded eDrawings Viewer and open the embedded eDrawings file.
- ☐ eDrawings HTML files
- ☐ eDrawings executable files
 - You can save files as self-extracting eDrawings executable (*.exe) files that contain the eDrawings Viewer and the eDrawings file. Some email programs, anti-virus programs, and internet security settings prevent receiving email with executable files as attachments.
- ☐ BMP, TIFF, JPEG, PNG or GIF image files
 - You can save all file types that you can open in the eDrawings Viewer as BMP (*.bmp), TIFF (*.tif), JPEG (*.jpg), PNG (*.png), or GIF (*.gif) files.

Save the eDrawing

15 Save the eDrawing.

Click **File, Save As**. For **Save as type**: click *eDrawing HTML Files (*.htm)* to save the eDrawing as an HTML file. This file can be viewed in a web browser. Click **Save**.

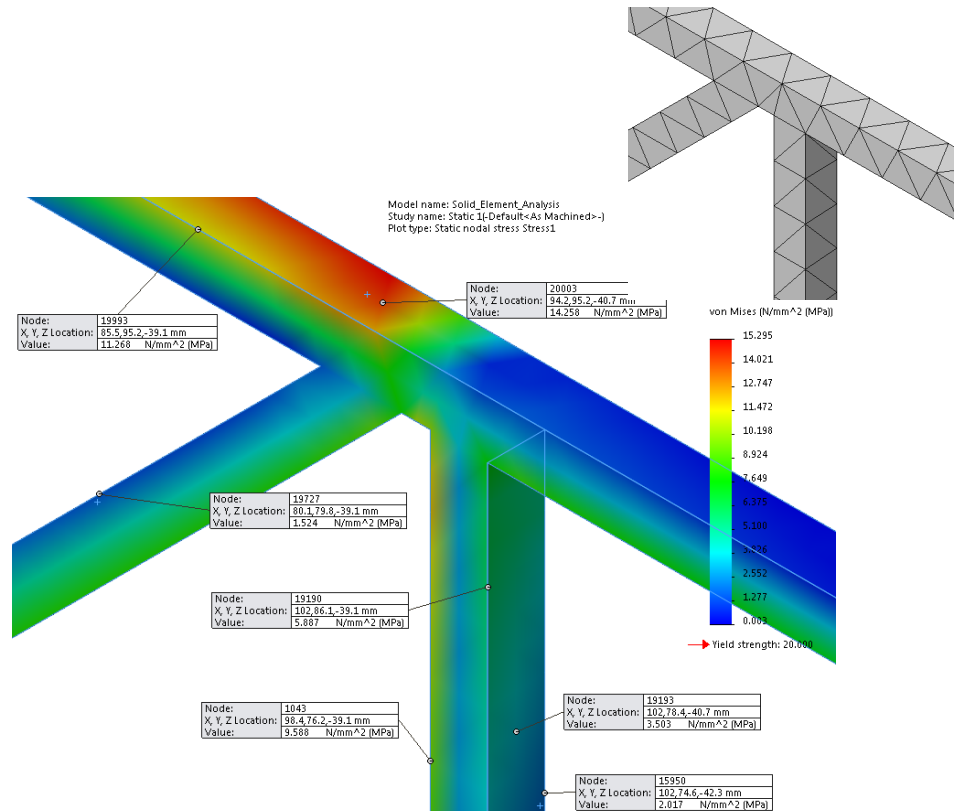
Save the file in the *Reports&eDrawings-Static 1* folder.

16 Close all files.

More to Explore

A beam mesh is very quick and efficient method to analyze a beam model and works well to reflect the overall state of the structure. However, beam elements cannot analyze what happens across the thickness of the beam because they only generate results at the nodes which lie at the centerline of the beam.

Using a solid mesh creates elements and nodes across the thickness of the model. This provides multiple nodes across the thickness and results across the thickness.



To explore, open the part file *Solid_Element_Analysis* and click **Simulation, Run**. The results plots are arranged the same way as those in a beam analysis.

Lesson 8: Building and Testing the Structure

Goals of This Lesson

- ☐ Open and print informational PDF files.
- ☐ Cut the beams to the proper lengths.
- ☐ Assemble the beams to create the truss.
- ☐ Test the truss by applying a load.

Building the Structure

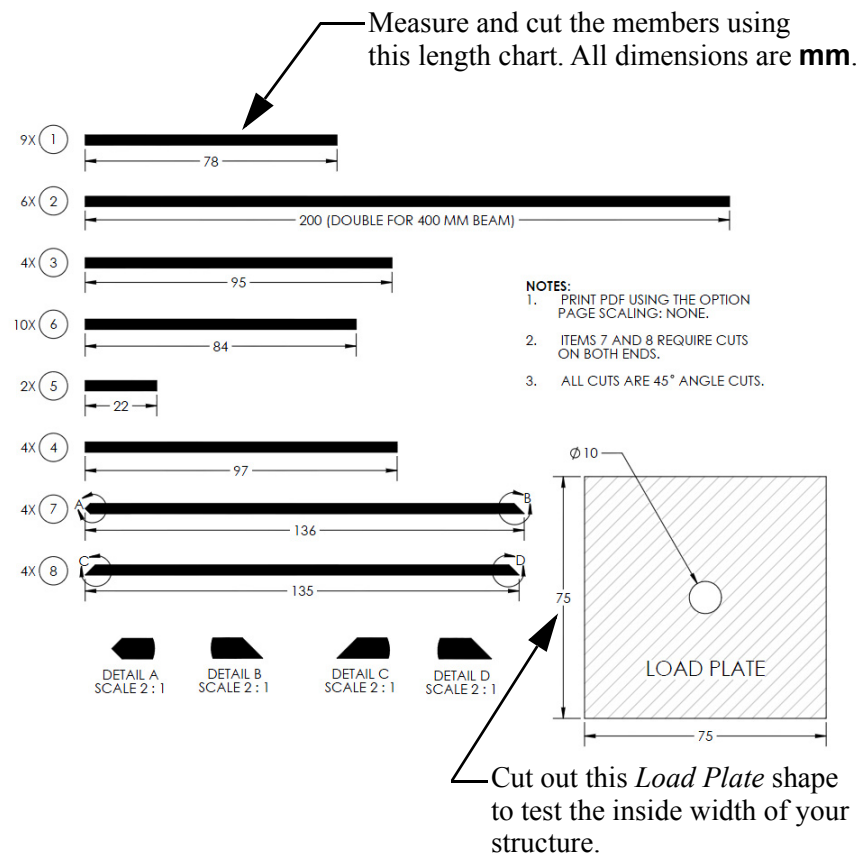
If your class chooses to build and test the structure, you will need 1/8" x 1/8" balsa wood sticks. Lengths of at least 24" or 400mm are required. Glue and a knife to cut the sticks are also required.

Cutting to Length

There are **43** members of **8** different lengths required to build this structure. There are 2 PDF files that can be helpful during the construction process. They are located in the same folder as these instructions.

1 Open and print *Measuring Chart.PDF* file.

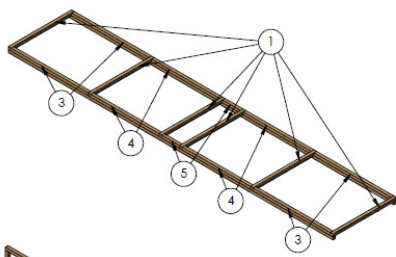
Browse to the *Lesson 8* folder, open the *Measuring Chart.PDF* file and print it using the precautions stated in the PDF and in the note below.



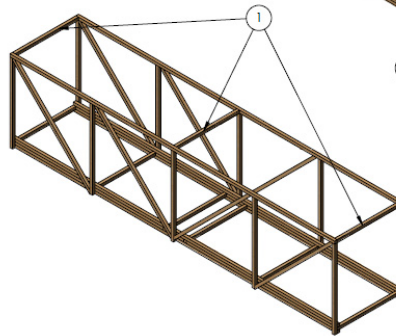
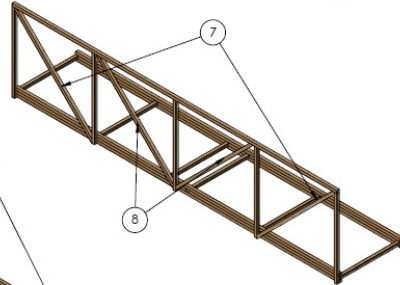
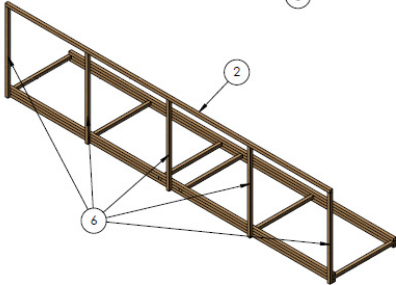
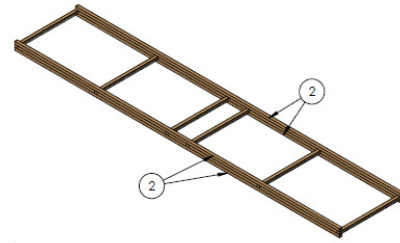
Note: Print this PDF using the **Page Scaling: None** option to get accurate values!

2 Open and print *Construction Guide.PDF* file.

Open the *Construction Guide.PDF* file and print it.

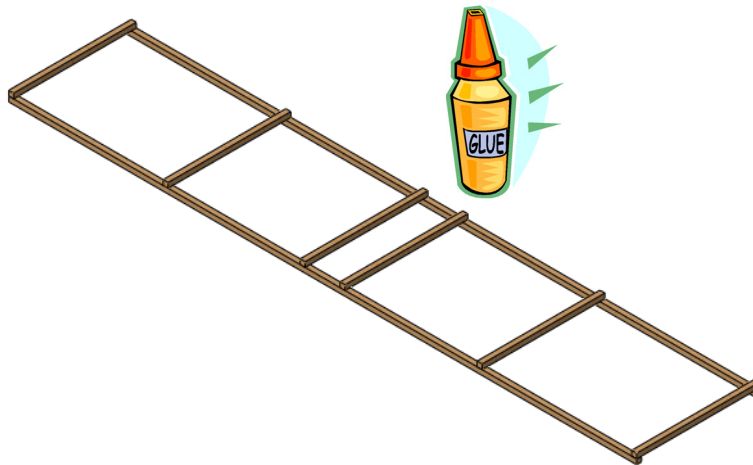
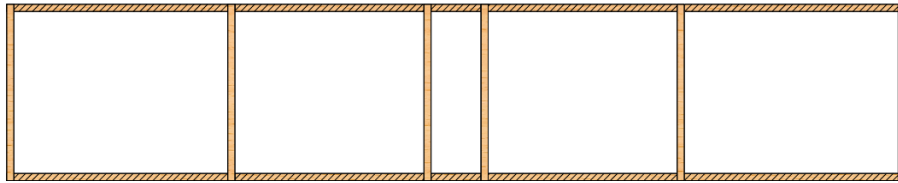


ITEM NO.	QTY.	LENGTH
1	9	78
2	6	400
3	4	95
4	4	84
5	2	22
6	10	97
7	4	136
8	4	135



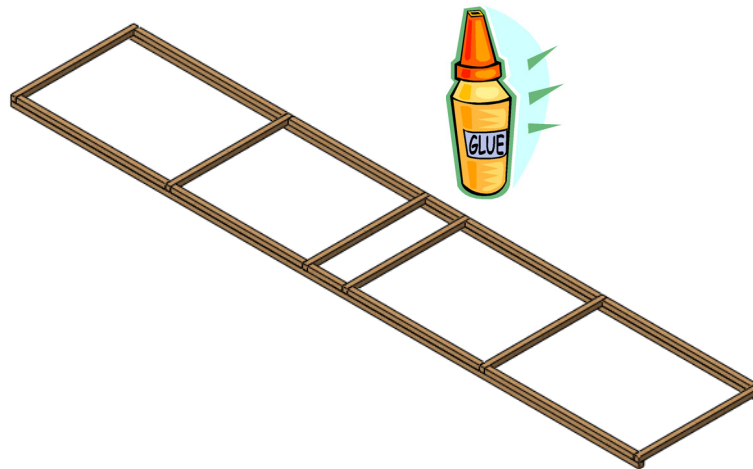
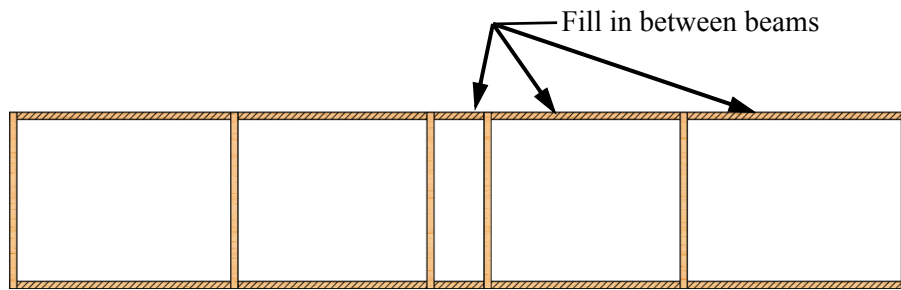
3 Lower frame.

Glue the end cross beams to the long beams. Do not glue the interior cross beams yet.



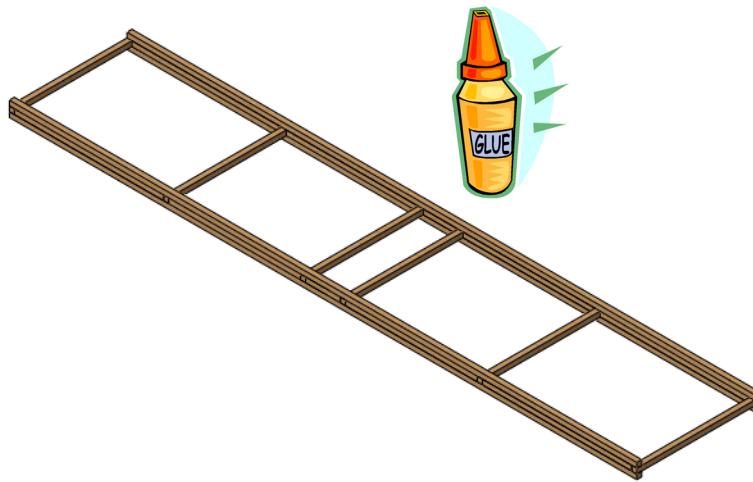
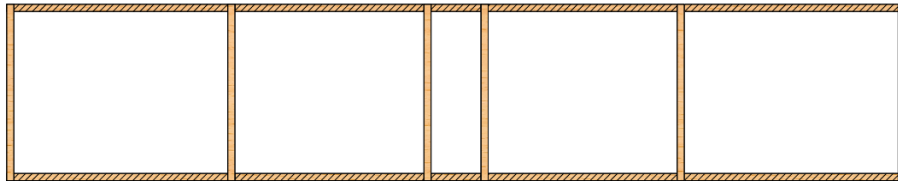
4 Fill in beams.

Fill in between the beams (cross hatched area) by cutting beams to fit and placing them in position. Glue all beams.



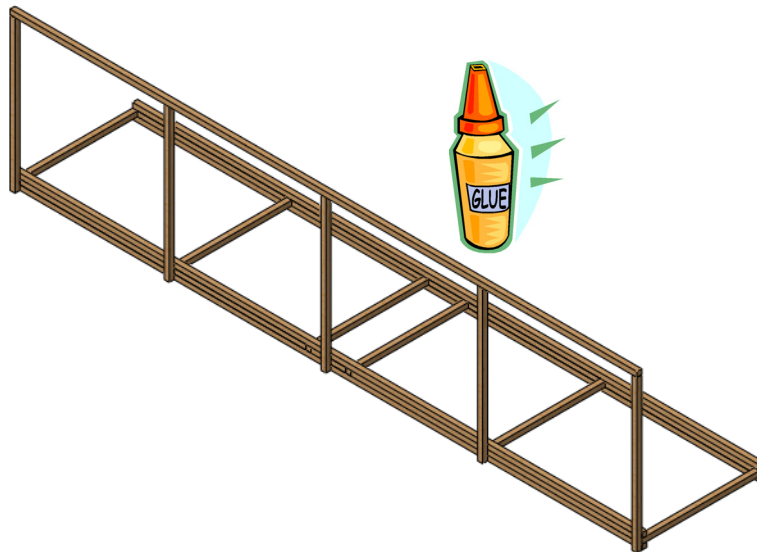
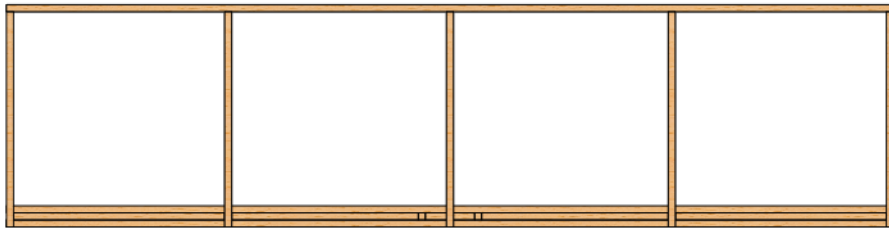
5 Triple outer rails.

Glue the long beams over the fill in beams as shown. Glue all beams.



6 Side walls.

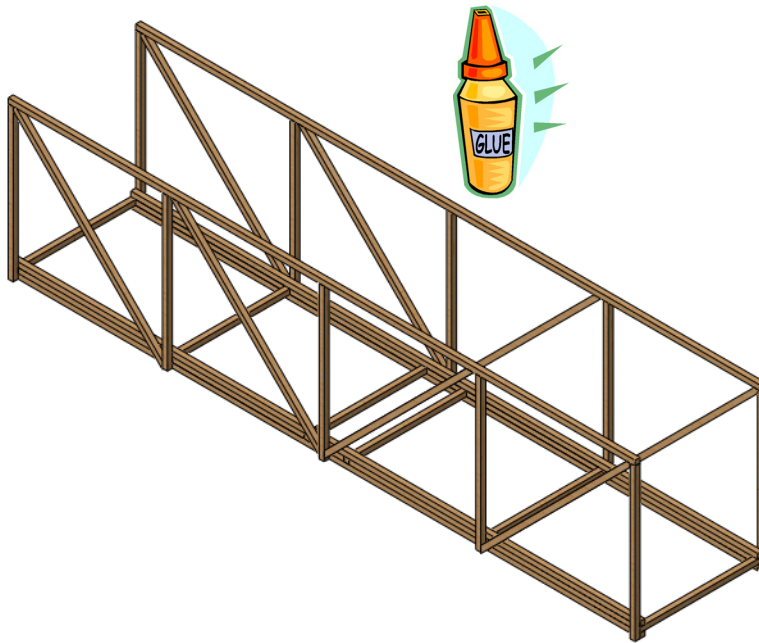
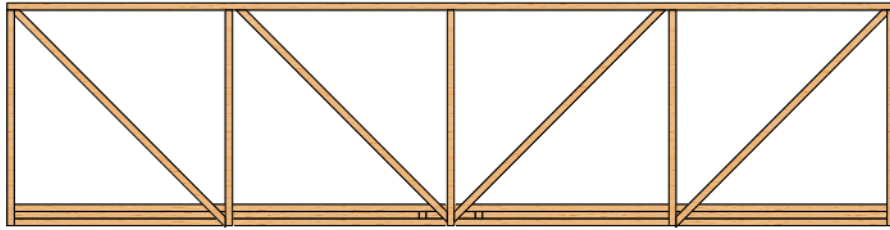
Glue all beams.



Tip: You might want to build a side and then cross brace it (step **7** on page 88) before starting on the opposite side.

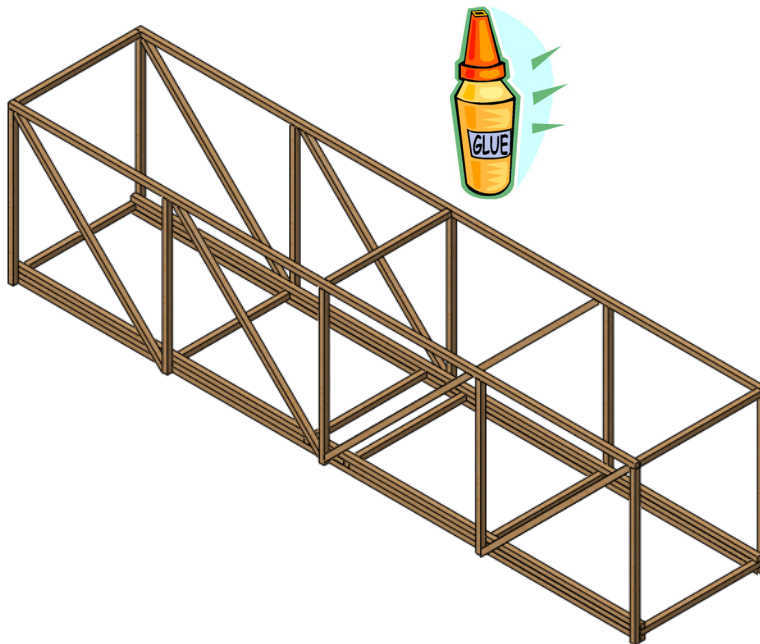
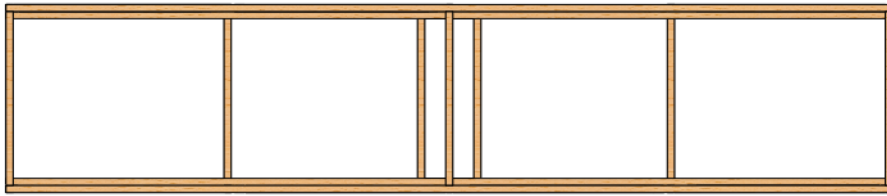
7 Cross bracing.

The brace members must all be cut (45 degrees) to fit within the existing framing. Glue all beams.



8 Top cross supports.

Glue all beams.



Testing the Structure

The structure can be tested by placing it across a gap and applying a load at the center of the bridge. See the following details for more information.

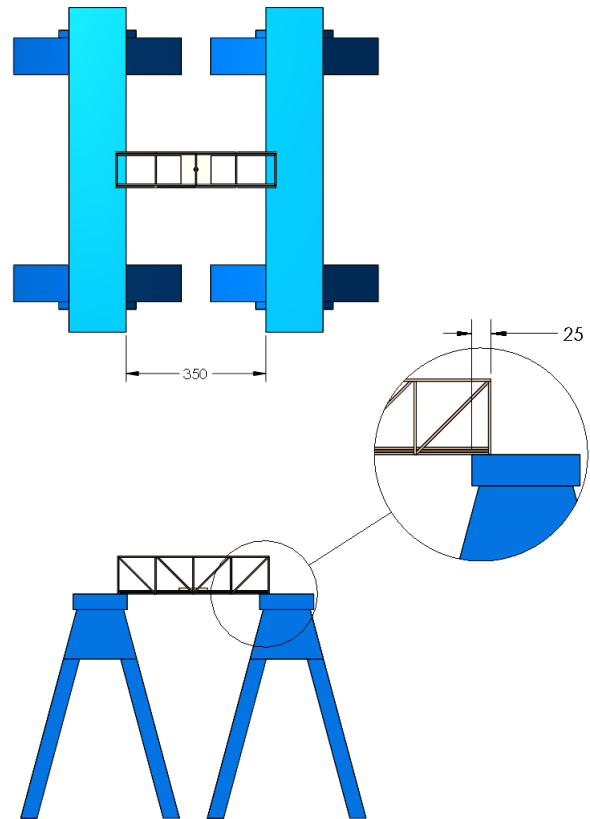
Creating the Span

One way to create a span is to place two sawhorses a set distance apart as shown below. Placing the model so that it overlaps each sawhorse the same amount simulates the environment of the analysis.

Details

Use two surfaces that are strong and equal height (sawhorses or tables work well) to create the span of **350mm** that is required. Each end of the structure should overlap the surface **25mm**.

Tip: Make sure that the tables or sawhorses are strong enough to be able to take the load without bending.



Applying the Load

In order to measure the strength of a structure, it should be loaded as it was modeled.

Using Common Objects with Known Weights

Many common objects can be used to apply the load. Food cans come in different sizes; they can be weighed and used. Coins can also be used to apply a load in very small increments. Let's take a penny as an example.

1 Penny applies about **0.0245N** of force to the structure. That isn't very much, not close to the total load that we would like to test. Does anyone really want to count hundreds or thousands of pennies for every test?

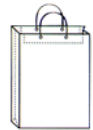
Coins can be rolled in large quantities for deposit at a bank. Pennies are rolled into groups of 50. Do some calculations for large numbers of pennies, rolls and cost.



Pennies	Load (N)	Penny Rolls	Cost (\$)
50	$50 \times 0.0245 = 1.225$	1	\$0.50
100	$___ \times 0.0245 = ______$	$______$	$______$
500	$___ \times 0.0245 = ______$	$______$	$______$
1000	$___ \times 0.0245 = ______$	$______$	$______$
5000	$___ \times 0.0245 = ______$	$______$	$______$

Holding the Load

Suspend a rope handled shopping bag or something stronger from the Load Plate by feeding the rope through the hole in the plate and pinning it with a nail or pen. Add the load by filling the bag, slowly, with the weights that you have chosen.



Lesson 9: Weldment Profiles and Structural Members

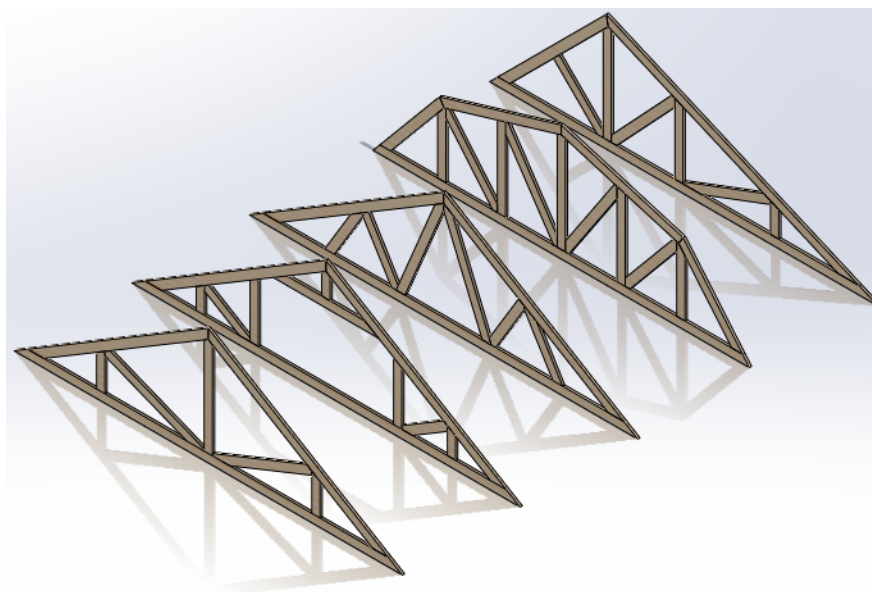
Goals of This Lesson

- ❑ Create a new weldment profile.
- ❑ Create a weldment sketch.
- ❑ Add structural members to a weldment sketch.

Creating Weldment Profiles and Structural Members

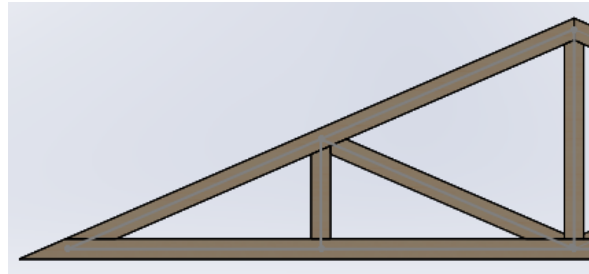
The parts that were analyzed in the previous lessons are considered to be weldment parts. This is a special type of part that is used to model a group of structural members that are connected together.

In this lesson you will learn how to create this type of part by modeling pre-fabricated roof trusses.



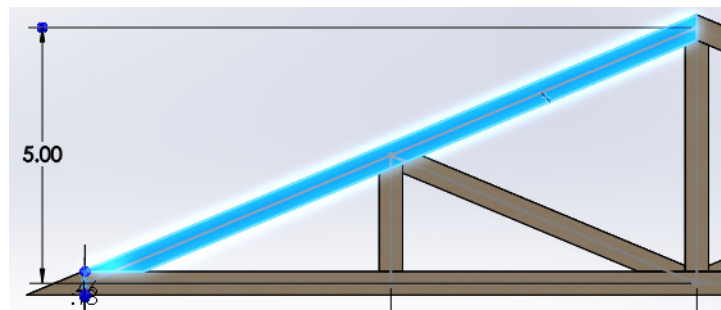
What is a Weldment?

A **Weldment** is a part that is made up of multiple solid bodies, known as structural members, connected together to form a single structure.



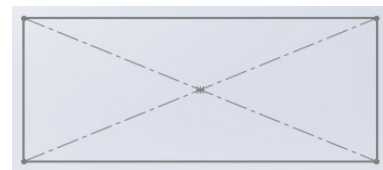
What are Structural Members?

The individual solid bodies represent steel beams or lumber and are called **Structural Members**. The position and length of the structural members are based on cross-sections, or weldment profiles, that are applied to the lines of a sketch.



What are Weldment Profiles?

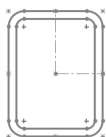
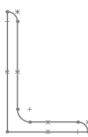
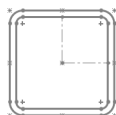
Weldment Profiles are library feature sketches that represent standard structural cross-section shapes like tubes, I-beams, angles, pipes, channels, and lumber. The profile is located by the origin position.






Existing Weldment Profiles

The existing *weldment profiles* folder is found in *C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english\weldment profiles*.

The sub-folders *ansi inch* and *iso* contain multiple sizes of these common shapes.

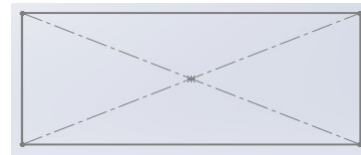
Default Weldment Profile Shapes		
<i>rectangular tube</i> 	<i>angle iron</i> 	<i>square tube</i> 

Default Weldment Profile Shapes		
<i>c channel</i>	<i>s section</i>	<i>pipe</i>
		

Warning for Paths

The full path used in this lesson is based upon the default installation of SOLIDWORKS onto the 'C' drive. If SOLIDWORKS has not been installed using the default settings, the path will require some adjustment.

Note: None of the existing weldment profiles resemble the cross-section of the balsa wood sticks or their much larger relation, 2x4 lumber. This means that a new weldment profile must be created.



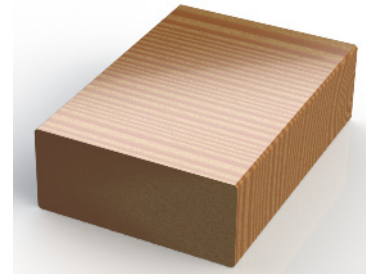
Creating a New Weldment Profile

A weldment profile is a 2D sketch that is saved as a specific file type under a specific folder.

In this example, the sketch will be a rectangle with center placement.

The file type is a library feature. The folder is the *weldment profiles* folder.

The first step is create a new folder in the proper place to hold the new file.



The Weldment Profile Folder

All of the weldment profiles are set to be found under a specific folder, the *weldment profiles* folder. You can add a new folder to the default set using **New folder**. New profiles can be placed within the new folder.

1 Folder.

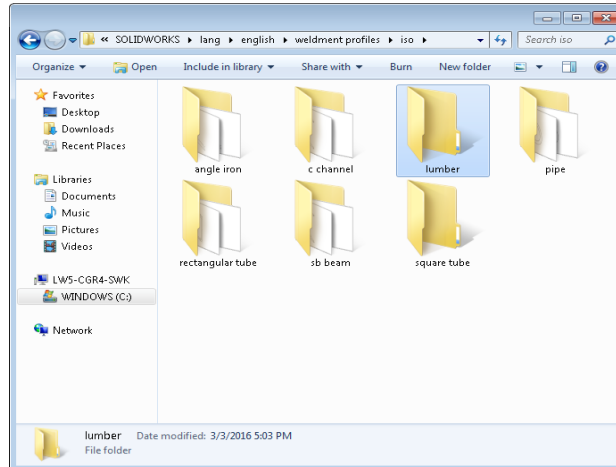
Click **Start, Computer** and browse to the folder *C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english\weldment profiles\ansi inch*.

See Warning for Paths on page 95 for more information.

2 New folder.

Click **New folder** and rename the folder to *lumber*.

2 b



Creating a New Part

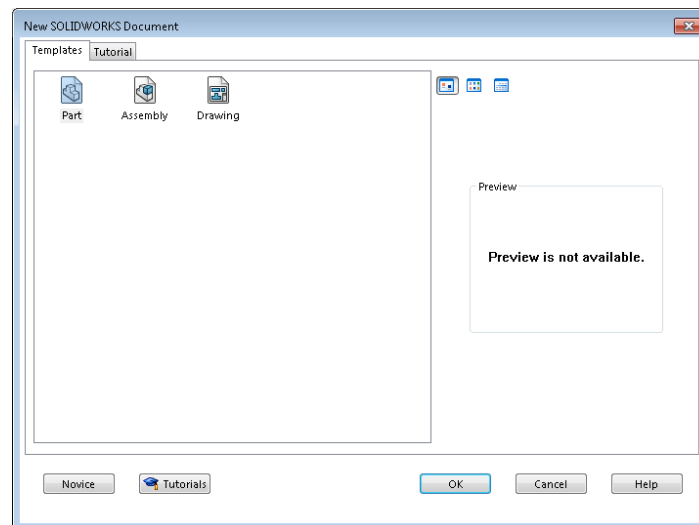
Sketches are created within parts, so the first step is to create a new part.

Where to Find It

- Menu Bar: **New**
- Menu: **File, New**
- Keyboard Shortcut: **Ctrl+N**

3 New part.

Click **New**. Click *Part* and **OK**.




Changing the Unit System

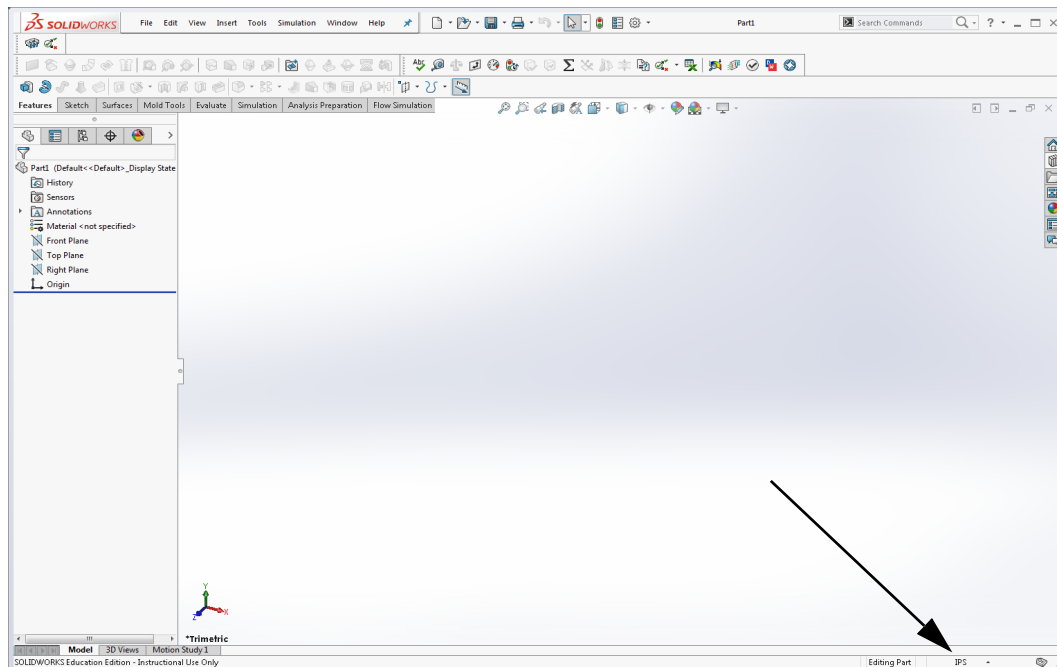
The **Unit System** is a combination of length, mass, and time units used for calculations by SOLIDWORKS. Your default unit system can be MMGS, IPS, or some other common grouping. In this example, we want the **IPS** unit system, which includes: **Inches**, **Pounds**, and **Seconds**.

Other types are:

- ❑ **MKS** (Meter, Kilogram, Second)
- ❑ **CGS** (CM, Gram, Second)
- ❑ **MMGS** (MM, Gram, Second)

4 Unit system.

Select **IPS (inch, pound, second)** from the **Unit System** menu . This changes the units to inches.



Note: The menu is located on the bottom of the screen to the right of the message *Editing Part* as shown.


Creating a New Sketch

Sketches are used in SOLIDWORKS to create 2D geometry like lines, arcs, circles, and rectangles. They are critical to the creation of geometry in SOLIDWORKS.

Where to Find It

- CommandManager: **Sketch > Sketch** 
- Menu: **Insert, Sketch**
- Shortcut Menu: Right-click a plane and click **Sketch** 



5 New sketch.

Click *Front Plane* and click **Sketch** .



Sketching a Rectangle

A rectangle is a piece of sketch geometry that is comprised of four lines. The **Center Rectangle** version is placed by the center and has two horizontal and two vertical lines.

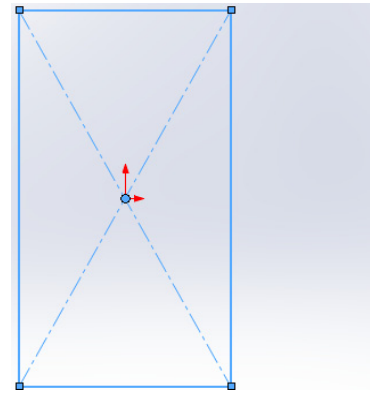
Where to Find It

- CommandManager: **Sketch > Rectangle**  > **Center Rectangle** 
- Menu: **Tools, Sketch Entities, Center Rectangle**
- Shortcut Menu: Right-click in the sketch and click **Center Rectangle**

6 Rectangle.

Click **Center Rectangle**  and click on the origin. Next, move diagonally away from center and click a location similar to that shown. Click .

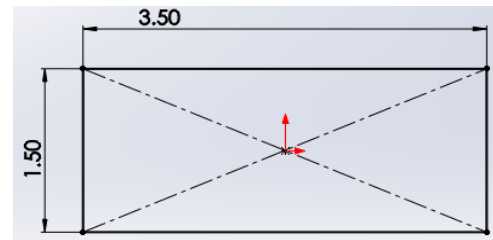
Note: The actual size is not important now. It will be defined in the following steps.




Dimensioning

The dimensioning tool is called **Smart Dimension** because the same tool is used to create multiple dimension types. Dimensions help to define the size of the profile and can be used to change it as well.



In this section it will be used to create linear dimensions, but it is also used to create angular, radial, and diameter dimensions.

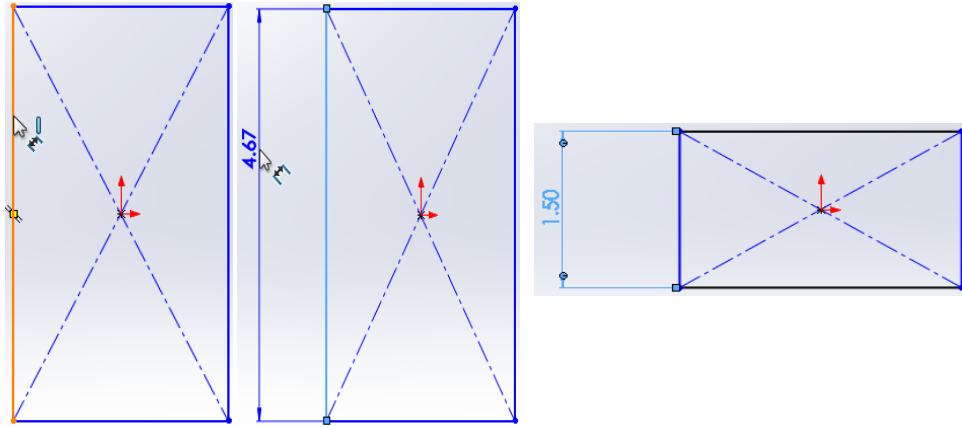


Where to Find It

-
- Menu: **Tools, Dimensions, Smart**
- Shortcut Menu: Right-click in the sketch and click **Smart Dimension** 

7 Vertical dimension.

Click **Smart Dimension**  and click the left vertical line as shown. Move to the left and click again to place the dimension. In the dialog that appears, type 1.5 to set the dimension value and click .



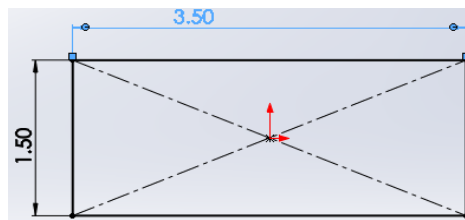
Geometry Color

Why do some lines remain blue while others are now black? A color code is used in the sketch to label the status of the geometry.

- ❑ *Blue* means it is under defined and requires more definition, in this case a dimension. This is the *unfinished* state.
- ❑ *Black* means that it is fully defined. This is the *desired* state.
- ❑ *Red* means that there is a conflict that must be resolved. An example of this is if you were to try to make a line be both horizontal and vertical; it cannot be both and a conflict, or over defined state, occurs. This is the *broken* state.


8 Horizontal dimension.

Click the upper horizontal line as shown. Move up and click again to place the dimension. In the dialog that appears, type 3.5 to set the dimension value and click ✓.



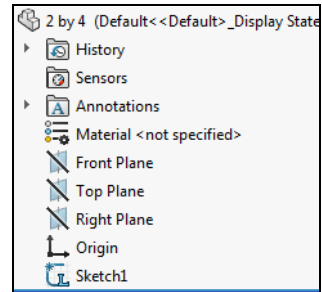
Note: The geometry is all black meaning that it is fully defined.

9 Exit the sketch.

Click **Exit Sketch**  in the upper right corner of the screen.

Saving the Sketch as a Library Feature

The sketch needs to be saved as a specific file type, a library feature, in order to be used as a weldment profile. It also must be placed in a folder that is defined to contain weldment profiles.



Where to Find It

- Menu Bar: **Save** , **Save As** 
- Menu: **File, Save As**

10 Library feature.

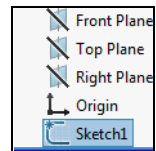
Click *Sketch1* in FeatureManager design tree.

Click **File, Save As**.

Click **Save as type: Lib Feat Part (*.sldlfp)**.

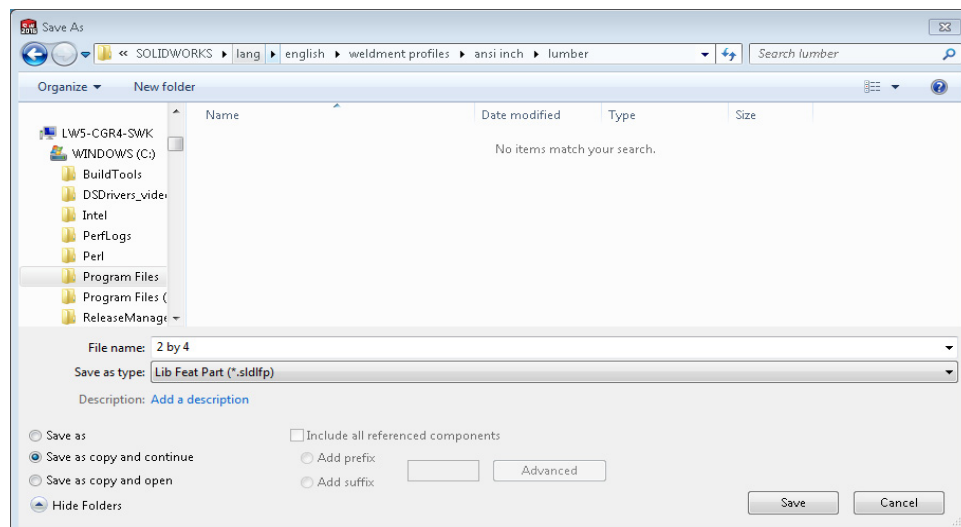
Browse to the folder *C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\lang\english\weldment profiles\ansi inch\lumber*.

Note: The selection of the sketch first is critical.



11 Name.

Type the name *2 by 4* and click **Save**.



Note: A warning might appear saying that you do not have permission to save in this location. If this occurs, save the file to another location and then manually move it to the lumber folder in the location above.

Creating a Similar Weldment Profile

Different profiles are needed to represent different shapes or sizes of weldment profiles. A larger lumber section, a 2x6, can be created from the existing 2 by 4 weldment profile by copying and editing the file.




12 Save as.

Click **Save As**, name the new library feature *2 by 6* and click **Save**.

Edit Sketch

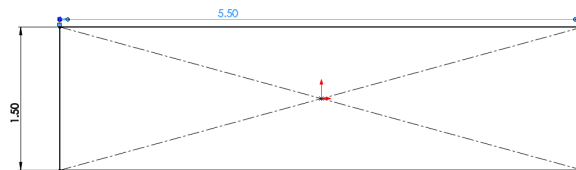
Existing sketches can be edited to add or change the geometry or dimensions.

Where to Find It


- Shortcut Menu: Right-click a sketch and click **Edit Sketch** 
- Menu: Select a sketch and click **Edit, Sketch**

13 Edit sketch.

Right-click the sketch *Sketch1* and click **Edit Sketch** . Double-click the **3.5** dimension and change it to **5.5**. Click .




14 Exit the sketch.

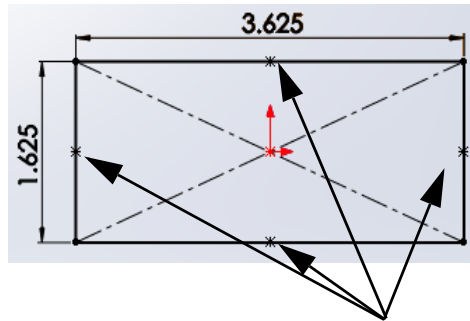
Click **Exit Sketch**  in the upper right corner of the screen.

15 Save and close.

Click **File, Close** and click **Save**.

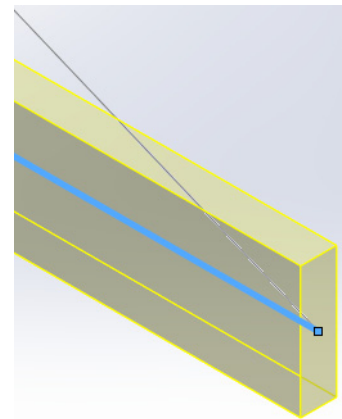
Additional Information About Weldment Profiles

A more general type of weldment profile for this shape would include a **Point**  at the midpoint of each edge as shown below. This would provide more flexibility in positioning the profile. They are not needed in this example.



Creating the Weldment Sketch

The weldment sketch is used to define the position and length of all the structural members.



16 Open the part **M Roof Truss**.

Click **Open** .

From the **Open** window, click the type:

SOLIDWORKS Files (*.sldprt; *.sldasm; *.slddrw) and browse to the *Bridge Design Project\Student\Lesson 9* folder.

Select *M Roof Truss.sldprt* and click **Open**.

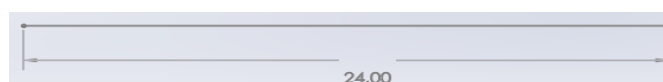
Note: The units of this part are **feet** with the material set as **Pine**.

17 Change to front view.

Click **View Orientation**  and click **Front** .

18 Edit sketch.


Edit the sketch *Sketch1*. For more information, see Edit Sketch on page 101.



Sketching a Line


Each line defines a single structural member in the weldment.

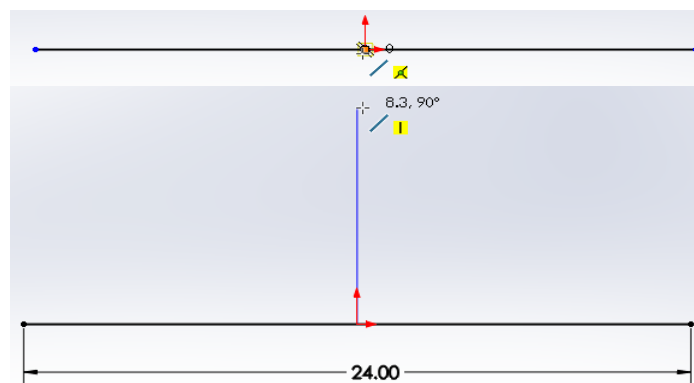
Where to Find It

- CommandManager: **Sketch > Line** 
- Menu: **Tools, Sketch Entities, Line**
- Shortcut Menu: Right-click in the sketch and click **Line**

19 Vertical line.

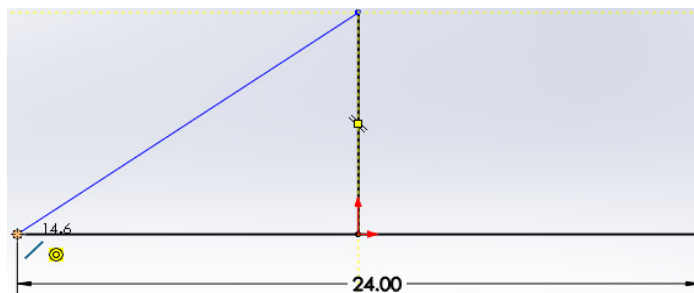
Click **Line** . Click at the origin, move vertically and click again as shown.

Note: The **Vertical** relation marker  indicates that the line is exactly vertical.



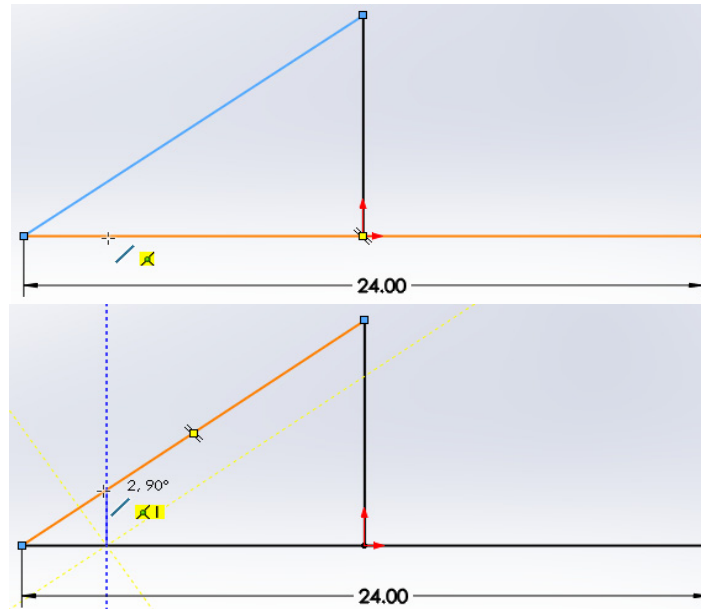
20 Angled line.

Click again on the lower left endpoint to add another line as shown.



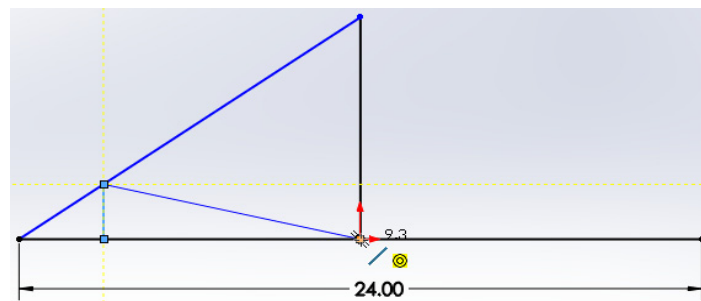
21 Vertical brace.

Create a line by clicking on the lower horizontal line and moving vertically upwards. Click again on the angled line to create a short, vertical line.




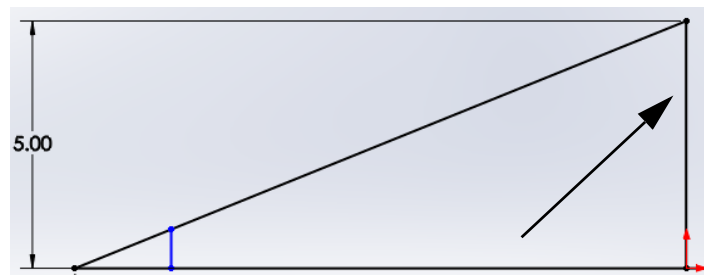
22 Angled brace.

Move back to the origin and click again to create an angled brace. Click ✓.



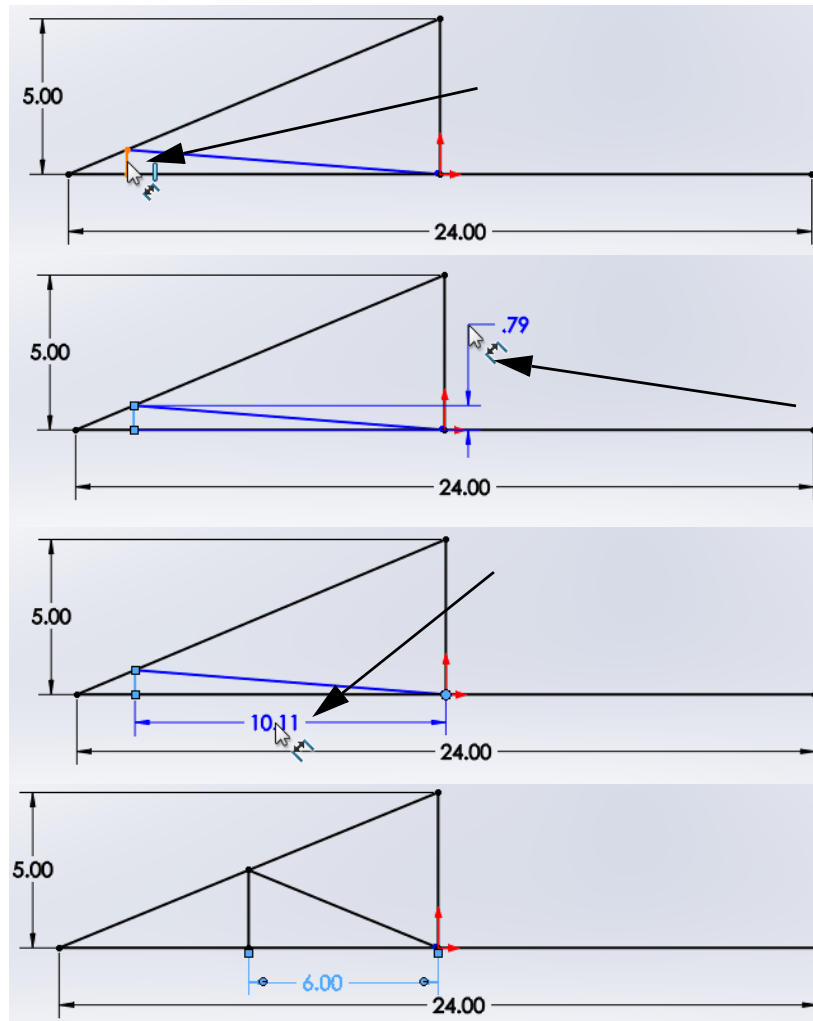
23 Vertical dimension.

Click **Smart Dimension** , click the vertical line and add a dimension. Set the dimension value to **5** as shown.



24 Dimension between lines.

Click the short vertical line followed by the center vertical line. Click below the sketch and place the dimension. Set the dimension value to **6** and Click **✓**.




Mirror Entities

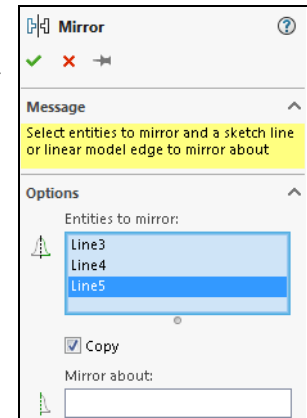
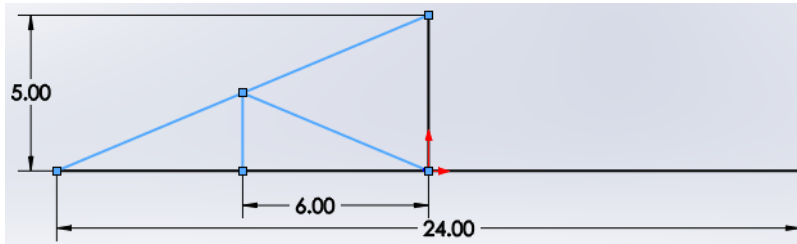
Sketch geometry can be copied by mirroring it across a line.

Where to Find It


- CommandManager: **Sketch > Mirror Entities** 
- Menu: **Tools, Sketch Tools, Mirror**

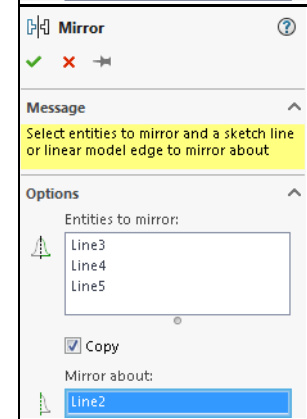
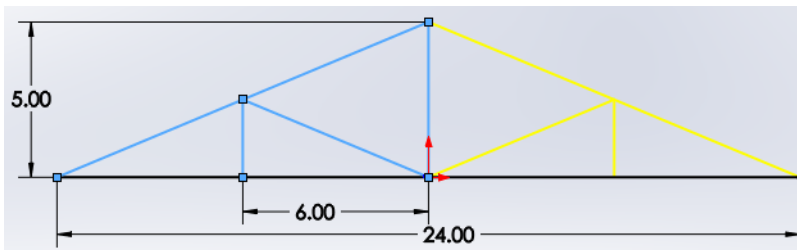
25 Entities to mirror.

Click **Mirror Entities**  and click the three lines shown below.




26 Mirror about.

Click in the **Mirror about** field and click the center vertical line as shown. Click .




27 Exit the sketch.

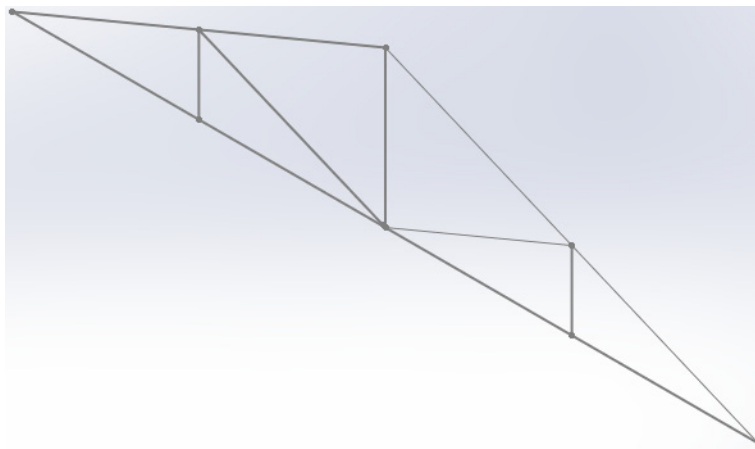
Click **Exit Sketch**  in the upper right corner of the screen.

28 Save.

Click **Save** .

29 Change to isometric view.

Click **View Orientation**  and click **Isometric** .



Adding the Structural Members

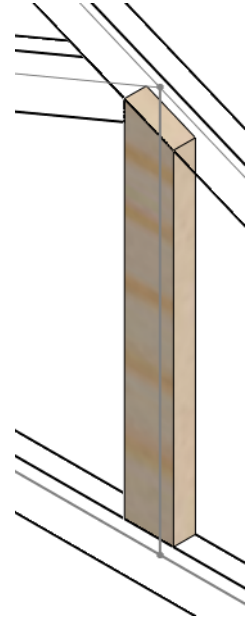
The structural members are added using the existing sketch geometry with a selected profile.

Structural Member

The length of a structural member is based on the length and position of the selected line.

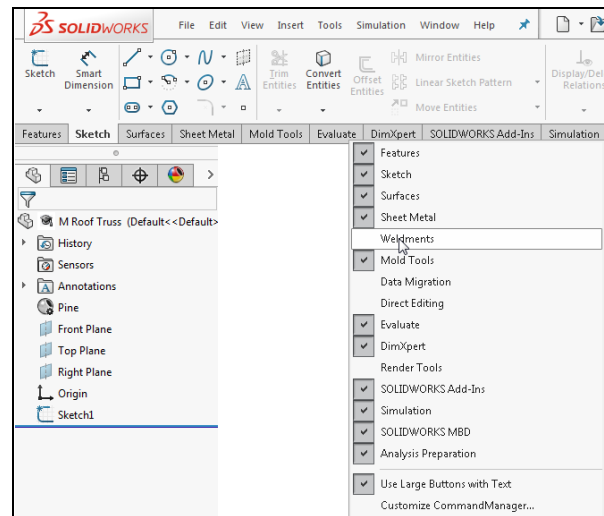
Group

Groups are sets of selections that force one structural member to be trimmed against others. This allows each structural member to be automatically trimmed to the proper size and shape.



CommandManager Tabs


Not all of the CommandManager tabs are visible. If the **Weldments** tab is not visible, right-click on any of the CommandManager tabs and click **Weldments** to show it.

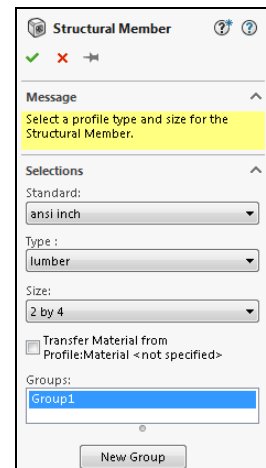
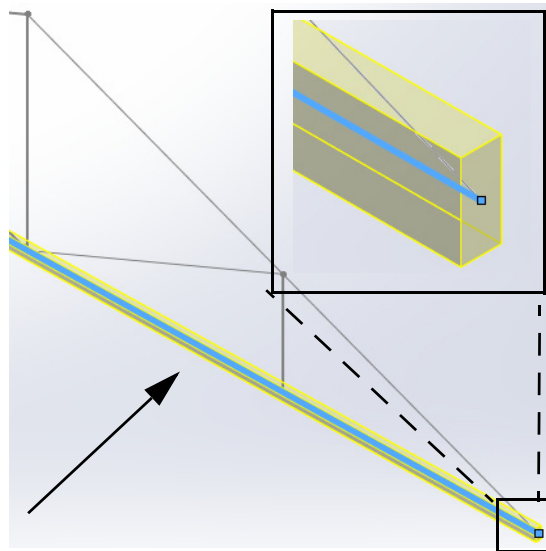


Where to Find It

- CommandManager: **Weldments > Structural Member** 
- Menu: **Insert, Weldments, Structural Member**

30 Structural member.

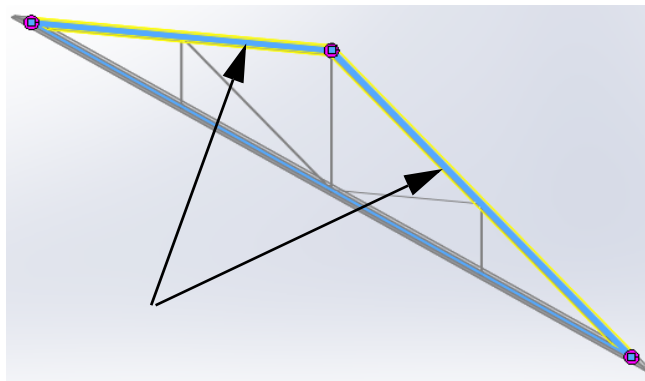
Click **Structural Member**  and click *ansi inch*, *lumber* and *2 by 4*.
Click the lower horizontal line as shown. Do not click ok yet.



Note:The preview shows the orientation of the profile as it is applied to the line.

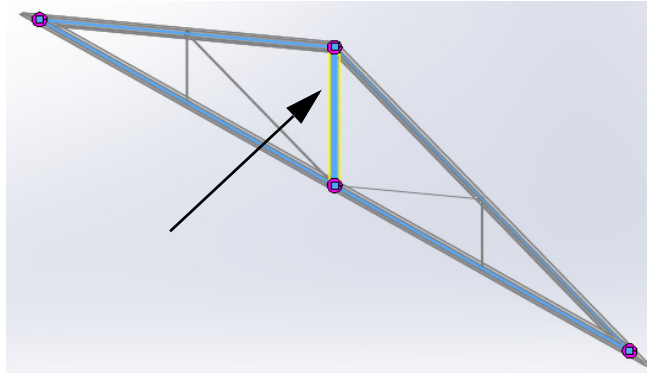
31 New group.

Click **New Group** and select both of the upper angled lines. Do not click ok yet.

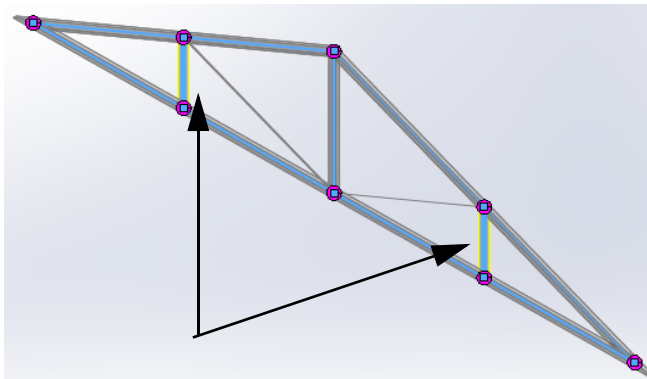


32 Next new group.

Click **New Group** and select the center vertical line. Do not click ok yet.

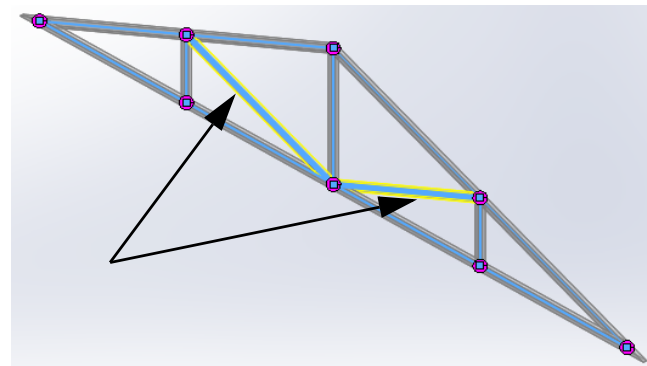
**33 Next new group.**

Click **New Group** and select both of the outer vertical lines. Do not click ok yet.

**34 Final new group.**


Click **New Group** and select both of the outer angled lines.

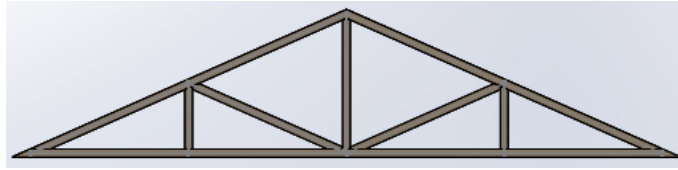
Click .

**35 Change to front view.**

Click **View Orientation**  and click **Front** .

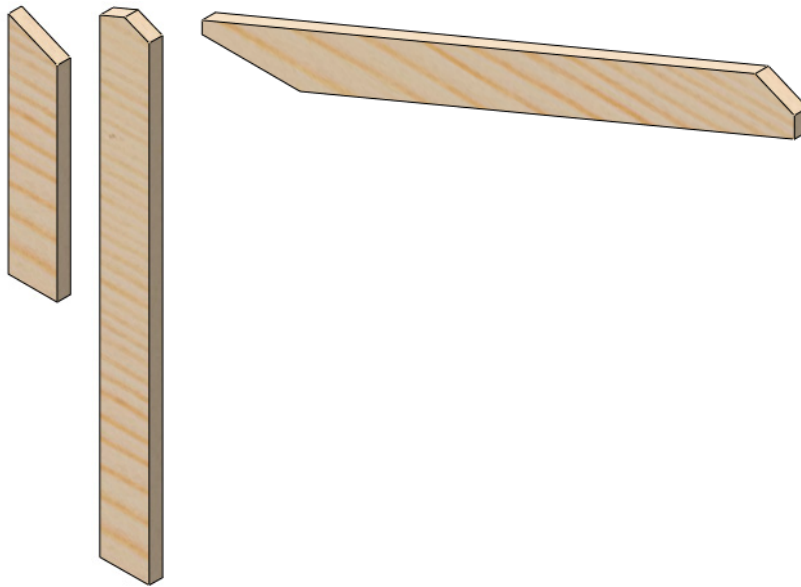
36 Hide sketch.

Right-click *Sketch1* and click **Hide** .



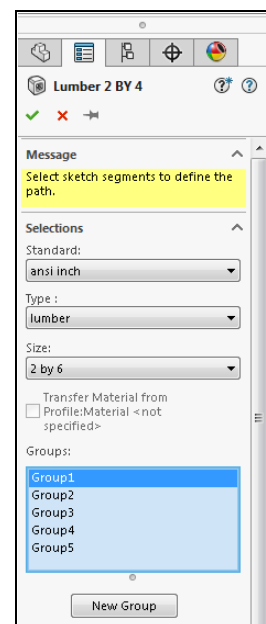
Multiple Bodies

The result of adding structural members is that multiple bodies are created in the part. Due to the use of **New Group**, each one is trimmed to the proper size and shape.




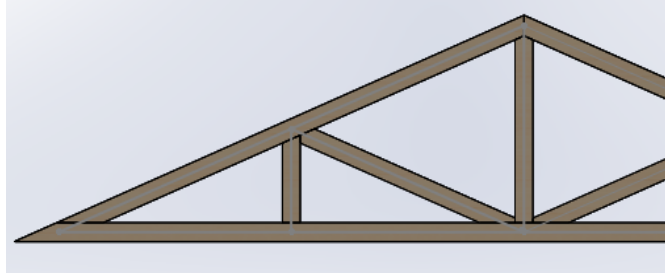
37 Edit feature.

Right-click the *Structural Member1* feature and click **Edit Feature**



38 Change size.

Select the **Size 2 by 6** and click .



39 Close the part.

Click **File, Close** and click **Save** to save changes.

Term	Definition
analysis	The process modeling the behavior of a structure to determine whether it can withstand the external loads it is designed to handle. Quantities such as displacements, stresses and factor of safety are calculated.
animate	View a model or eDrawing in a dynamic manner. Animation simulates motion or displays different views.
assembly	An assembly is a document in which parts, features, and other assemblies (sub-assemblies) are mated together. The parts and sub-assemblies exist in documents separate from the assembly. The extension for a SolidWorks assembly file name is *.sldasm.
beam	A beam is a structural member with a constant cross-section. It is usually loaded so that it bends.
bending	What happens to a beam when it is loaded along its length. Also called flexure.
component	A component is any part or sub-assembly within an assembly.
displacement	The movement of a beam from its original position after a load is applied.
document	A SolidWorks document is a file containing a part, assembly, or drawing.
drawing	A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is *.slddrw.
drawing sheet	A drawing sheet is a page in a drawing document.
eDrawing	Compact representation of a part, assembly, or drawing. eDrawings are compact enough to email and can be created for a number of CAD file types including SolidWorks and SolidWorks data.
element	A simple shape used to represent a small piece of the the model. The sum of all the elements represents the entire model.
environment	The outside factors that affect the structure. They include external loads applied to it and places where it is restrained from movement.
external load	A force or pressure that is applied to a structure from the outside. For a truss, it might be the weight of a train.

Term	Definition
face	A face is a selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces.
factor of safety	A value that is calculated in an analysis that determines whether a structure is strong enough to withstand the external loads applied to it.
feature	A feature is an individual shape that, combined with other features, makes up a part or assembly. Features are always listed in the FeatureManager design tree.
FeatureManager design tree	The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.
fixture	Fixtures are used to limit movement of points in the model. They are also called constraints or restraints.
graphics area	The graphics area is the area in the SolidWorks window where the part, assembly, or drawing appears.
line	A line is a straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.
material	What is used to create the beams in the structure. In a real structure, it would commonly be steel, but it can be wood or concrete. We are using wood.
meshing	The process of dividing the model into small pieces called elements.
model	A model is the 3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.
named view	A named view is a specific view of a part or assembly (isometric, top, and so on), or a user-defined name for a specific view. Named views from the view orientation list can be inserted into drawings.
newton	The SI (m-kg-s) unit of force. A force of one newton will accelerate a mass of one kilogram at the rate of one meter per second per second. In traditional English terms, one newton is about 0.225 pounds of force (lbf). The newton is named for Isaac Newton (1642-1727). He was the first person to understand clearly the relationship between force (F), mass (m), and acceleration (a), expressed by the formula $F = ma$.
node	A point used to connect and shape elements.
part	A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .sldprt.

Term	Definition
pascal	The SI (m-kg-s) unit of pressure and stress. It is defined as one newton per square meter. In traditional English terms, one pascal is about 145.04×10^{-6} pounds per square inch (psi). Because it is a very small amount, the related units MPa (Megapascals) and kPa (thousand Pa) are often used. The pascal is named for Blaise Pascal (1623-1662), a famous mathematician and physicist.
rectangle	A combination of four lines forming a rectangular shape in a sketch.
restraint	The restraint describes that part of the model that cannot move in the analysis.
sketch	A 2D sketch is a collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is non-planar and can be used to guide a sweep or loft, for example.
SolidWorks Simulation	The software within SolidWorks that is used to perform a structural analysis.
simulation study	A folder used to store a complete analysis including: materials, fixtures, external loads and mesh.
simulation study tree	A tree structure, similar to the FeatureManager Design Tree, that contains the features that make up a simulation.
strength	The strength or stiffness of a beam includes both the cross section shape (area moment of inertia) and material.
stress	Stress is a quantity measured by force per unit area inside a structure that is caused by external loads applied outside of the structure. Common units are Pascals and pounds per square inch.
stress distribution	A “map” of colors that displays the amount of stress anywhere on the part. Colors are used to represent stress value ranges.
structure	A collection of beams that are used to form a single part. In SolidWorks, this type of part is called a weldment; multiple pieces welded into one.
structural analysis stages	The stages in a generic analysis including pre-processing (setup), analysis and post-processing (looking at the results). Specifically, we use SolidWorks Simulation.
structural member	A single body in a weldment structure that represents a beam or length of lumber.
tension and compression	Internal forces in a beam that are caused by bending.
truss	A simple bridge structure usually used by railroads.
unit system	A combination of length, mass and time units usually described by the selections such as IPS or MMGS.
weldment	A structure based on a 2D or 3D sketch, a profile and multiple bodies in a single part.

Term	Definition
weldment profile	A 2D sketch that represents the cross-section of a structural member.
yield strength	The limit of a beam's strength based on the stresses in the beam.