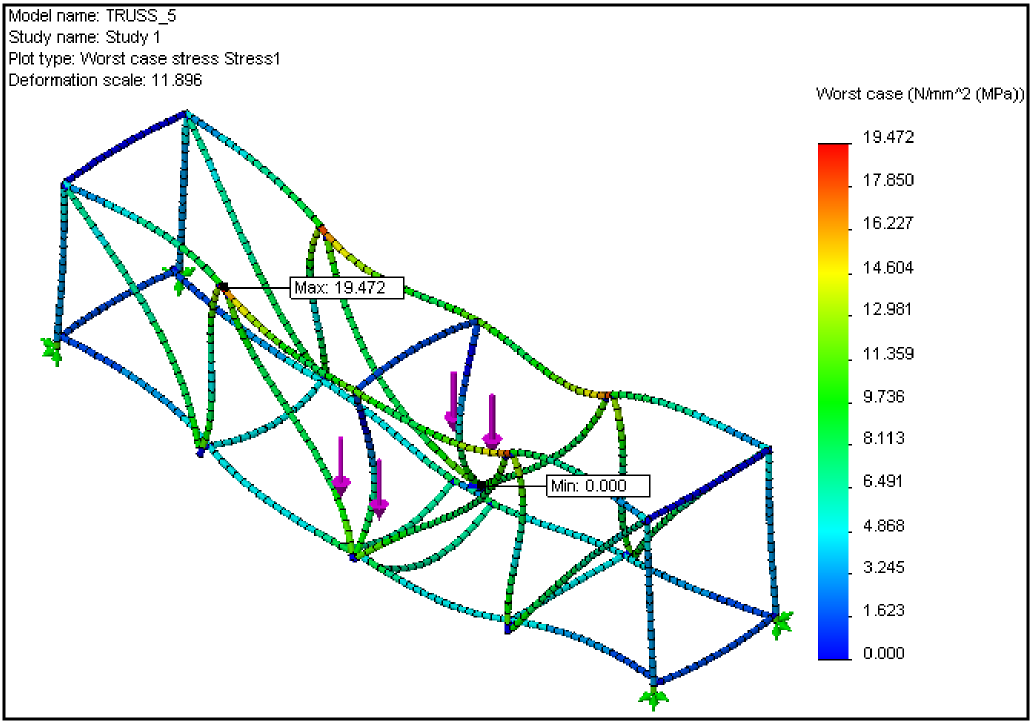
Engineering Design and Technology Series

Bridge Design Project with SolidWorks® Software



Dassault Systèmes SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA
Phone: +1-800-693-9000

Outside the U.S.: +1-978-371-5011
Fax: +1-978-371-7303
Email: info@solidworks.com
Web: http://www.solidworks.com/education

Document Number: PME0518-ENG

Table of Contents

Lesson 1: Introduction **1**
Using This Book 2
What is SolidWorks Software? 2
Prerequisites 2
Conventions Used in This Book 3
Before You Begin 3
Analyzing a Structure Using SolidWorks and SolidWorks Simulation 5

Lesson 2: Structure Design **6**
What is a Structure? 7
Structure Designs 8
Trusses 8
Beams 9
Strength 10
Cross Section Shape 11
Try it! 11
Displacement 11
Material 13
Truss Walls 14
Triangles 14
Try it! 15

Lesson 3: Using the Beam Calculator **16**
Using Beam Calculations 17
Order of Magnitude 17
Starting SolidWorks and Opening a Part 18
Adding in SolidWorks Simulation 18
The Model Geometry 19
Simplifying the Analysis 19

The Simply Supported Beam 20
Fixtures 20
External Loads 20
Theoretical Model 20
Why are Simply Supported Beams Important? 20
Required Data for the Beam Calculation 22
Collect the Data 23
Assign a Material 24
Section Properties 25
Using Measure 27
Beam Calculator 28

Lesson 4: Analyzing the Structure 30
Analysis of the Structure 31
What is SolidWorks Simulation? 31
Structural Analysis 32
Structural Analysis Stages 33
Design Cycle 34
Changes in the Model 34
Create a Study 35
FeatureManager Design Tree and Simulation Study Tree 36
The Environment 36
Pre-Processing 39
Material 39
Fixtures 40
External Forces 41
Meshing the Model 43
Analysis 44
Expectations 44
Some Terminology 45
Bending and Displacement 45
Tension and Compression 46
Stresses 46
Yield Strength 46
Factor of Safety 47
Post-Processing 47
Interpreting the Results 48
Creating a New Plot 49
Iterating Changes 50
Determine the Load 50
Editing Simulation Data 50
Conclusion 51

Lesson 5: Making Design Changes 52
Adding to the Design 53
Open the Model 53
Existing Study 53
Increase the Load 54
Cross Bracing 55
Open the Model 55
Existing Study 55
What did the Cross Bracing do? 56
Working with Plots 57
Deformation Plot Factor 57
Superimposing the Model 57
The Weakest Link 58
Using a Probe 60
Adjusting the Number Format 61
Solution 62
Finishing the Bracing 62
Compare Stresses 63
Top Beams 64
Strength to Weight Ratio 65
Efficiency Comparison 66
More to Explore 67
Reading the Plot 68

Lesson 6: Using an Assembly 69
Creating an Assembly 70
Testing using the Test Block 70
Changing the Model 71
Collision Detection 72
Updating the Analysis 73

Lesson 7: Making Drawings of the Structure 76
Drawings 77
Creating a Drawing and Views 77
What is a Weldment Cut List Table? 79
Why are there two Items of the Same Length? 80
Balloons 81

Lesson 8: Reports and SolidWorks eDrawings® 82
Reports and SolidWorks eDrawings 83
Creating a Report 83
SolidWorks eDrawings® for Sharing Information 86

Advantages of eDrawings 86
Viewing eDrawings 86
Creating a SolidWorks eDrawing 87
The eDrawings User Interface 89
eDrawings Functions 89
Playing an eDrawings Animation 90
Saving eDrawings 90
Save the eDrawing 90
More to Explore 92

Lesson 9: Building and Testing the Structure 93
Building the Structure 94
Cutting to Length 94
Testing the Structure 102
Creating the Span 102
Details 102
Applying the Load 103
Using Common Objects with Known Weights 103

Glossary 104

Lesson 1
Introduction

When you complete this lesson, you will be able to:

- 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 online tutorials that are integrated in the SolidWorks software:

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

You can access the online tutorials by clicking **Help**, **SolidWorks Tutorials, All SolidWorks Tutorials (Set 1)**. The online tutorial resizes the SolidWorks window and runs beside it.

As an alternative, you can complete the following lessons from *An Introduction to Engineering Design With SolidWorks*:

- 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:

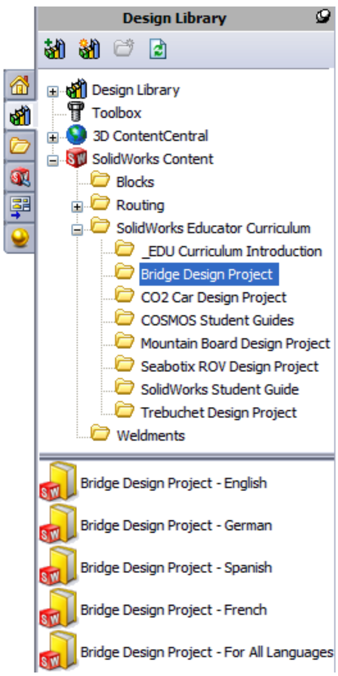
Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, **Insert, Boss** means choose the **Boss** option from the **Insert** menu.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step.	The steps in the lessons are numbered in sans serif bold.

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 Content.
Click **Design Library** to open the design library task pane.



Click on `SolidWorks Content` to show the folders below it.

Click on `SolidWorks Educator Curriculum`.

Click `Bridge Design Project - English`.

Note: There may be more curriculum folders listed in addition to the `Bridge Design Project`.

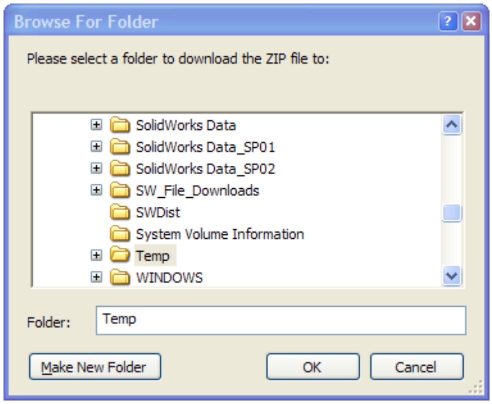
The lower pane will display an icon representing a Zip file that contains the companion files for this project.

3 Download the Zip file.
Press **Ctrl** and click the 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.

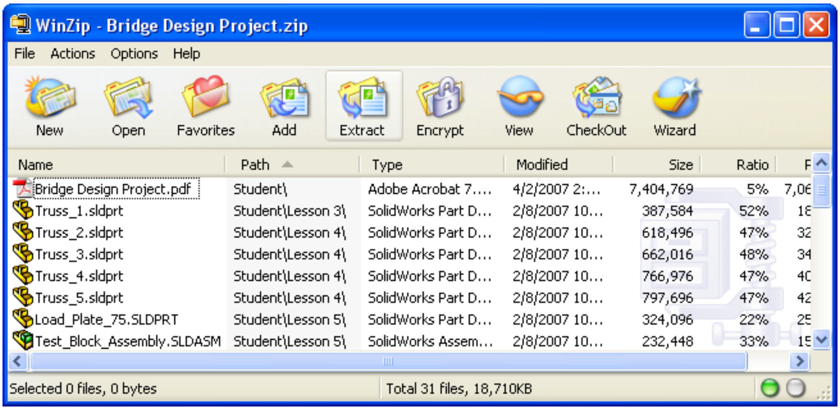
Click **OK**.



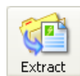
Tip: Remember where you saved it.

4 Open the Zip file.
Browse to the folder where you saved the Zip file in step **3**.

Double-click the Bridge Design Project.zip file.



5 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 94).

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 2
Structure Design

When you complete this lesson, you will be able to:

- 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.

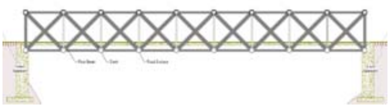
Trusses

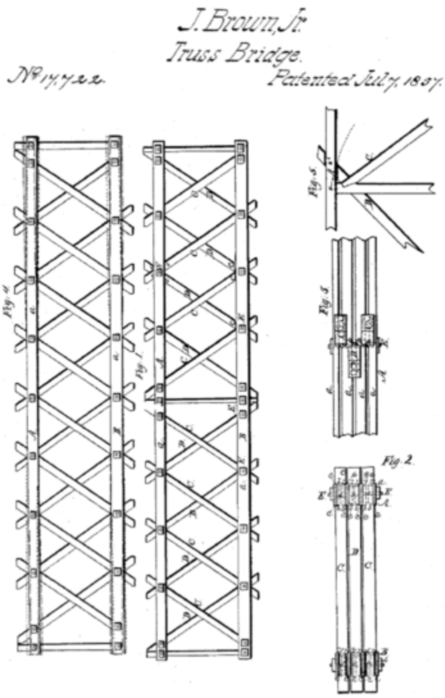
Trusses are specific types of structures commonly used a railroad bridges. They usually consist of a road or rail surface (deck), two walls and sometimes bracing on the top. You will be analyzing a truss design.

Search on **truss** for more information.

Brown Truss

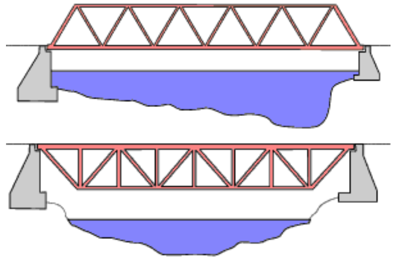
The **Brown Truss** (patent shown here) was used in the design of covered bridges. This truss is a “box” truss (named for it’s boxy shape) that was so efficient that it could be constructed using only the (diagonal) cross bracing beams to support it.





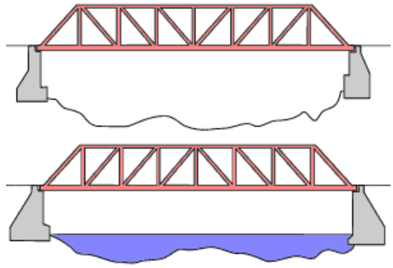
Warren Truss

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



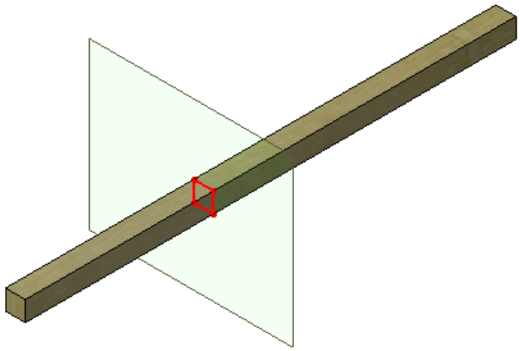
Pratt and Howe Trusses

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



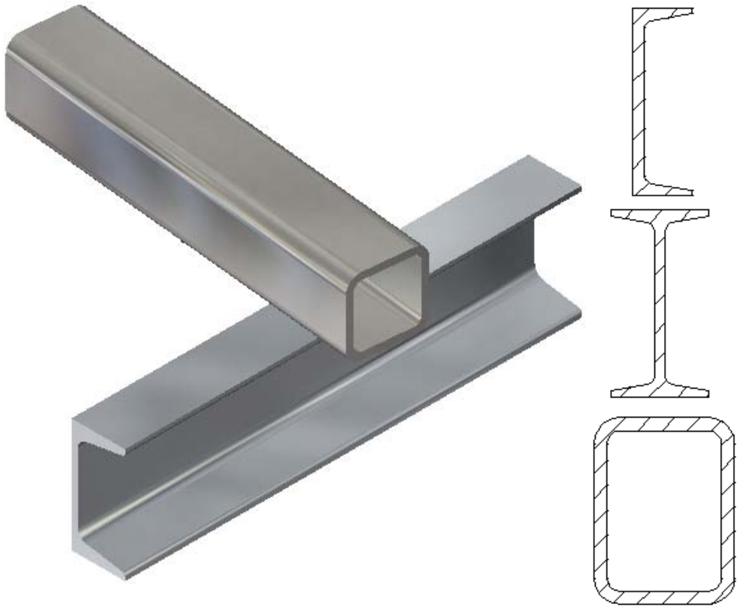
Beams

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



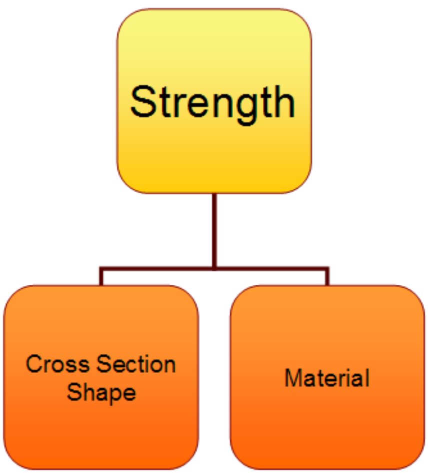
Steel Beams

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



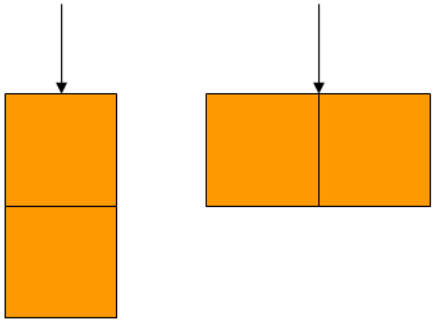
Strength

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



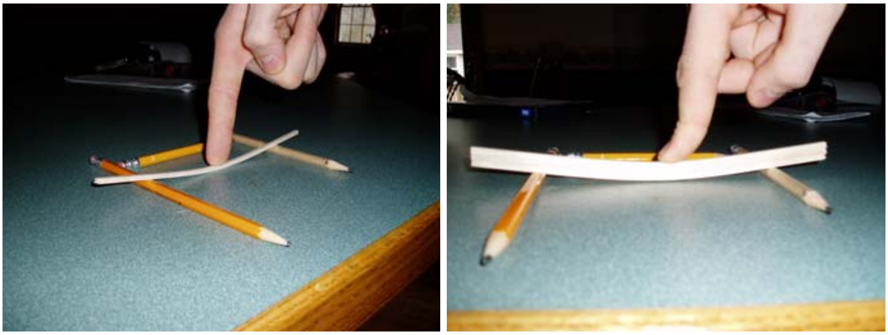
Cross Section Shape

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.



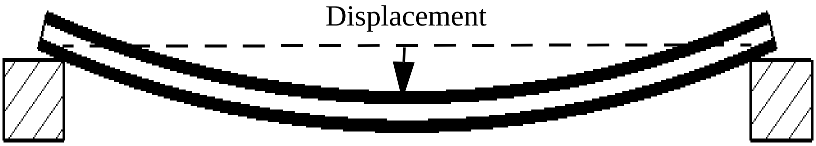
Try it!

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.

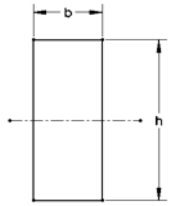


Displacement

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.

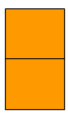
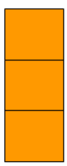
Calculating the Area Moment of Inertia

Using the formula below, you can calculate this value for several arrangements to square 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.

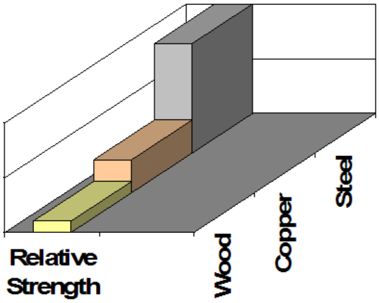
Number of square sections	Arrangement of square sections	b	h	Area Moment of Inertia
1		3.175mm	3.175mm	__________
2 Stacked		3.175mm	2 X 3.175mm	__________
2 Side by Side		2 X 3.175mm	3.175mm	__________
3 Stacked		3.175mm	3 X 3.175mm	__________

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 that the beam is made of is another critical factor in 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 right. In general, steel is stronger than copper which is in turn 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 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.

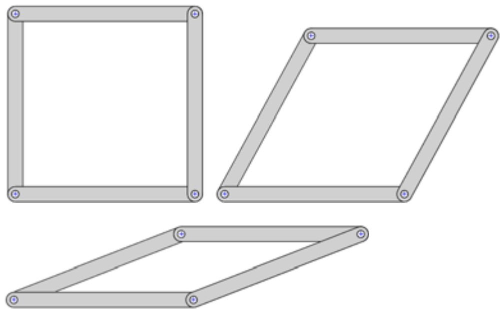
Truss Walls

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.

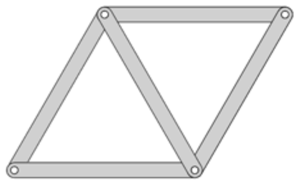
Triangles

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.

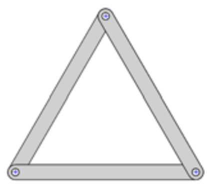
Consider a collection of members connected in a square shape by bolts or pins. Holding the bottom still, push on the top or side. It can form a square but can also be easily pushed into a flattened parallelogram.



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.

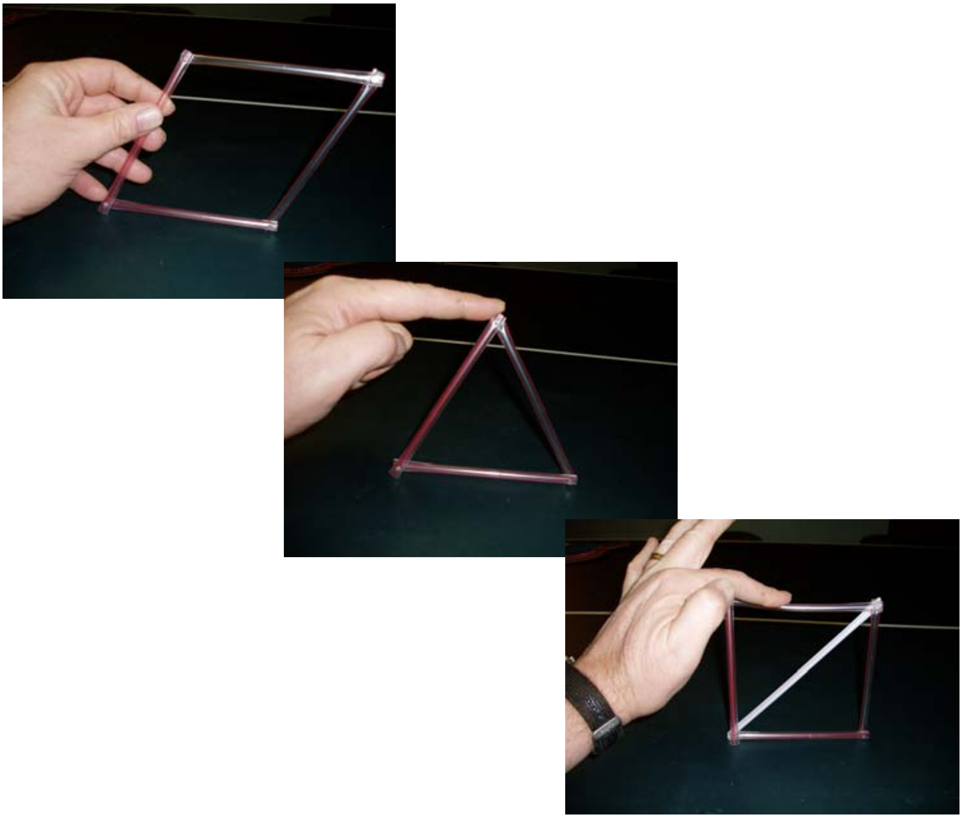


Using the same members and fasteners, create a triangle. This time fewer members are used but stability is achieved.



Try it!

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



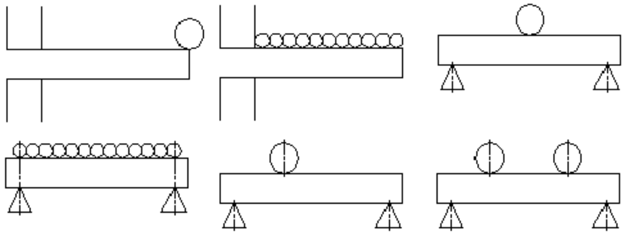
Lesson 3
Using the Beam Calculator

When you complete this lesson, you will be able to:

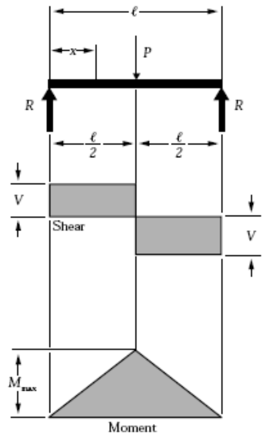
- Start SolidWorks;
- Add-in the SolidWorks Simulation software;
- Open an existing SolidWorks part;
- Understand a simply supported beam;
- Assign a material;
- Calculate section properties;
- Use the measure tool;
- Use the beam calculator to calculate a displacement.

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 11) 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

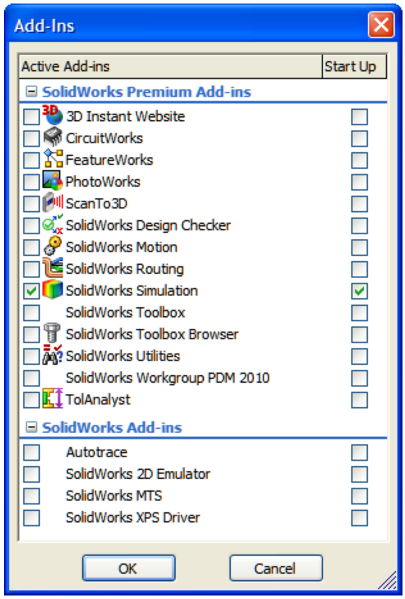
Starting SolidWorks and Opening a Part

1 Start the SolidWorks application.
From the **Start** menu, click **Programs**, **SolidWorks**, **SolidWorks**.

Adding in SolidWorks Simulation

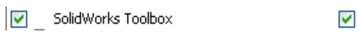
SolidWorks Simulation software is included with **SolidWorks Education Edition**. To use it, it must be activated using **Tools, Add-Ins**. Click both **Active Add-ins** and **Start Up** for SolidWorks Simulation and click ✓.

2 Add-in SolidWorks Simulation.
Click **Tools, Add-ins** and make sure that both **Active Add-ins** and **Start Up** for **SolidWorks Simulation** are checked.



Note: If the SolidWorks Simulation software is not added in the project cannot be completed.

3 Add-in SolidWorks Toolbox.
Click **Tools, Add-ins** and make sure that both **Active Add-ins** and **Start Up** for **SolidWorks Toolbox** are checked. Click ✓.



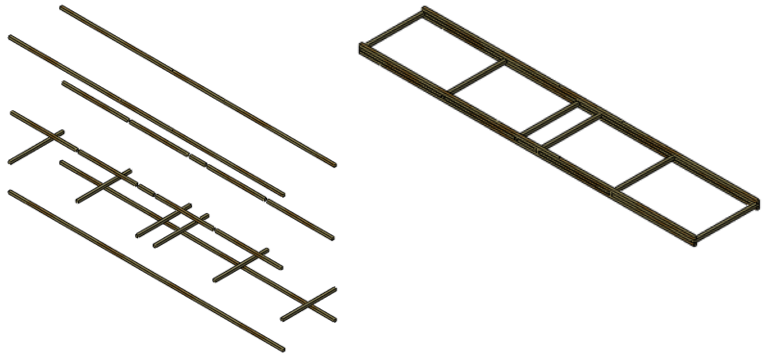
4 Open the part file.
Click **Open**.

From the **Open** window, browse to the Bridge Design Project\Student\Lesson 3 folder.

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

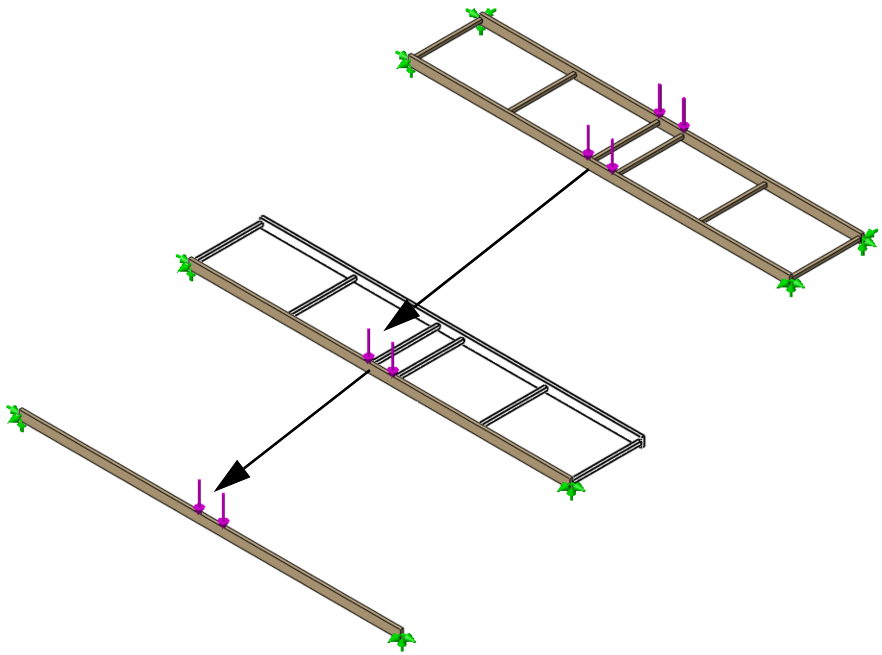
The Model Geometry

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

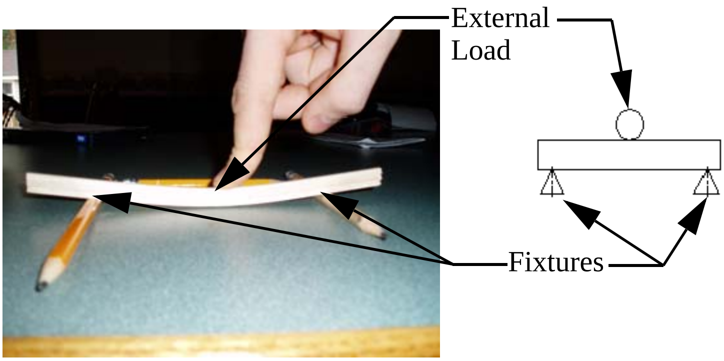
Fixtures are used to limit movement of certain points in the model. These are usually points of contact. They are also called constraints or restraints.

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.

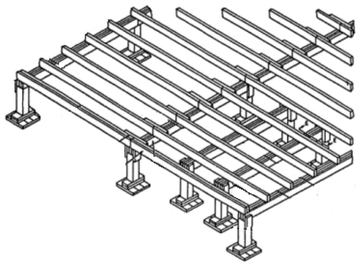


Why are Simply Supported Beams Important?

Although the theoretical model may seem very simple, it has far reaching effects. 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

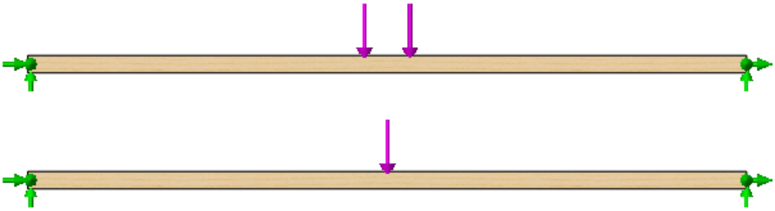
If you were standing in the middle of a mountainboard, you would be the external load and the wheels would be the fixtures. The structure can be approximated with a simply supported beam.

Note: This example is a “simplified analysis” that takes 3 dimensional problem and simplify it to 2 dimensional problems. 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 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 by 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, 40N/2 = **20N**, 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)

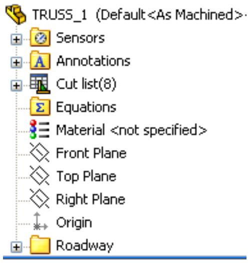
Assign a Material

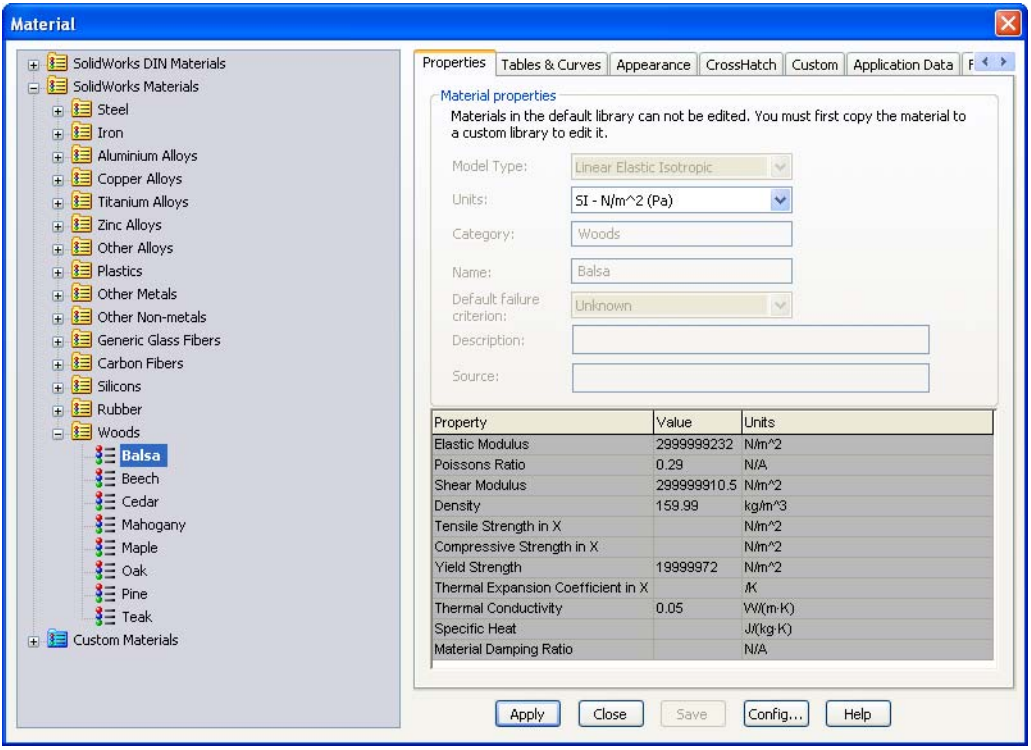
The first step is to assign a **Material** to the beams of the model. We want to make the structure out of balsa wood.

5 **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/m^2 (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.

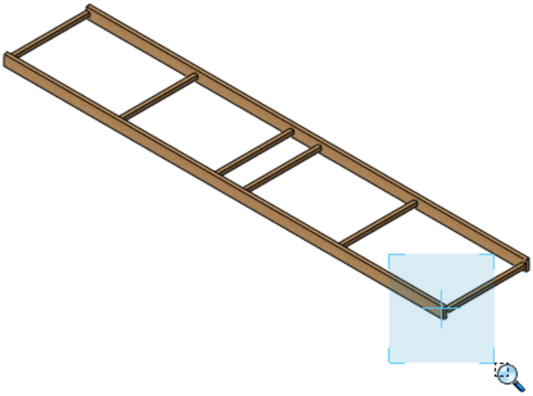
The value of the Elastic Modulus or Modulus of Elasticity = 2999999232 N/m^2.

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

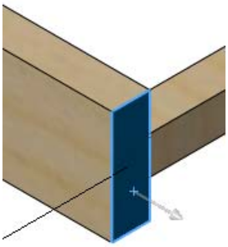
Section Properties

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

6 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.



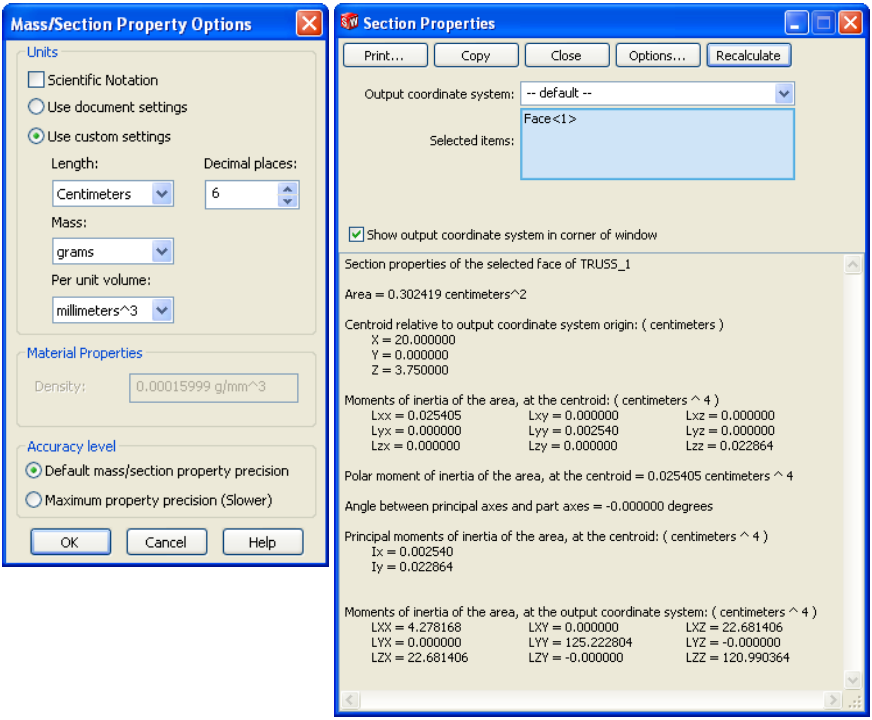
7 Face selection.
Select the face as shown.



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

Click ✓ and **Recalculate**.

Moments of inertia of the area, at the centroid: (centimeters ^ 4) Lxx = 0.025405. Click **Close**.



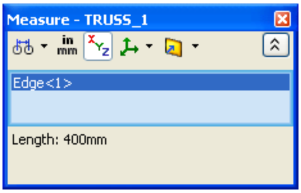
9 Zoom.
Click **View, Modify, Zoom To Fit** or click the **F** key to return to the full view.

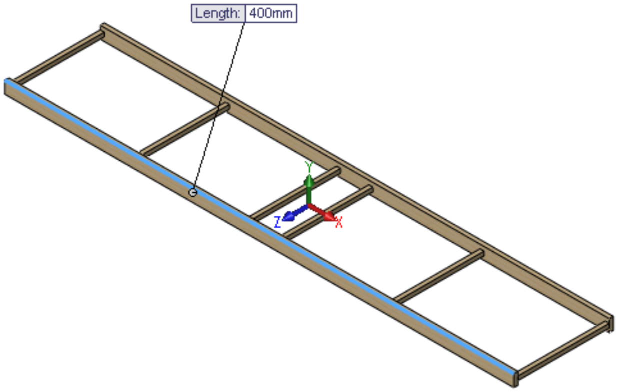
Using Measure

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

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

Length: 400mm.





11 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 of the beam. Retrieve the data from the chart “Collect the Data” on page 23.

Note: This dialog uses *deflection* but we will be referring to it as *displacement* throughout the manual.

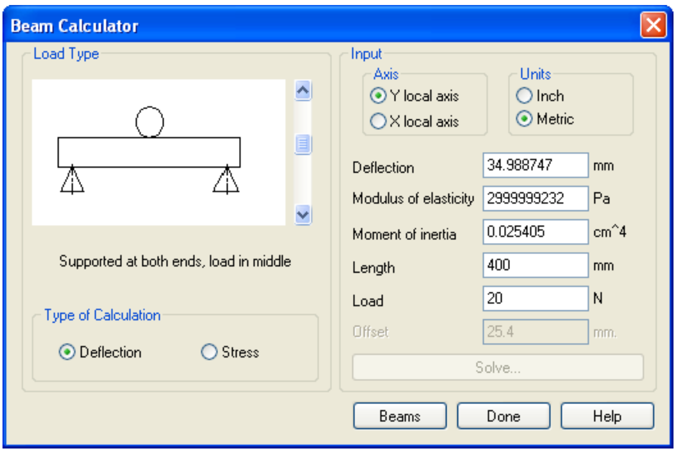
12 Start beam calculator.
Click **Toolbox, Beam Calculator** .

13 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**.

14 Type in values.
Type 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)

Click **Solve**. The displacement is about **35mm** at the load (center of the beam). Click **Done**.



Note: The **Solve** button will not be available until the **Deflection** field is cleared.

Questions

1. Is this more or less than an inch?______________
2. Convert to inches: 35mm/25.4 = ______________in

15 Close the part.

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

At the `Save changes to TRUSS_1?` message, click **No**.

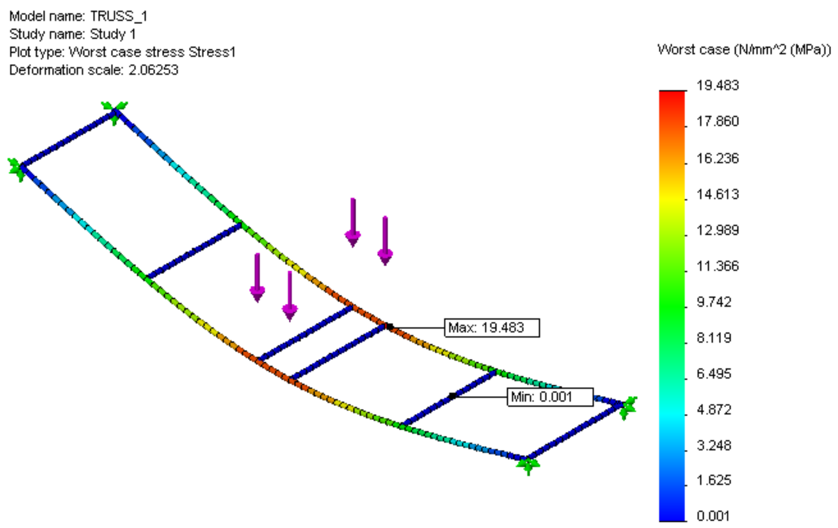
Lesson 4
Analyzing the Structure

When you complete this lesson, you will be able to:

- 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.

$$\frac{MaximumStressunderLoading}{MaximumStressoftheMaterial} = FOS$$

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 it's unloaded position. The displacement is the distance a point moves from it's 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

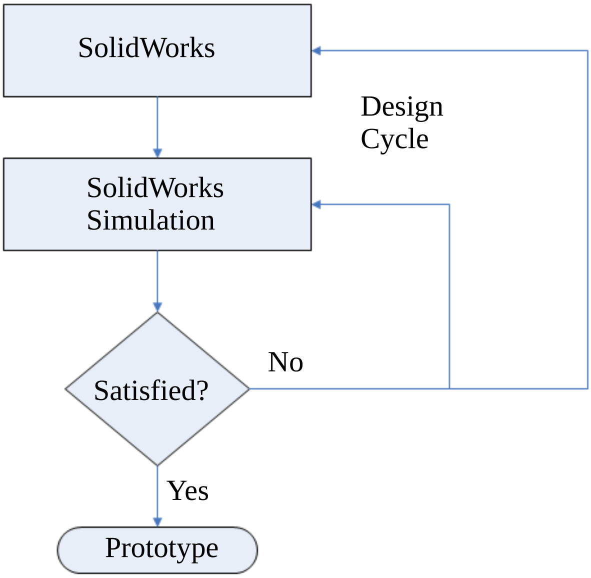
SolidWorks Simulation walks you through several stages of structural analysis. This is what is happening behind the scenes:

- **Pre-Processing**- In this stage, you add the required information about the structure and the environment where lives. 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.

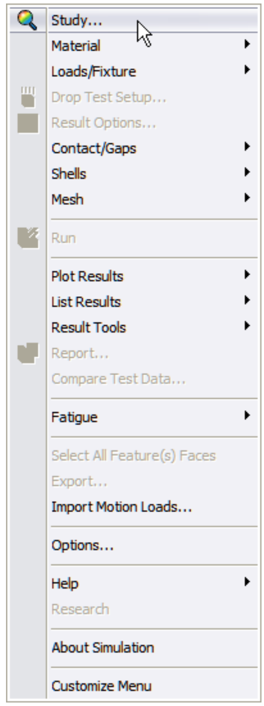
Click **Open**.

From the **Open** window, browse to the Bridge Design Project\Student\Lesson 3 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:

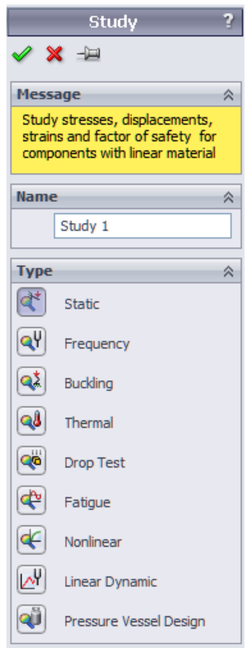
- **Static**
- **Frequency**
- **Buckling**
- **Thermal**
- **Drop test**
- **Fatigue**
- **Nonlinear**
- **Linear Dynamic**
- **Pressure Vessel Design**

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

Access the study from the **Simulation** pull-down menu.

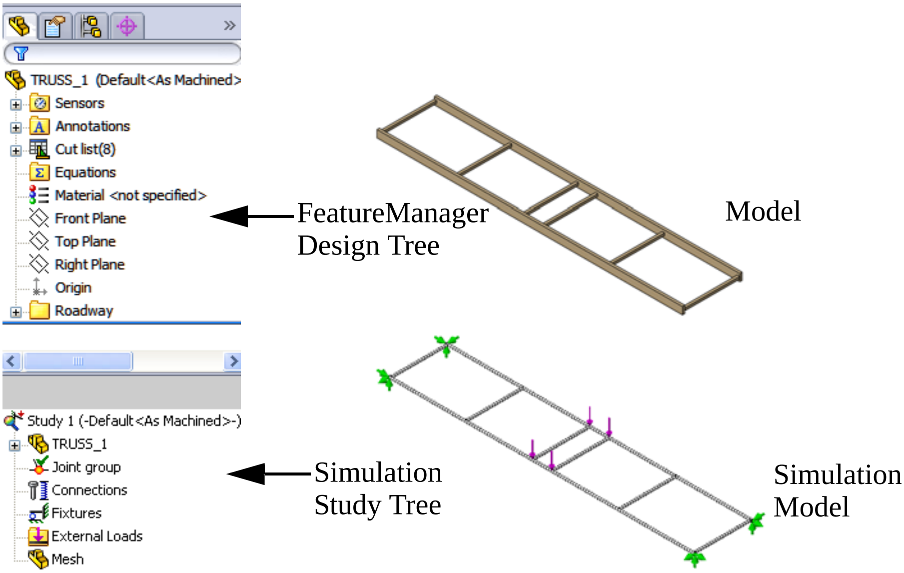
2 Create a new study.
Click **Simulation, Study**. Use the default name `Study 1` and click **Static**.

Click ✔.



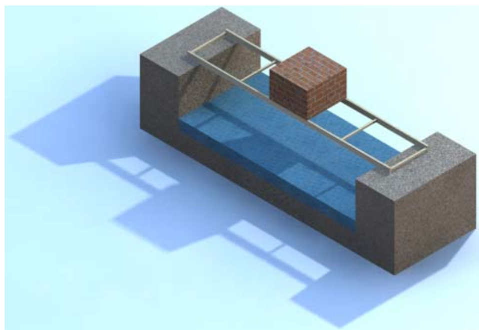
FeatureManager Design Tree and Simulation Study Tree

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.



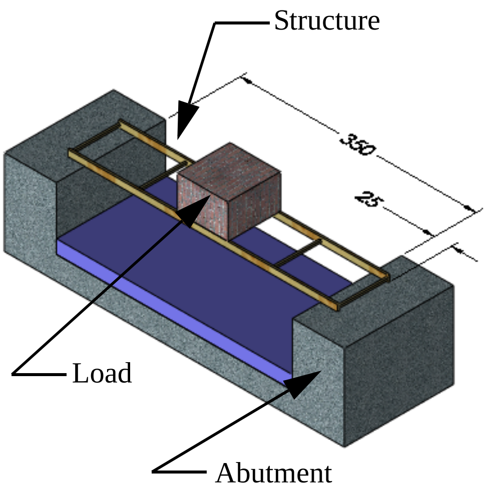
The Environment

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**.

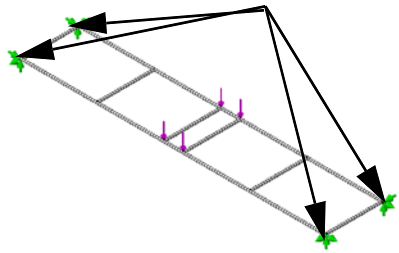


Fixtures

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.

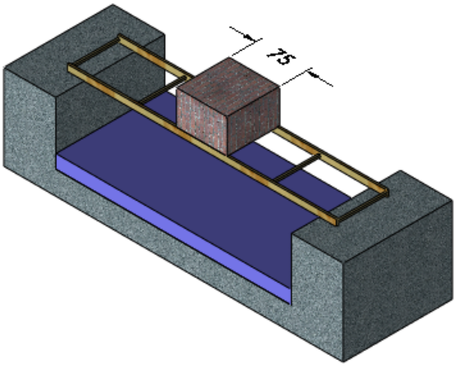


The fixtures are defined at the ends of the model in four places.

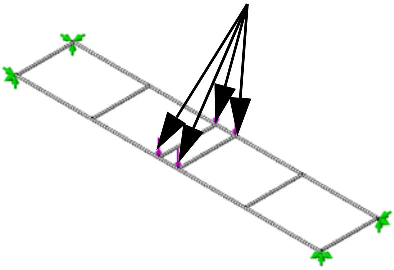


External Loads

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**.

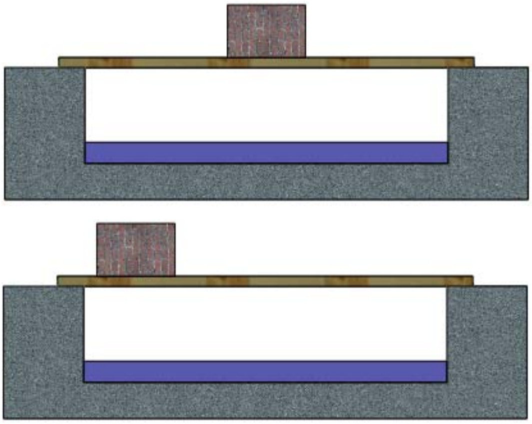


There are four loading points, one for each point where the beams connect near the center of the span. This means that the load on each point is 40N/4 = **10N** (about 2.25lbs).



Why is the load in the center?

When using 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 manual. 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 lb = 4.4482 N

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

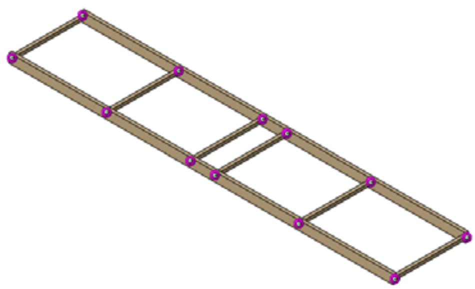
The Material is a required value that sets the material properties and appearance of the model geometry. In this case it will be applied to all the beams at once.

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

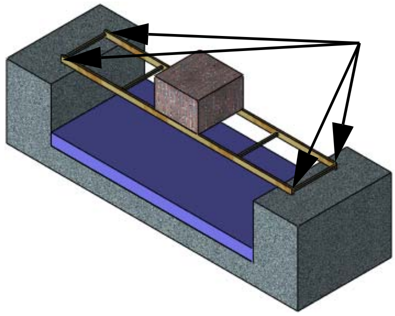
What are Joints?

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

Note: The joints will appear when you are adding fixtures and external loads.



Fixtures

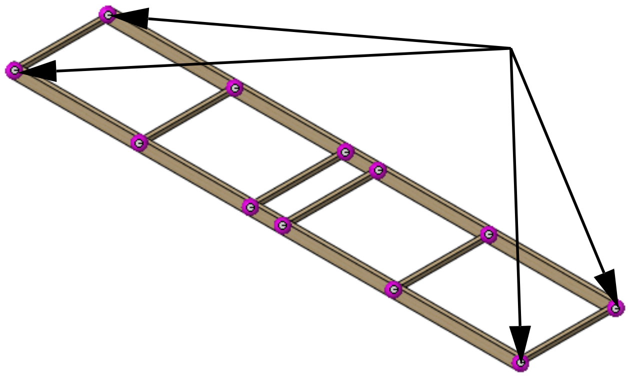


Fixtures are used to limit movement of certain points in the model. The points where the ends of the structure sit on the abutment will be assigned fixtures.

What type of fixtures?

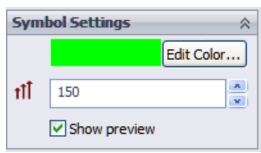
In this project, the bridge will be placed on the abutment so that it crosses the span. The bridge will contact the abutment but it will not be glued or attached in any way.

4 **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.

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

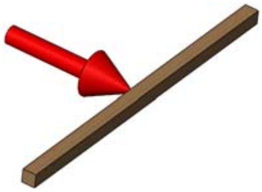


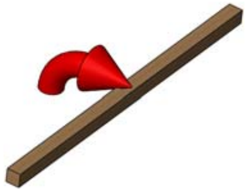
External Forces

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 a direct *force* like hanging a weight or a *moment* that twists or bends like turning a doorknob.

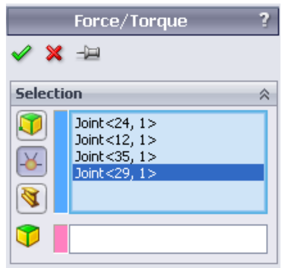


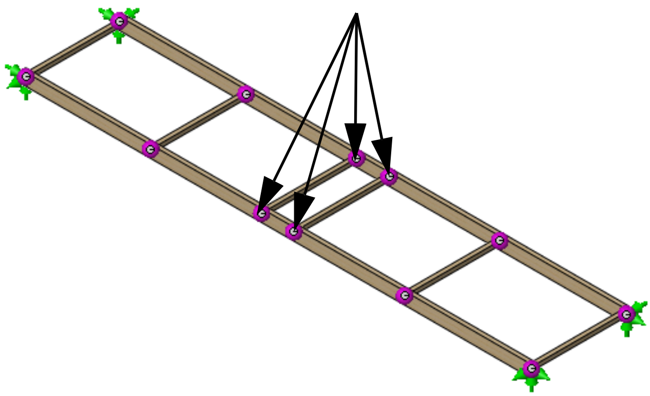


Gravity

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

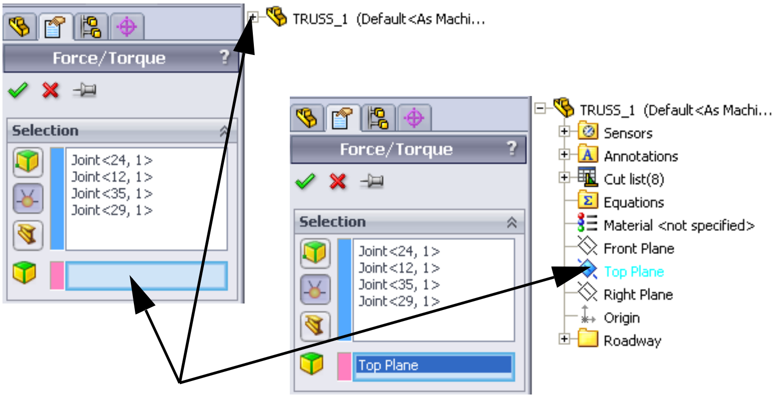
6 Select joints.
Click **Simulation, Loads/Fixture, Force**. Click **Joints** and select the visible joints as shown.





7 Set direction.

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



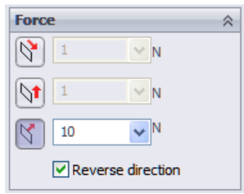
8 Set units.

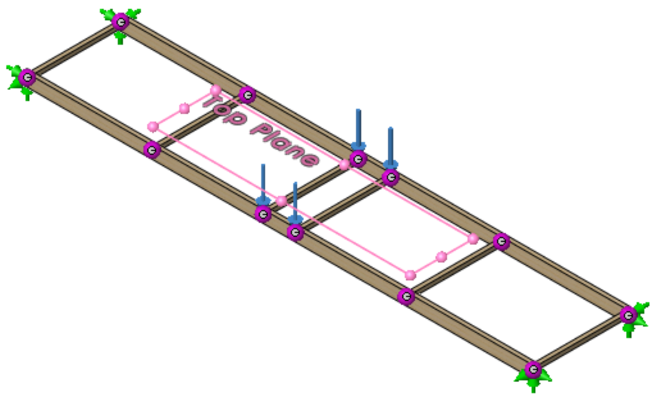
Make sure that **Units** are set to **SI**.

9 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**.

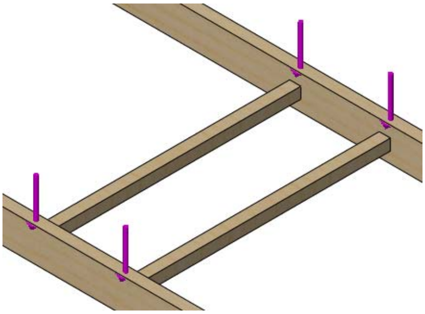
10 Save.

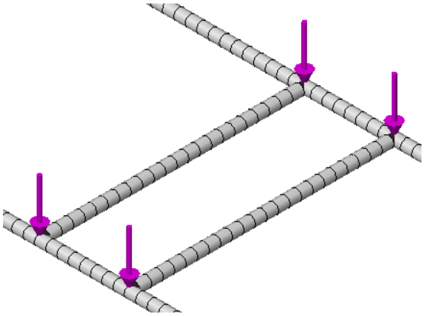
Click **Save** to save the model and simulation data.

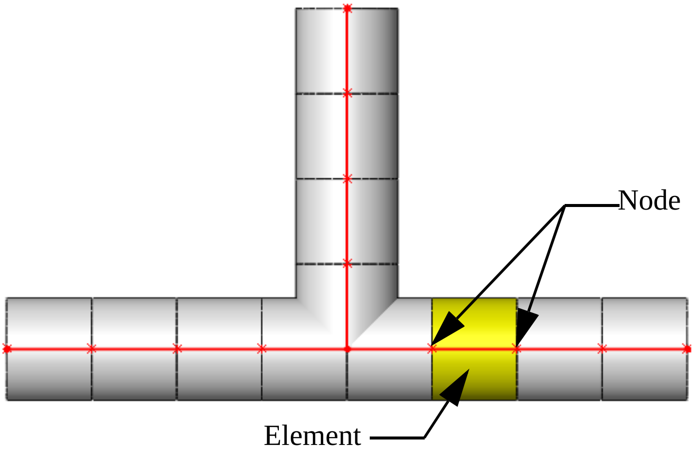
Tip: It is a good idea to save periodically and prevent unintentional loss of data.

Meshing the Model

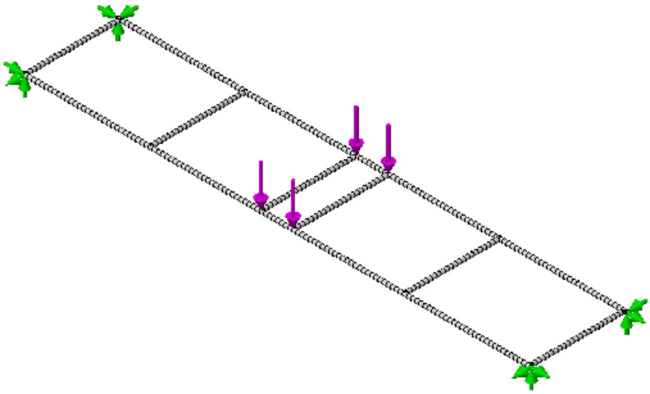
The mesh must be created to generate the small pieces used in the analysis. The analysis model is made up of a series of connected nodes and elements.







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



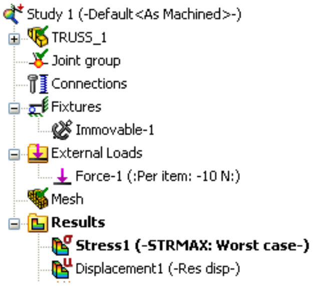
Analysis

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.

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



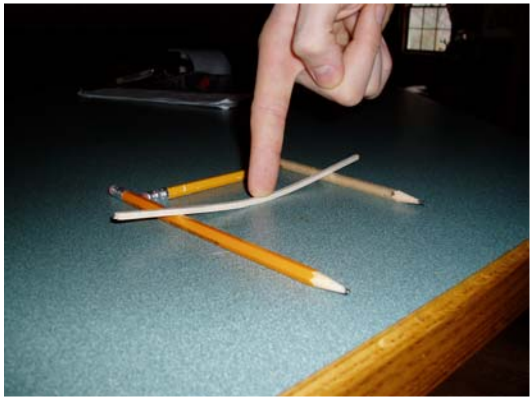
The simulation is ready for post-processing.

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.



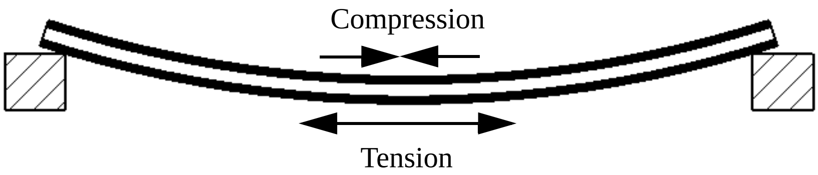
The **Displacement** is the movement of the beam from it's original position. The "worst case" displacement occurs when the load is at the center of the beam.

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

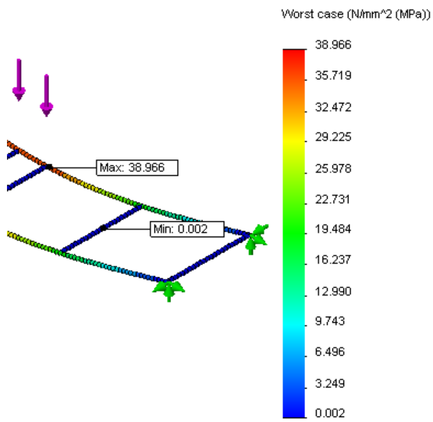
While the beam bends, the top portion of the beam (the face where the load is applied) compresses (pushing together) while the opposite face sees tension (pulling apart).



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 design is ✔. If the **FOS < 1**, the design is failing.

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.

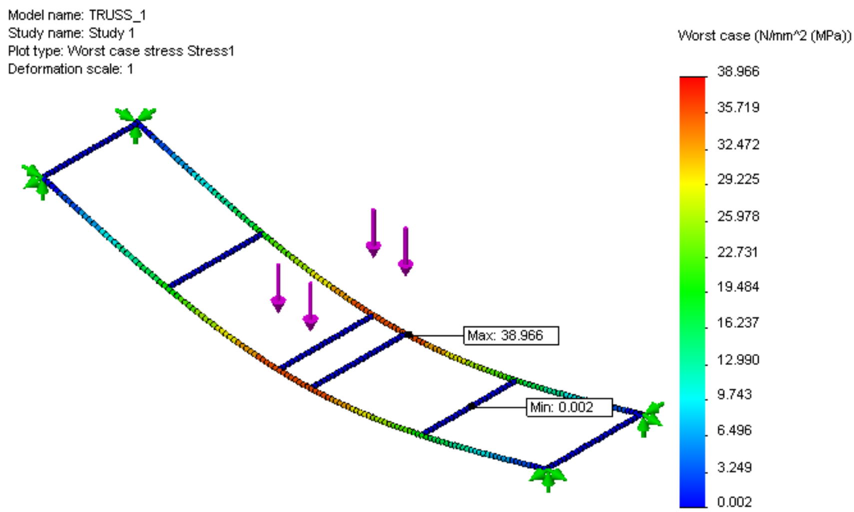
Post-Processing

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.

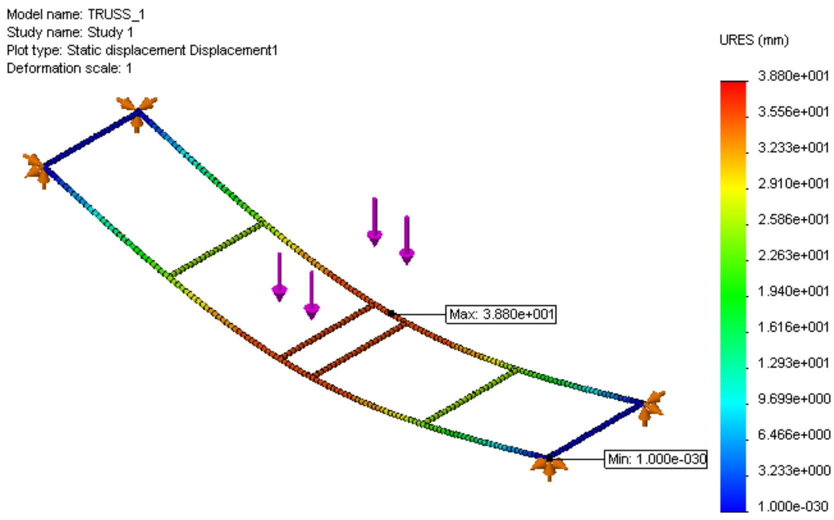
As post-processing begins, two plots are posted in the `Results` folder: `Stress1 (-STRMAX: Worst case-)` and `Displacement1 (-Res disp-)`.

The stress plot is selected and viewed automatically.

13 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.



14 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. Is 20MPa too much? What is a MPa? Lets get some sense of what the results mean. Here are results so far (*yours may vary*):

Stress	**Displacement**
38.966 MPa (Megapascals)	3.880e+001mm

Numbers

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

3.880e+001 means 3.880 X 10^1 or 3.880 X 10^1= 3.880 X 10 =______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 = 4.136854e+005 Pa = 0.4136854 MPa (1MPa = 1N/mm^2=1,000,000 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.

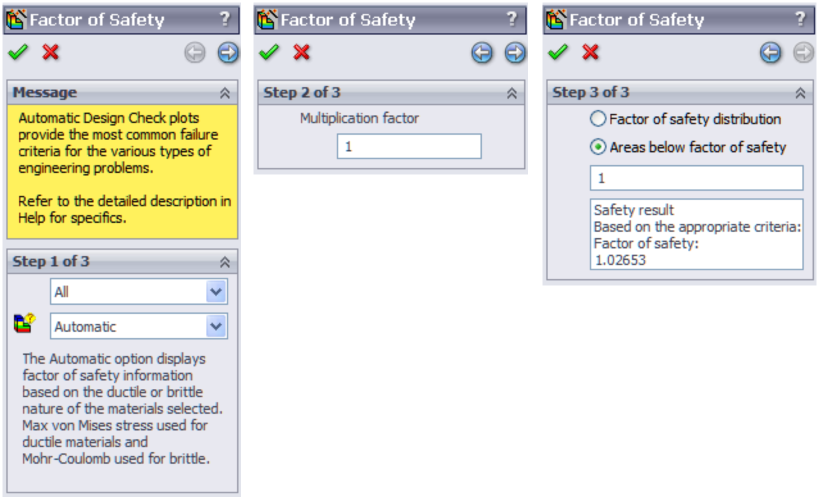
15 Factor of Safety Plot.

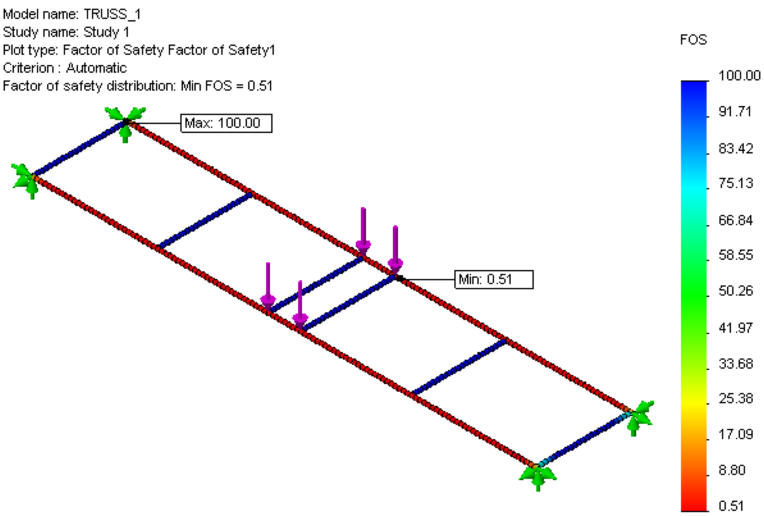
Right-click on the Results folder in the Simulation Study Tree and select **Define Factor of Safety Plot.**

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

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

Click **Areas below factor of safety** and click .





What does the Factor of Safety Plot Tell Us?

The minimum FOS is displayed as **0.51** 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 increase the load, we need to decide how much of an increase is required. The current information tells us that the FOS is about **0.5** for a load of 4 X **10N** = **40N**.

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

FOS X Total Load = **0.5** X **40N** = **20N** 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.

16 Edit external load.
Right-click the feature `Force-1 (:Per item: -10 N:)` and select **Edit Definition**. Set the load to **5N** and click ✓.

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

18 Close the part.
Click **File, Close** and click **Yes** 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.

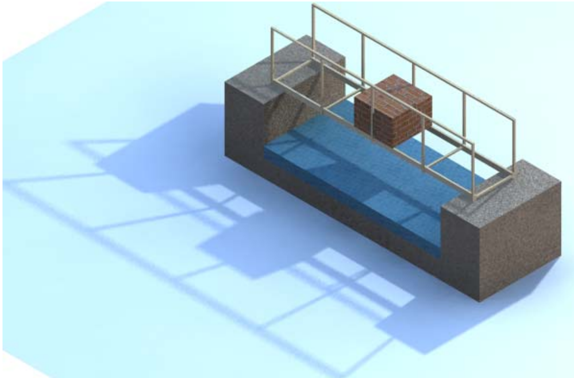
Lesson 5
Making Design Changes

When you complete this lesson, you will be able to:

- 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

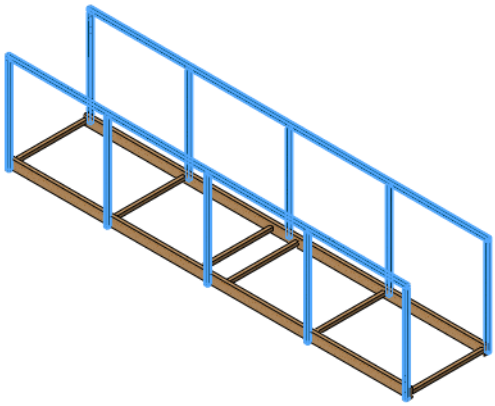
1 Open the part file.

Click **Open** .

From the **Open** window, browse to the `Lesson 5` folder.

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

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

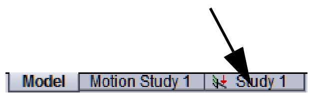


Existing Study

This part is the same as the previous one with the addition of the walls. It has 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.

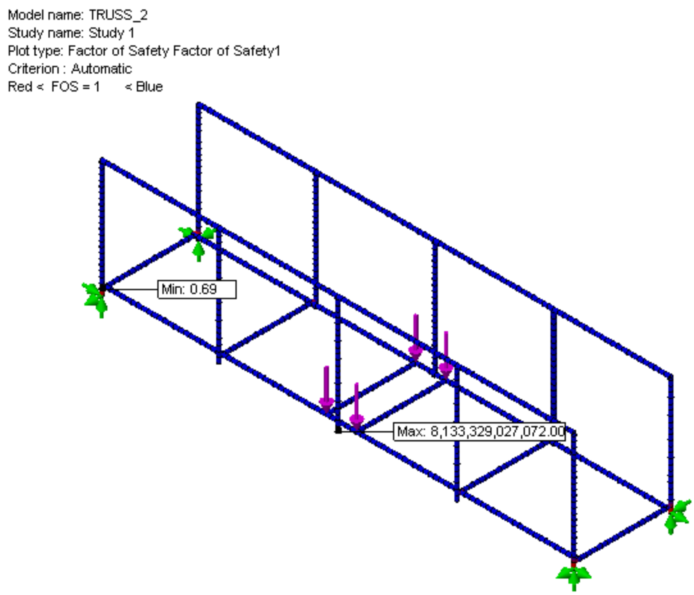


3 Run the analysis.

Click **Simulation, Run**.

The simulation is ready for post-processing. 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 49.



The results show that the FOS has been increased slightly so that the value is below **1**.

Why is it stronger?

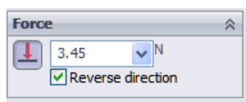
The addition of the walls increases the load that the structure can take. The members that make up the wall absorb some of the external loads and the stress, reducing the stress on the lower portion of the structure.

Increase the Load

Since the FOS has been increased, let’s increase and 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.

20N x 0.69 = 13.8N each load is 13.8N/4 = **3.45N**

5 Edit external load.
Right-click the feature `Force-1 (:Per item: -10 N:)` and select **Edit Definition**. Set the load to **3.45N** and click .

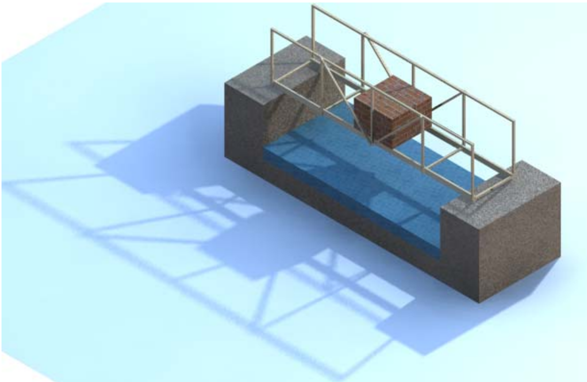


6 Rerun.
Click **Simulation, Run** to rerun the analysis. The minimum FOS should again be very close to **1**.

7 Close the part.
Click **File, Close** and click **Yes** to save changes.

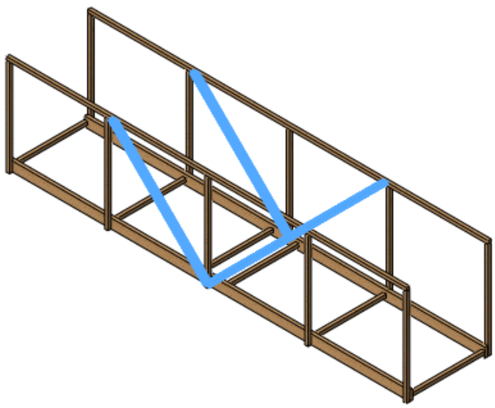
Cross Bracing

In a previous lesson, the value of triangles and cross bracing was discussed (“Triangles” on page 14). 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 (**3.45N** 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 5` folder.

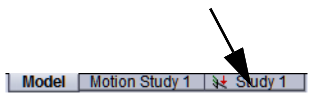
Select `TRUSS_3.sldprt` and click **Open**.

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 the walls. It has 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. The factor of safety plot is not created automatically.

4 Factor of Safety Plot.
Use the same procedure as “Creating a New Plot” on page 49. Have the braces increased the strength of the structure? What is the current FOS?________

How much should we increase the load to get a FOS of 1?

_________ x 3.45N = _________

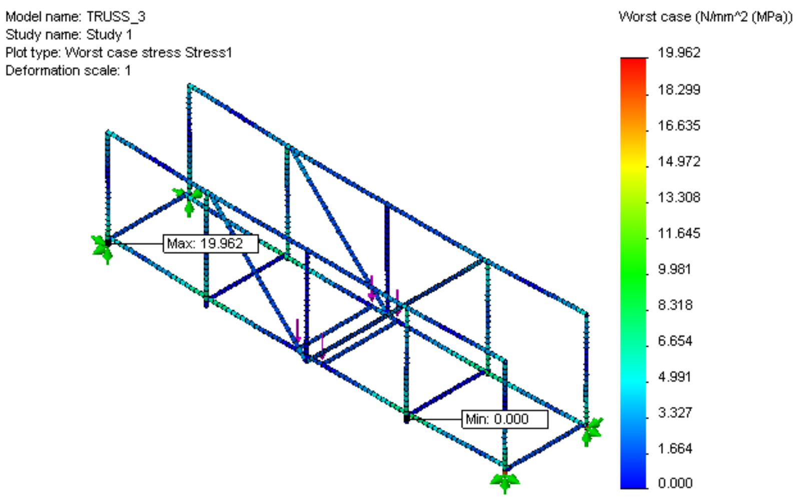
5 Edit external load.
Right-click the feature `Force-1 (:Per item: -3.45 N:)` and select **Edit Definition**. Set the load to **5.55N** and click .

6 Rerun.
Click **Simulation, Run** to rerun the analysis. The minimum FOS should again be very 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.

7 Stress plot.
Double-click the `Stress1 (-STRMAX: Worst case-)` plot to see the stress plot.



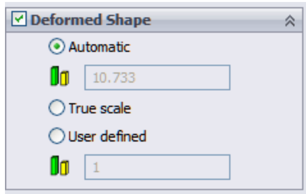
Working with Plots

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.

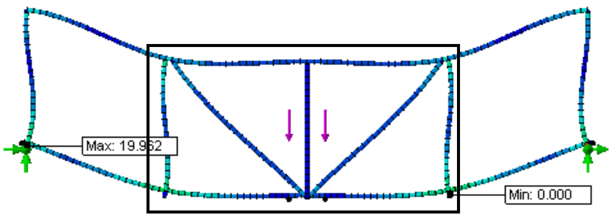
Deformation Plot Factor

The deformed shape of the stress plot uses the actual displacement, which is fairly small. To exaggerate the displacement, you can set the deformed shape to an **Automatic** or **User defined** value of your choice.

8 Deformed shape.
Right-click the `Stress1 (-STRMAX: Worst case-)` plot and select **Edit Definition**. Under **Deformed Shape**, click **Automatic**. Click ✓.



9 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 it's shape better than the end portions due to the strength added by the bracing.

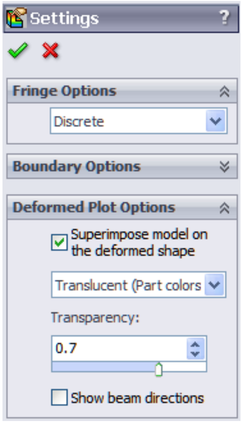


Superimposing the Model

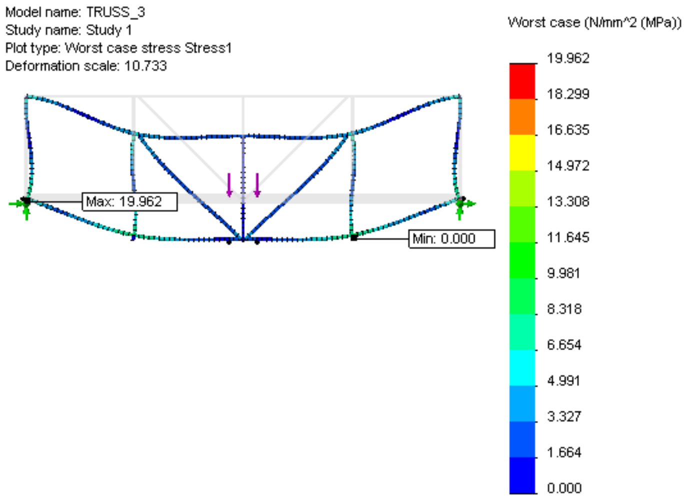
The settings options allow you to superimpose the undeformed shape and change the appearance of the chart to have show distinct color changes.

10 Settings.

Right-click the `Stress1 (-STRMAX: Worst case-)` plot and select **Settings**.



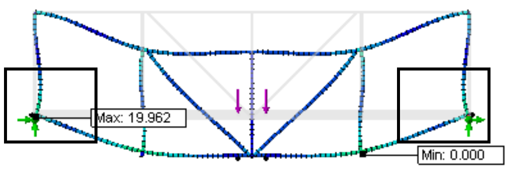
Under **Fringe Options**, select **Discrete**. Under **Deformed Plot Options**, click **Superimpose model on the deformed shape** and set the **Transparency** to **0.7**. Click ✓.



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

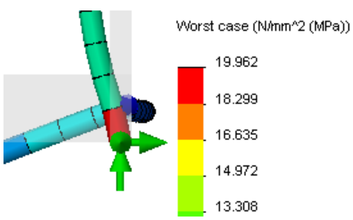
The Weakest Link

Are you are 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.



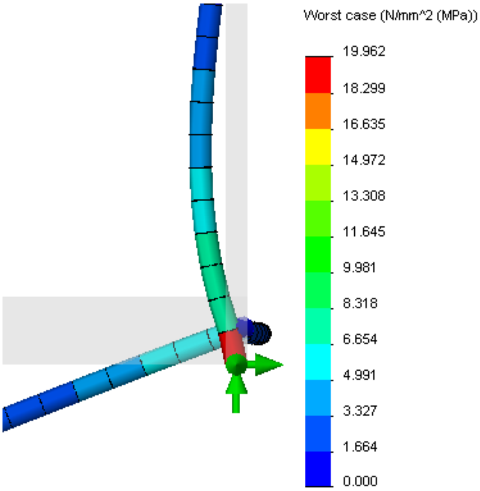
If you look closely at the lower left section of the image, you will see the label or the highest stress value. This is the weakest link, a *high stress* area.

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 yellow/red/orange, the lowest in shades of blue.



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

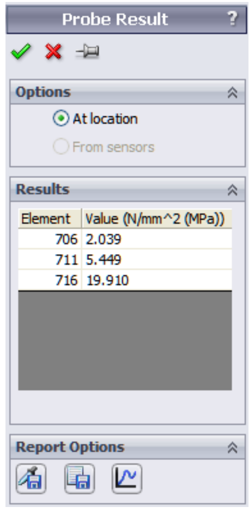
Keep in mind that the highest stress may not cause the structure to fail. Follow the **Yield strength** arrow, that is the failure point.

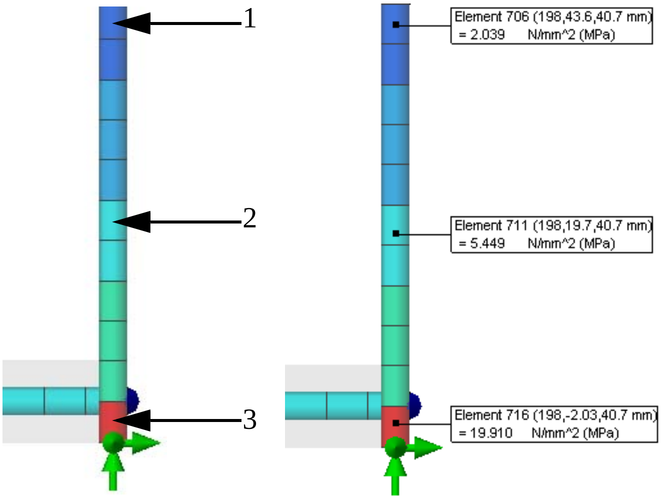
Using a Probe

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.

11 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.



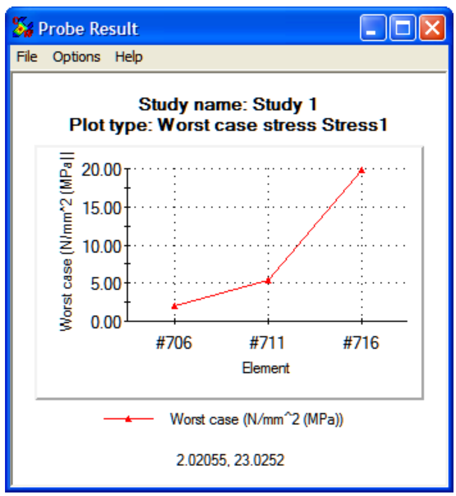


Note: Make selections similar to those shown here. The labels that you see may have slightly different values.

12 Plot.

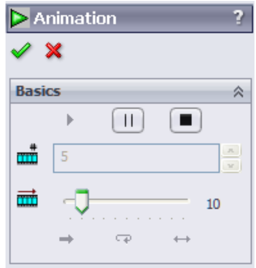
Click **Plot** to create the **Probe Result**. The change in the stress value across those few elements is dramatic as demonstrated by the plot.

Click **File, Close**. Click to close the **Probe Result** Property Manager.



13 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.

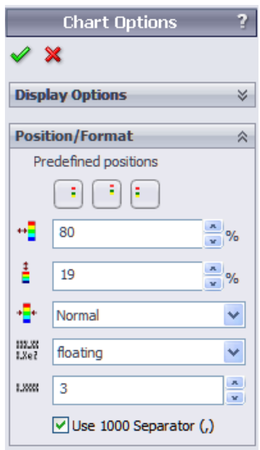
Scientific	Floating	General
3.727e+000	3.727	3.73

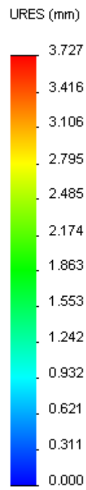
14 Displacement.

Double-click the `Displacement1 (-Res disp-)` plot. Displacement numbers tend to be small, and in this chart they run between 0 and about 4mm. They are in scientific notation but would be easier to read in a decimal format.

15 Chart options.

Right-click he `Displacement1 (-Res disp-)` plot and select chart options. Under **Position/ Format**, select **Number Format floating**.





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?

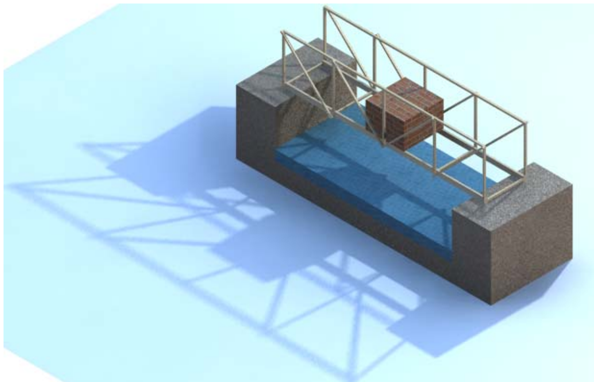
1. Increase the load to increase the FOS to a value greater than 1.
2. Add cross bracing to the unbraced sections.

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

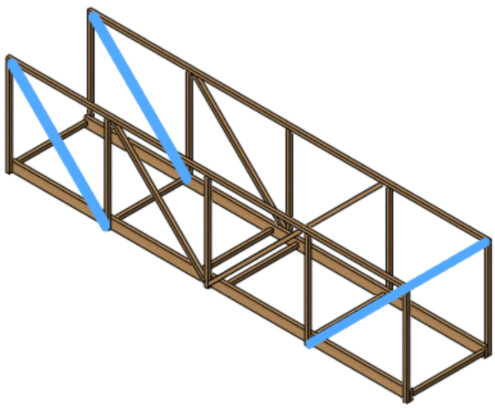
16 Close the part.
Click **File, Close** and click **Yes** to save changes.

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 Structure directory.

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 Study 1 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 from about **20MPa** in the previous model to about **3MPa** in this one.

The stress has been reduced to 3/20 x 100 = ______% of the previous value.

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

3 **Factor of safety plot.**
Create a factor of safety plot. Use the same procedure as "Creating a New Plot" on page 49.

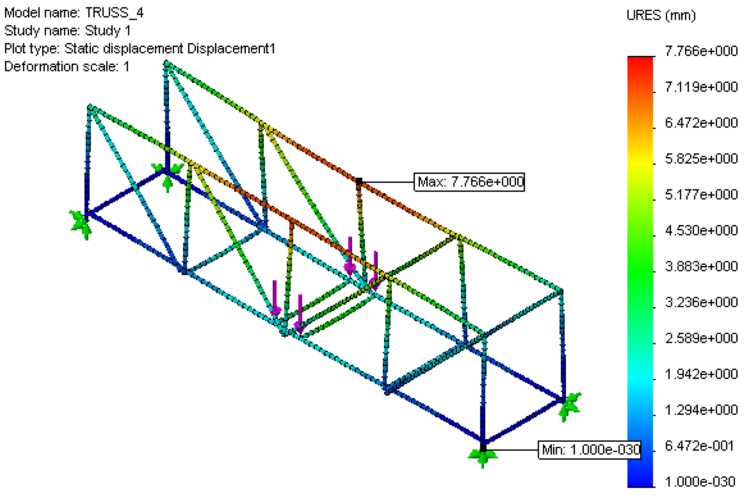
4 **Maximize the external load.**
Again, we will maximize the load for a factor of safety of **1**.

7.57 x 5.55N = ________ N

Edit the external load `Force-1 (:Per item: -5.55N:)` and set it **42N**.

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

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

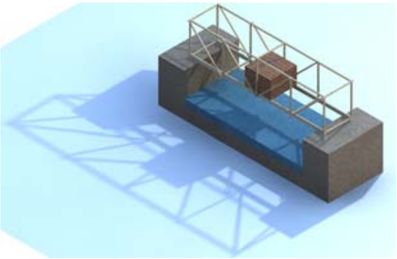


The displacements are small but you may notice that the model seems oddly shaped. The upper portions of the walls are bending inward. Some additional bracing is required.

7 **Close the part.**
Click **File, Close** and click **Yes** 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.



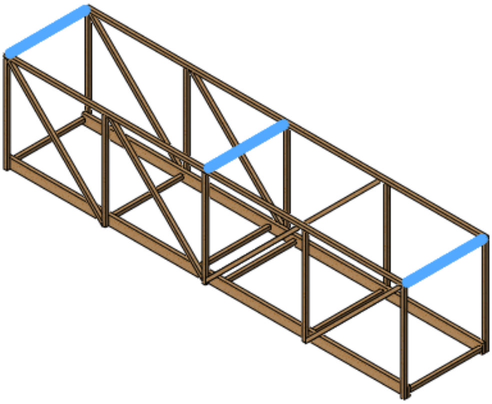
1 Open the part file.

Click **Open** .

From the **Open** window, browse to the `Structure` directory.

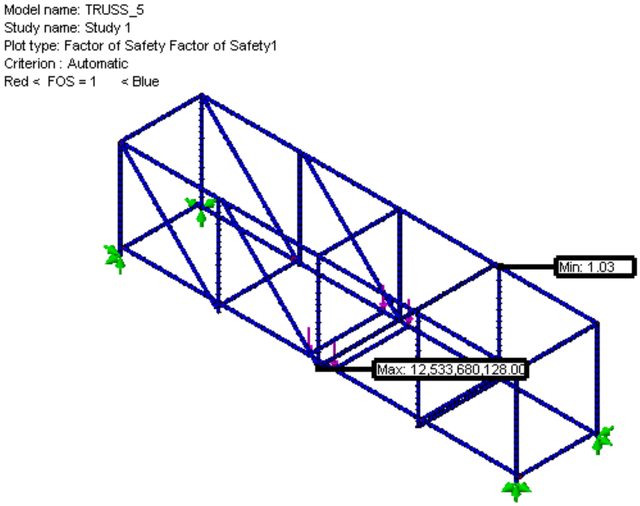
Select `TRUSS_5.sldprt`, and click **Open**.

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

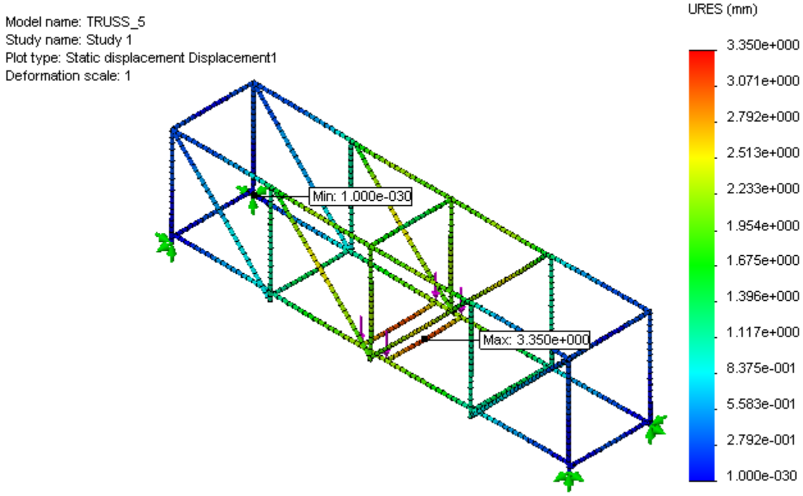


2 Maximize load.

Run the analysis and create a FOS plot using the same procedure as “Creating a New Plot” on page 49. The FOS should fall just under **1**. To increase the FOS a little, edit the external load to **40N** and rerun.



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



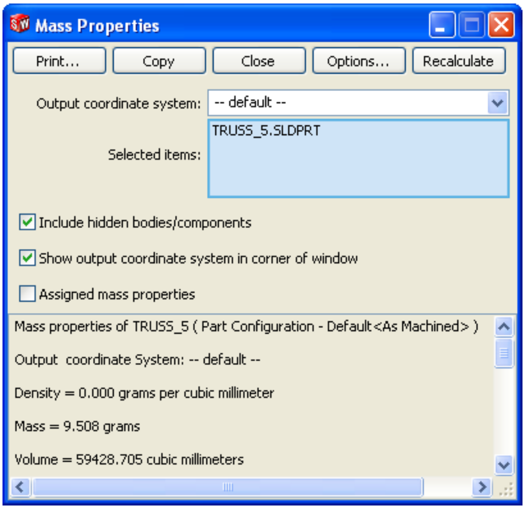
Strength to Weight Ratio

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.

4 Mass properties.
Click **Tools, 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?

Track the results of the last analysis using the same **40N** total load:

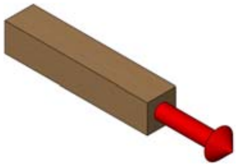
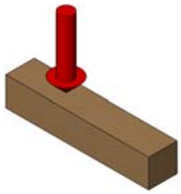
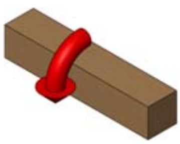
Structure	Max Load	Weight of Structure	Efficiency (Max Load/ Weight)
TRUSS_1	20N	4.566 g = ______N	________
TRUSS_2	13.8N	7.418 g =______N	________
TRUSS_3	22.2N	8.266 g =______N	________
TRUSS_4	40N	9.130 g =______N	________
TRUSS_5	40N	9.508 g =______N	________

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

5 Close the part.
Click **File, Close** and click **Yes** to save changes.

More to Explore

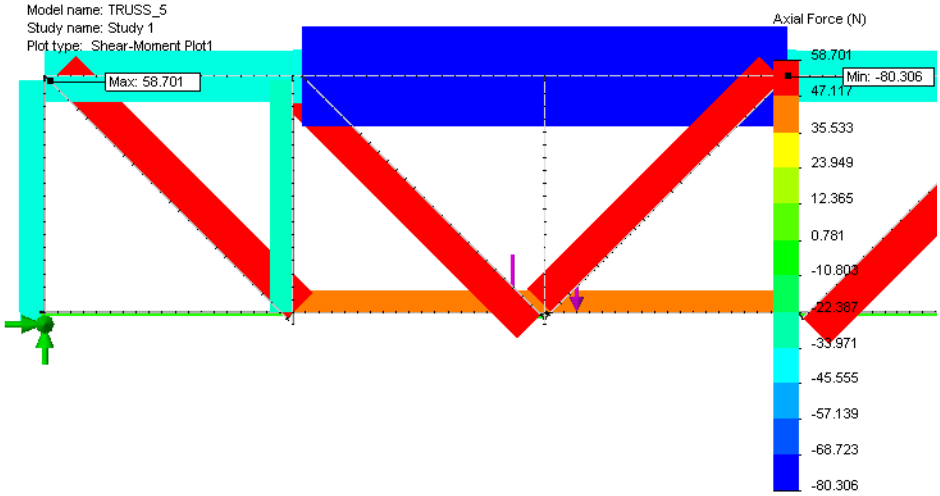
Each simulation can have multiple plots to display the results in different ways, but the beam analysis has a unique type, the **Beam Diagram**. This plot can be used to display several quantities directly on the beams. The forces and shears are displayed in Newtons (**N**), the moments and torque in 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 red, meaning that the value is between **47N** and **53N**. The braces are in tension, because their axial force values are positive.



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

Lesson 6
Using an Assembly

When you complete this lesson, you will be able to:

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

Creating 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

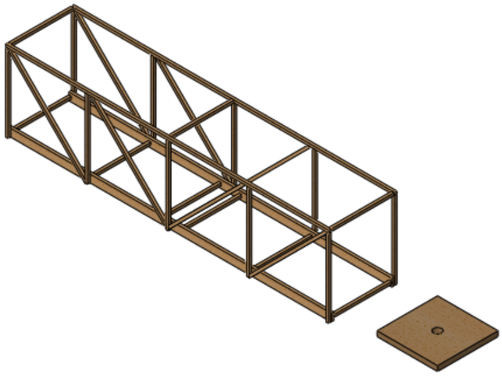
If you were to build and test this structure, it would have to meet certain criteria for length, width and height. One of the criteria would be a test to see if a wooden block of a certain size and length could fit through.

1 Open the assembly file.

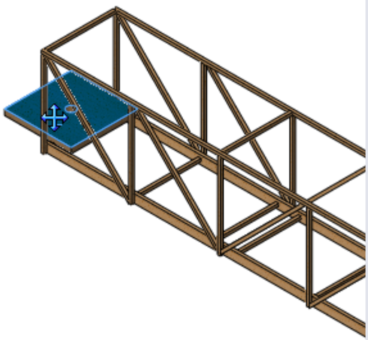
Click **Open**.

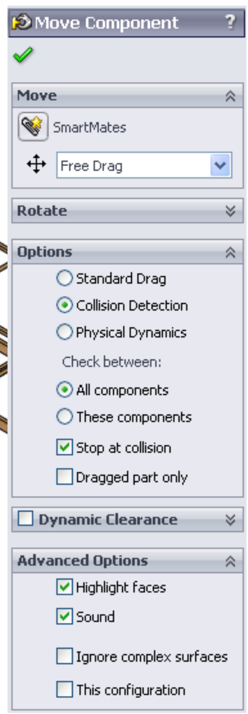
From the **Open** window, browse to the Lesson 6 folder. Select Test_Block_Assembly.sldasm, and click **Open**.

The assembly includes a copy of the previous structure and a representation of a wood block.



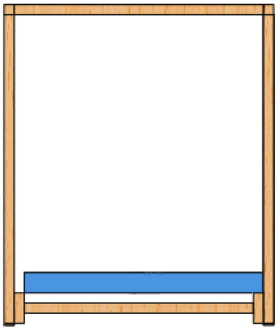
2 Move component.
Select the Load_Plate_75 component and click **Tools, Component, Move** from the Assembly toolbar. In the dialog, click **Collision Detection**, **All components**, **Highlight faces** and **Sound**. Select and drag the Load_Plate_75 through the structure. It should move smoothly through 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.



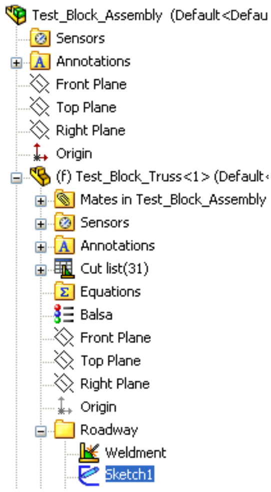
Changing the Model

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

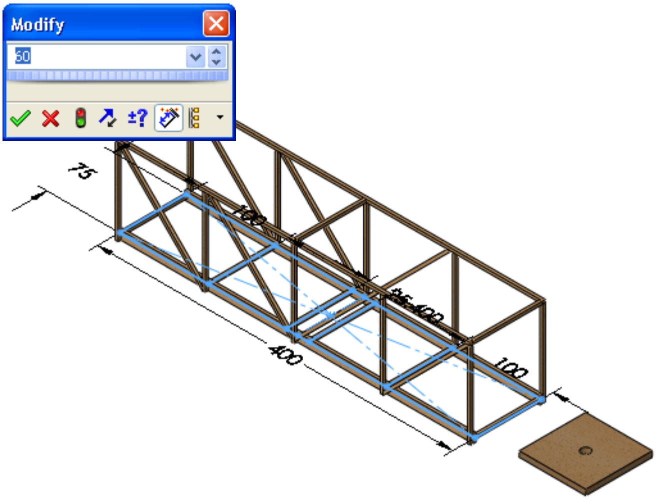
4 Expand features.

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

Double-click the Sketch1 feature.

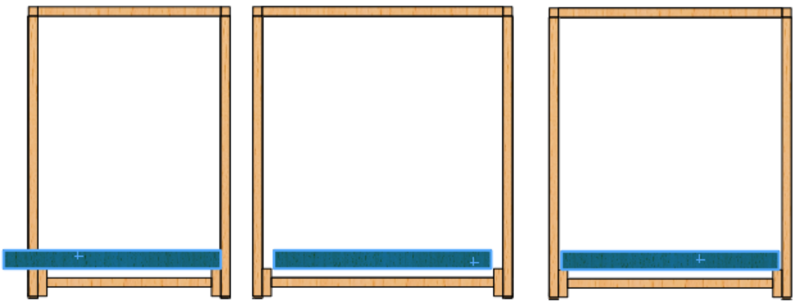


5 Change dimension.
Double-click the **75** dimension and change it **60**. Click **Rebuild** and . The structure part changes size.



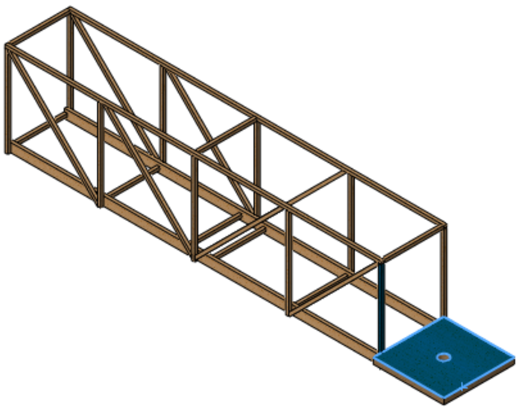
Collision Detection

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.

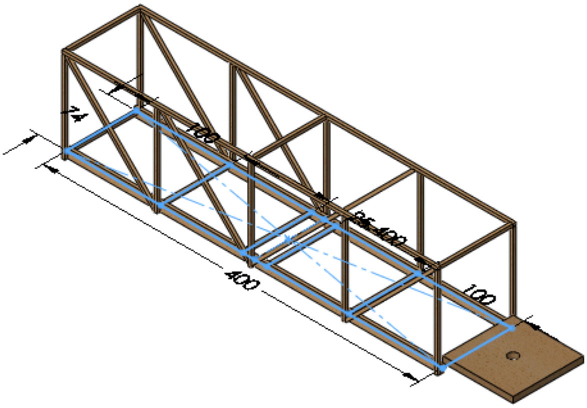


Collision | Too Wide | Proper Clearance

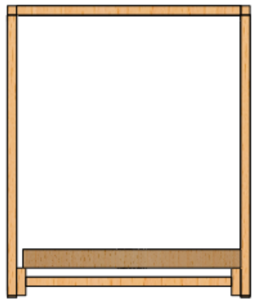
6 Move.
Using the same **Move** procedure as before (step **2**), try to move the block through the structure. It collides with the structure.



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.



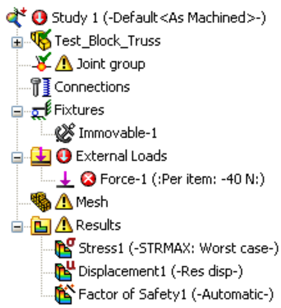
Note: The plate is shown centered in this image.

9 Open the part.
Right-click on the `Test_Block_Truss` in the FeatureManager and select **Open Part**. The structure part opens in it's own window.

Updating the Analysis

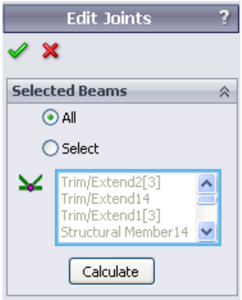
The model has changed and has actually become narrower. The model change will cause several errors in the joints which in turn cause errors in the fixtures, loads and mesh.

10 Warnings and errors.
Click the study `Study 1`. There are warning and error markers on several features.



11 Joint group.

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

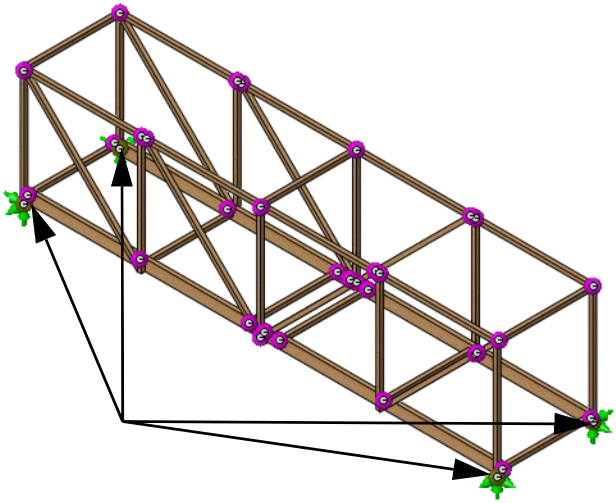


Click **Calculate** and ✔.

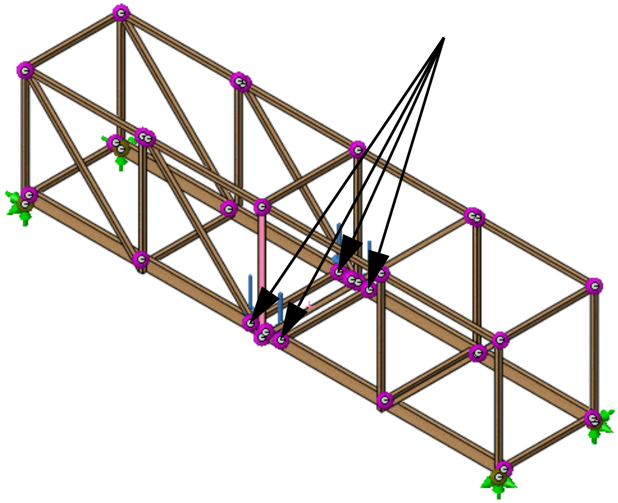
The message says Joints are recalculated. Calculated joints may look same, but the order may be different. Re-definition of fixture/load/connection may be required. Click **OK**.

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: -40 N:) and select **Edit Definition**. Click in the selection field and select the same four joints as shown and click ✓.



14 Mesh and run.
Right-click the Mesh feature and select **Mesh and Run**. The changes to the factor of safety are insignificant. It remains near the target of **1**.

15 Close the assembly.
Click **File, Close** and when the message "**Save changes to Test_Block_Assembly?**" appears, click **Yes**.

Lesson 7
Making Drawings of the Structure

When you complete this lesson, you will be able to:

- 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:

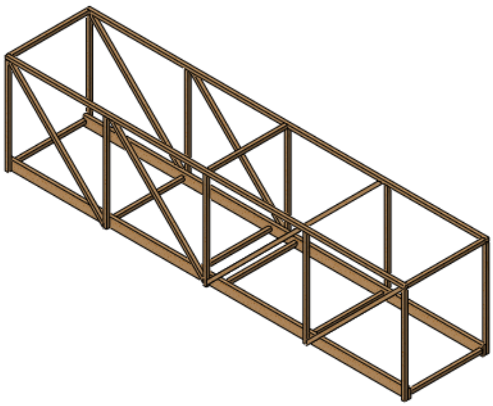
- **Shape** – *Views* communicate the shape of an object.
- **Size** – *Dimensions* communicate the size of an object.
- **Other information** – *Notes* communicate nongraphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs and so forth.

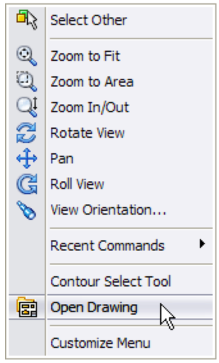
Creating a Drawing and Views

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 `Lesson 7` folder.

The part is a completed model of the structure.



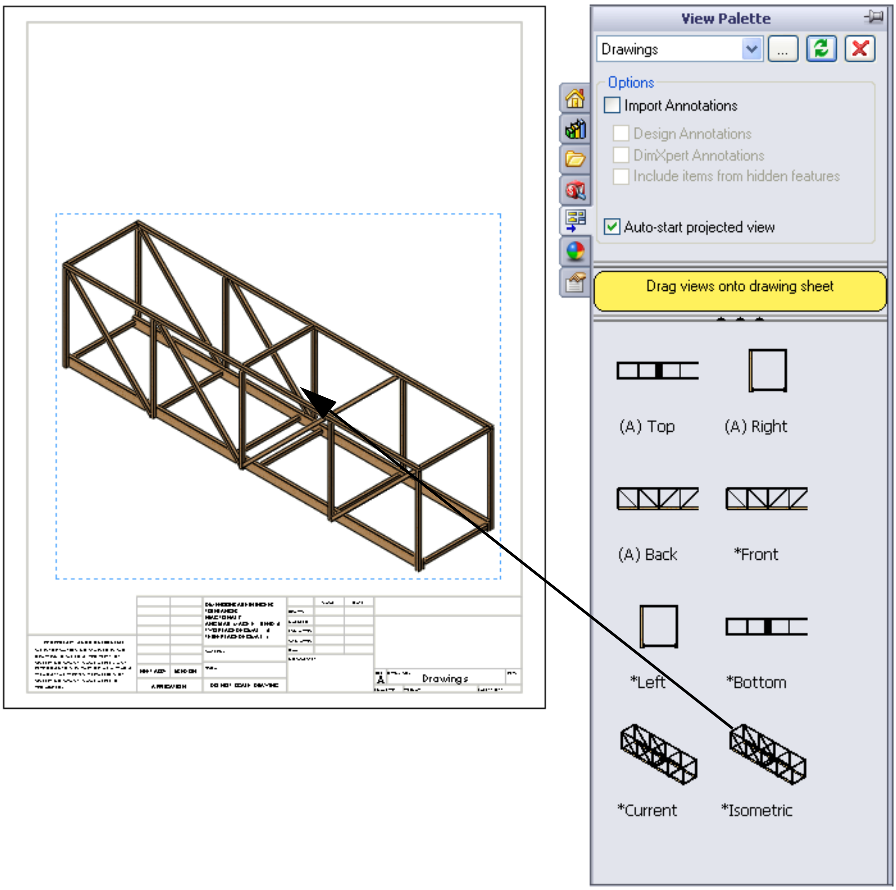


2 Open drawing file.

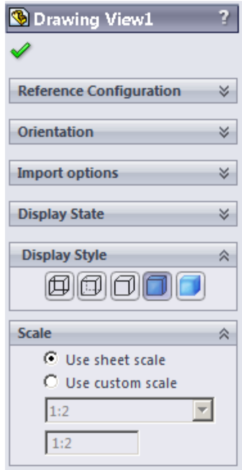
This part has an associated drawing file. It has no drawing views or notes, but it contains many of the setting that we need. Top open it, right-click in the graphics area and select **Open Drawing**.

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.

5 Weldment Cut List.

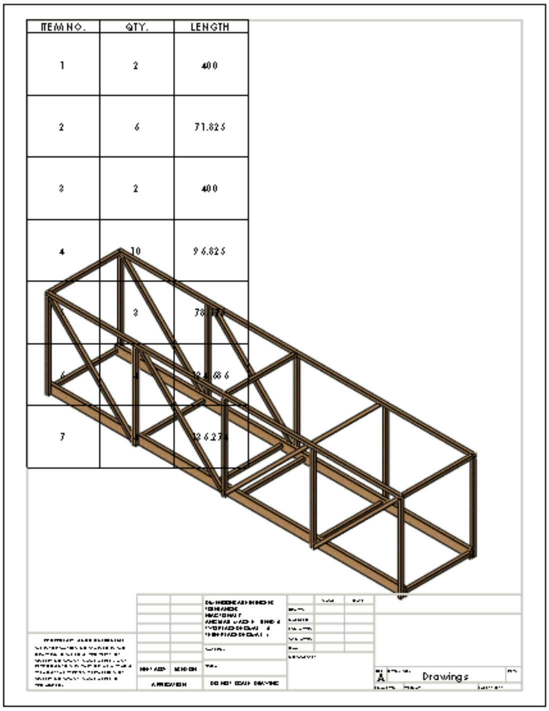
Click **Insert, Tables, Weldment Cut List** and select the drawing view.

Select the file `Bridge_Weldments.sldwldtbt` from the same folder as the part as the **Table Template**.

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.

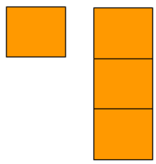


7 Resize columns.

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

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 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.

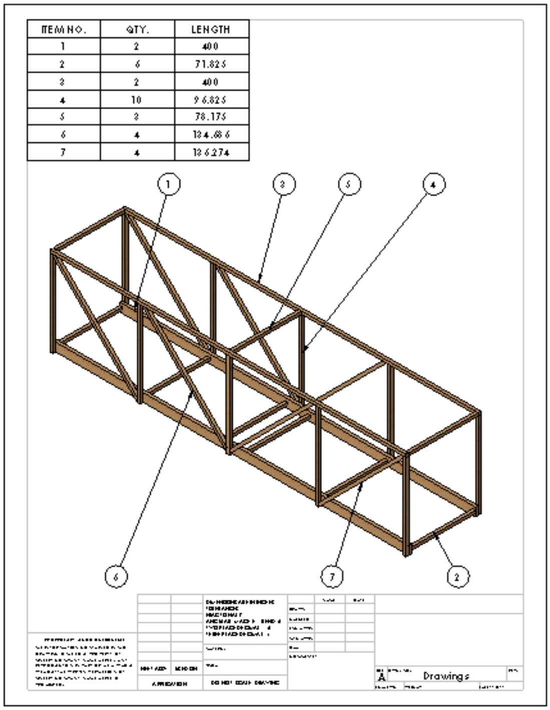
Tip: In the building section the actual sizes of the individual beams will be listed.

Balloons

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

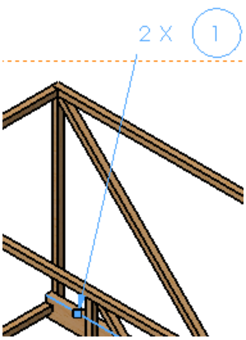
8 Balloons.

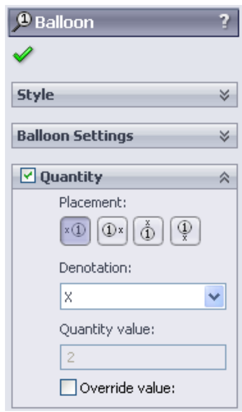
Select **Balloon** from the Annotation toolbar. Click on the member and then click to place the text. Repeat the process to add some balloons. Click ✓.



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 click **Yes** to any questions.

Lesson 8
Reports and SolidWorks eDrawings®

When you complete this lesson, you will be able to:

- 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.

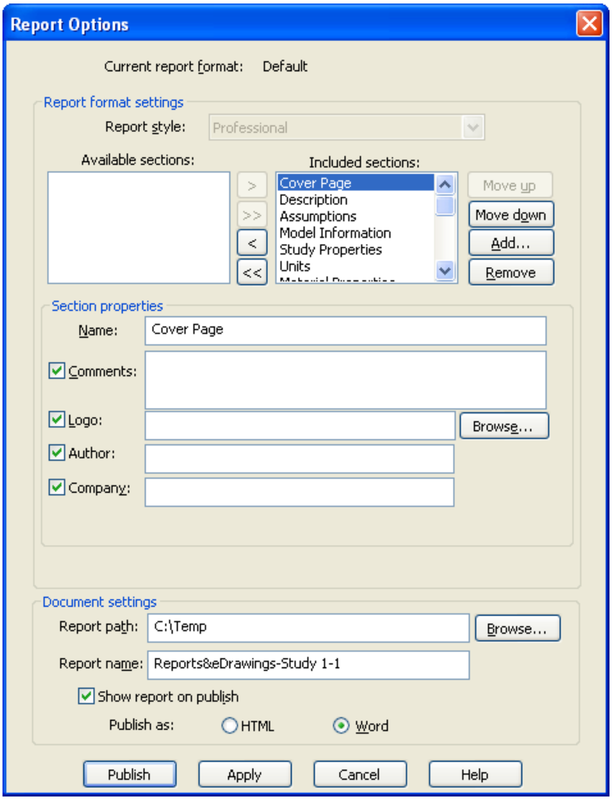
1 Open the part file `Reports&eDrawings`**.**
From the **Open** window, browse to the `Lesson 8` folder. Run the analysis.

Creating a Report

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

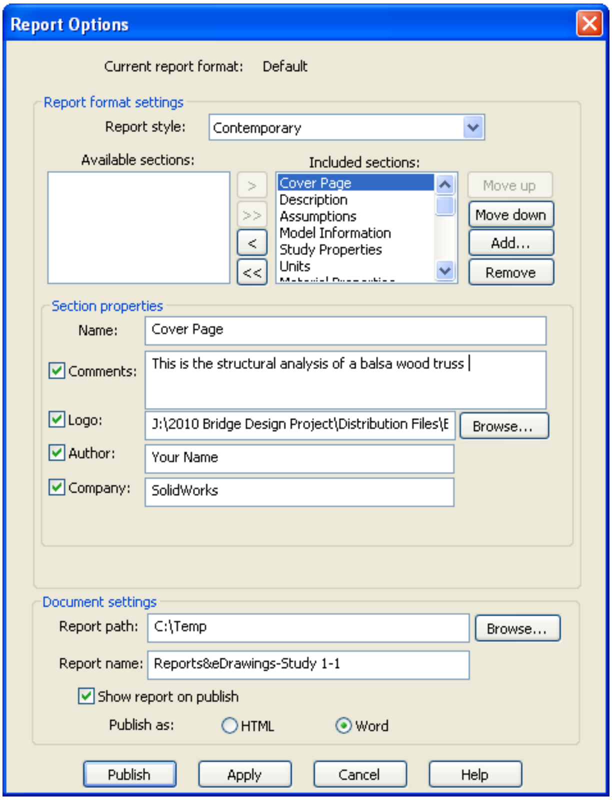
2 Report.
Click **Simulation, Report**.

3 Dialog.
Click **Comments, Logo, Author, Company, HTML** and **Word**.



4 Logo.
Click the **Browse** button and select the file logo.bmp from the Lesson 8 folder.

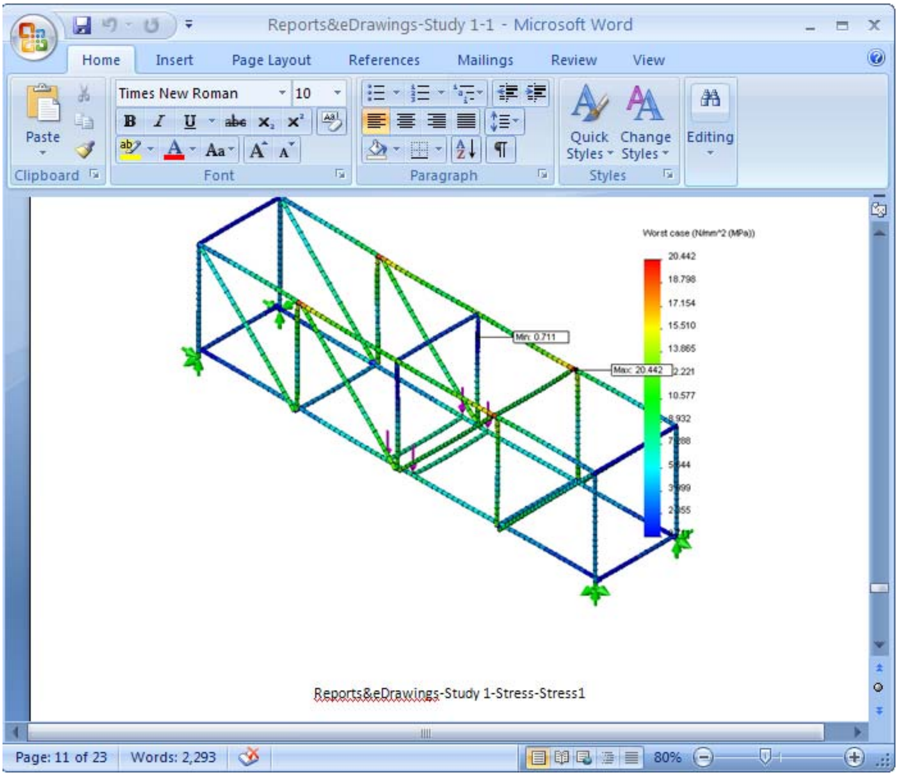
5 Comments.
Type This is the structural analysis of a balsa wood truss into the comments section and click **Publish**.



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. Click **Next**.



The results are placed in the `Structure Project\Analysis Results` folder. 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;
- Click  to 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 “animate” 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

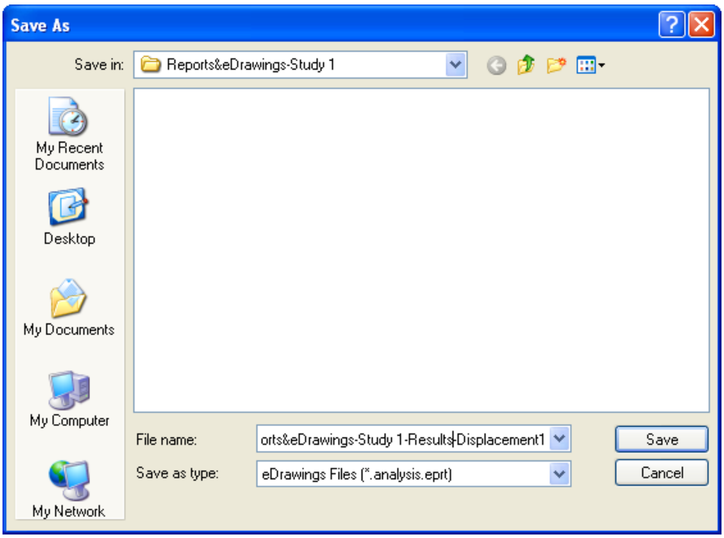
eDrawings are an easy way to share data, especially the image data that is generated by SolidWorks Simulation.

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**.

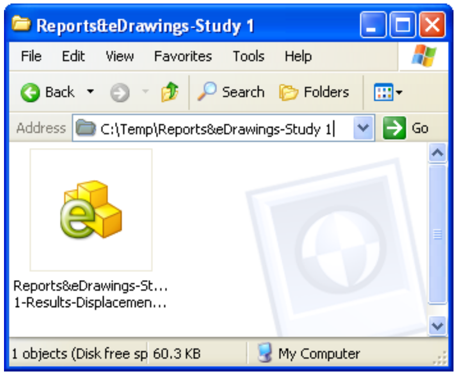
The default name is of the form:
part name-study name-results-plot type. In this case it is Reports&eDrawings-Study 1-Results-Displacement1.

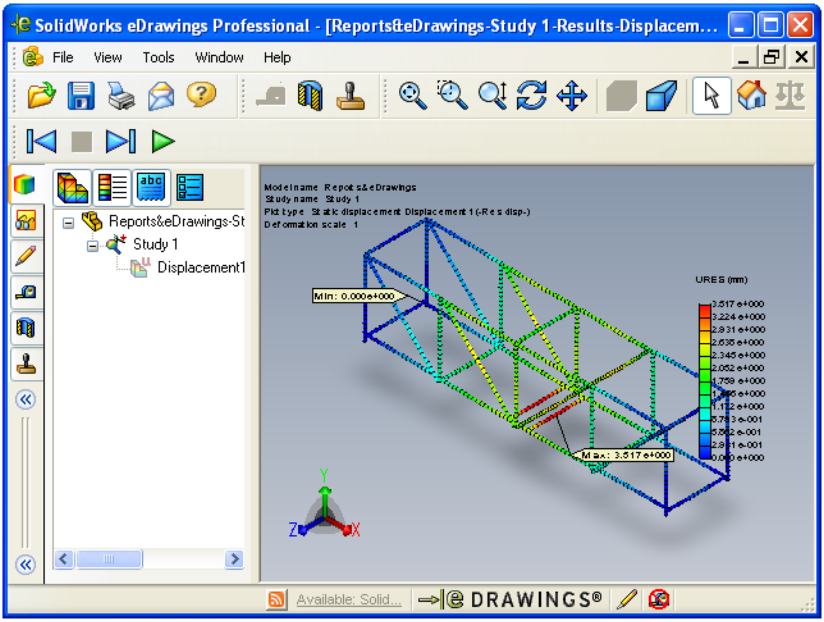
It is stored in the Reports&eDrawings-Study 1 folder that was created by the report.



9 Open the eDrawing.

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



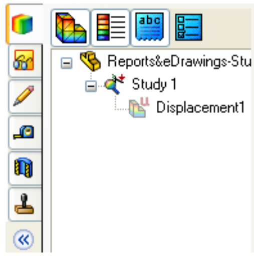


Note: If eDrawings has not been used before, it may ask permission to load.

The eDrawings User Interface

You have a choice of large [Play] or small buttons on the eDrawings toolbars. The large ones have text labels. To switch between large and small, click **View, Toolbars, Large Buttons**.

10 Settings.



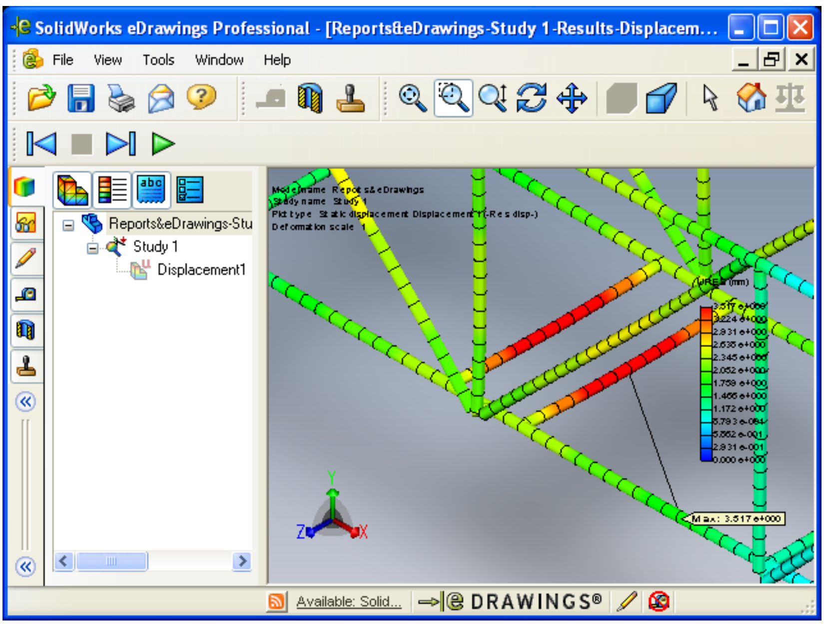
Click the **Analysis** tab, and the options **Show Mesh**, **Show Legend**, **Show Title** and **Simulation Options**.

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 **Home** .

Saving eDrawings

Click **Save** , or **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

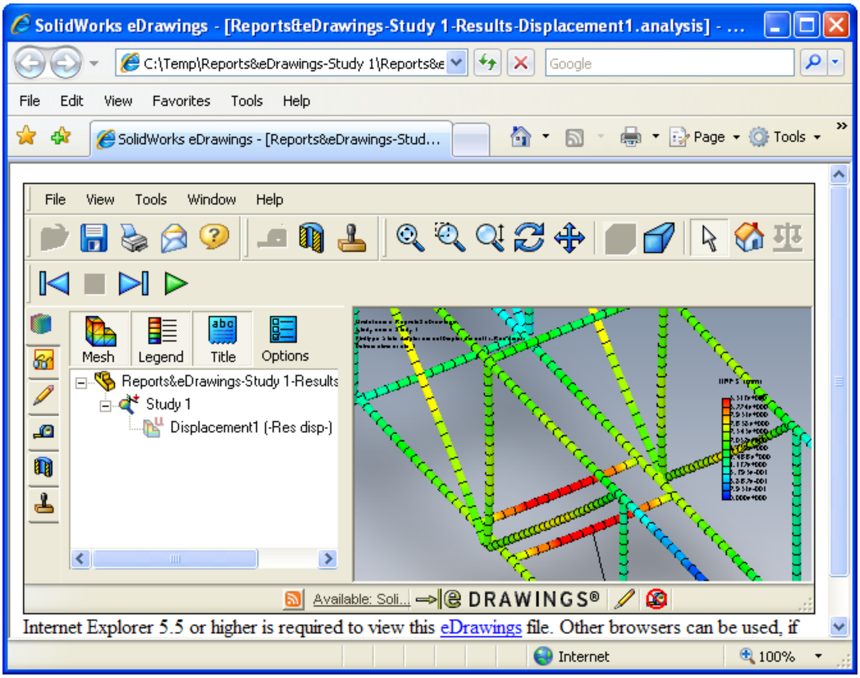
15 Save the eDrawing.

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-Study 1 folder.

16 Email the eDrawing to your teacher.

Open the HTML file. Click **Send**  and send the completed eDrawing to your teacher as an email attachment.

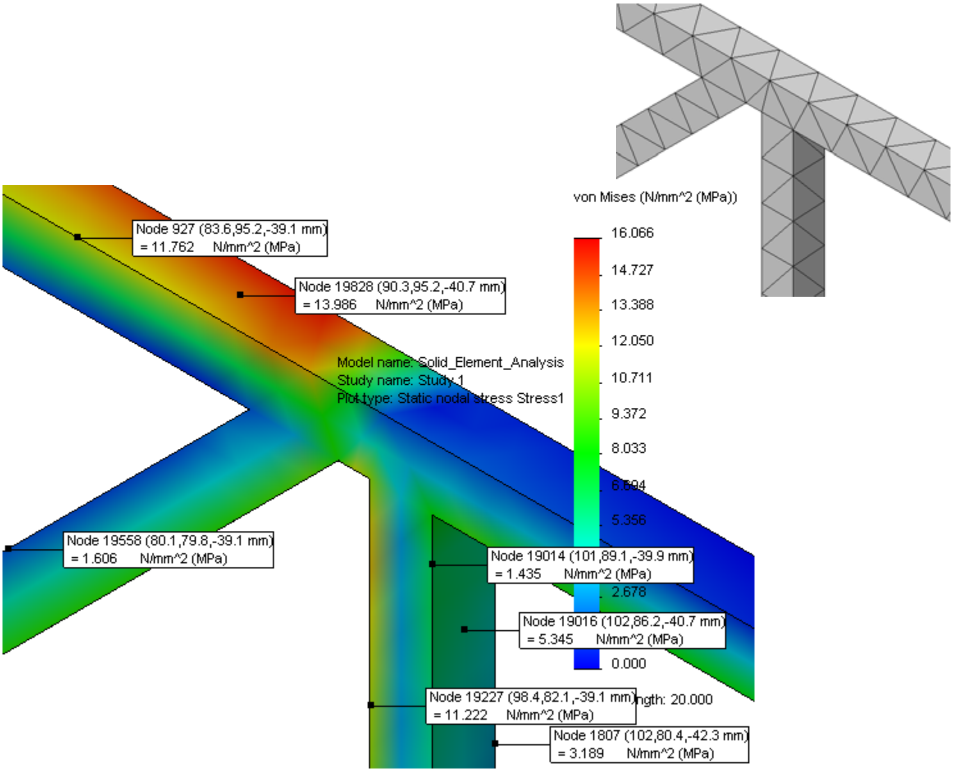


17 Close all files.

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 9
Building and Testing the Structure

When you complete this lesson, you will be able to:

- 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

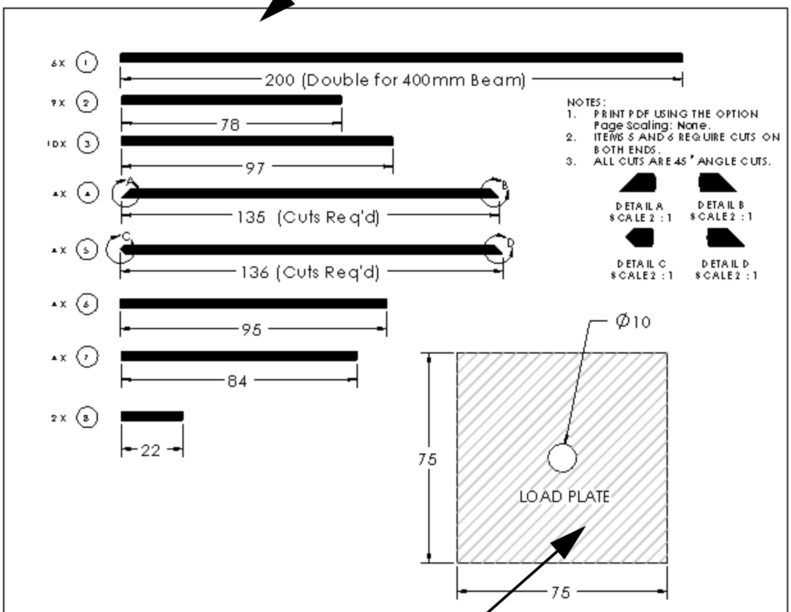
If your class chooses to build and test the structure, you will need 1/8” x 1/8” balsa wood sticks. Lengths of at least 24” or 400mm are required. Glue and a knife to cut the sticks are also required.

Cutting to Length

There are **43** members of **7** 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 PDF files.**
Browse to the _ENG folder, open the Measuring Chart.PDF file and print it using the precautions stated in the PDF and in the note below.

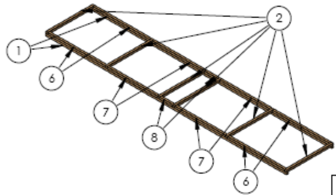
Measure and cut the members using this length chart. All dimensions are **mm**.

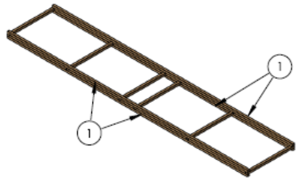


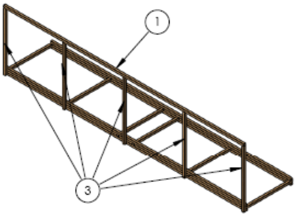
Cut out this Load Plate shape to test the inside width of your structure.

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

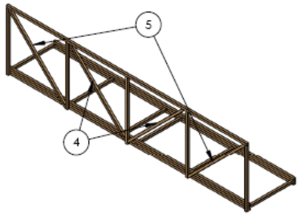
Open the `Construction Guide.PDF` file and print it.

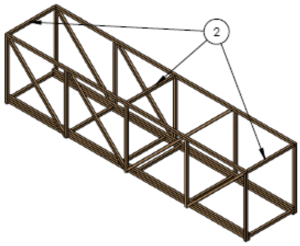




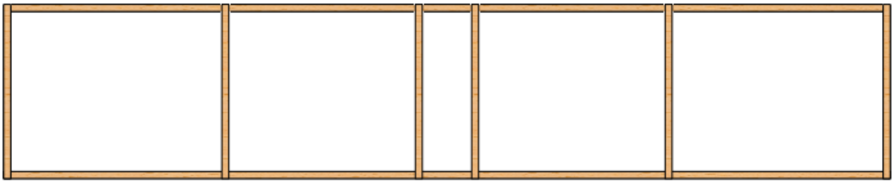


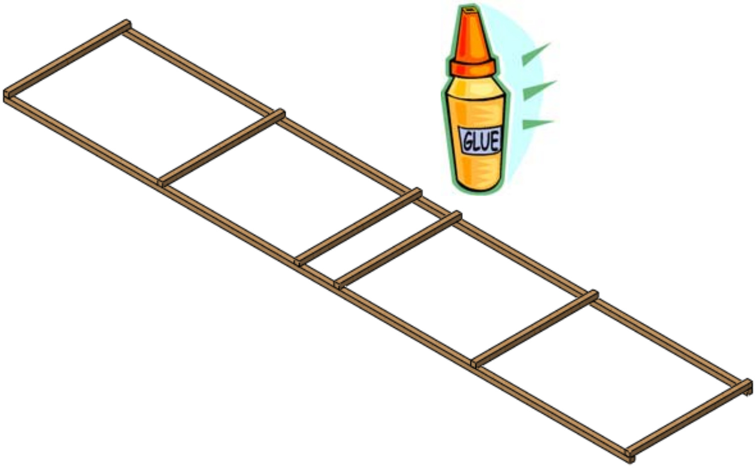
ITEM NO.	QTY.	LENGTH
1	6	400
2	9	78
3	10	97
4	4	135
5	4	136
6	4	95
7	4	84
8	2	22





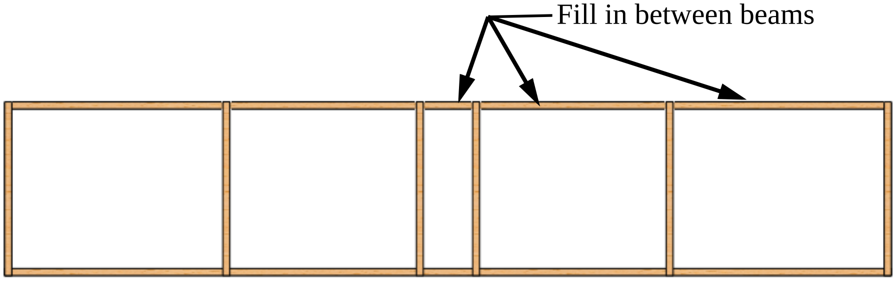
2 Lower frame.
Glue the end cross beams to the long beams. Do not glue the interior cross beams yet.

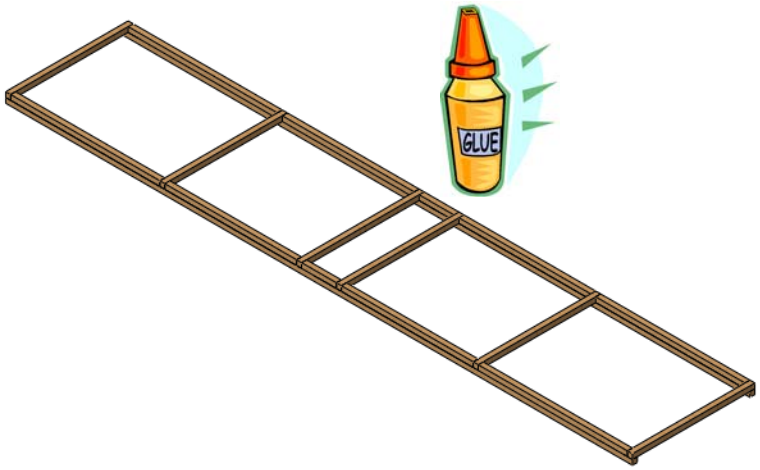




3 Fill in beams.

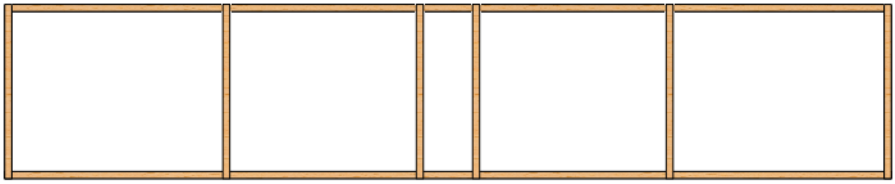
Fill in between the beams (cross hatched area) by cutting beams to fit and placing them in position. Glue all beams.

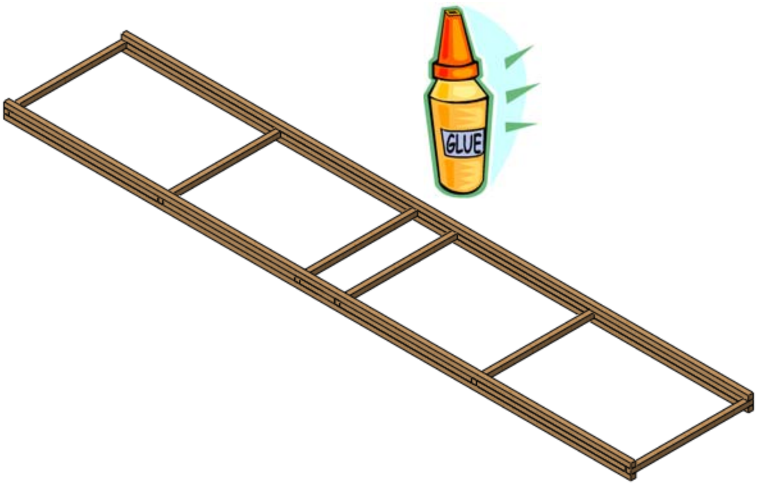




4 Triple outer rails.

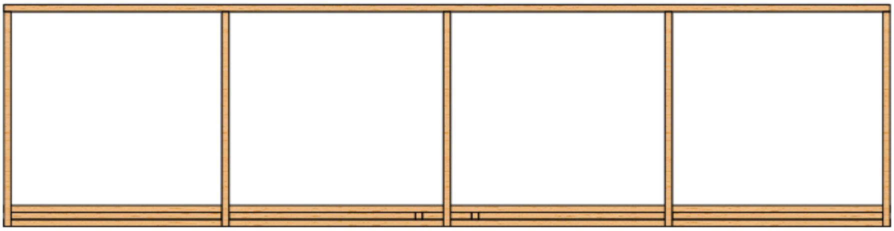
Glue the long beams over the fill in beams as shown. Glue all beams.

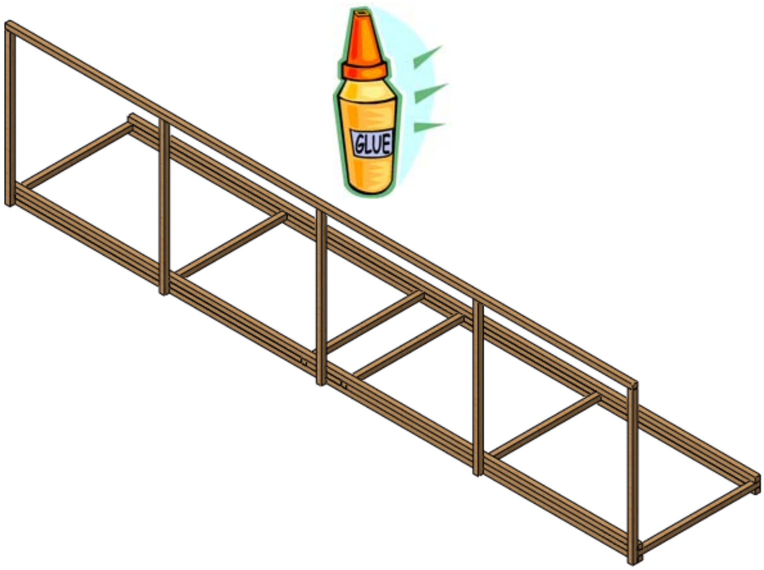




5 Side walls.

Glue all beams.

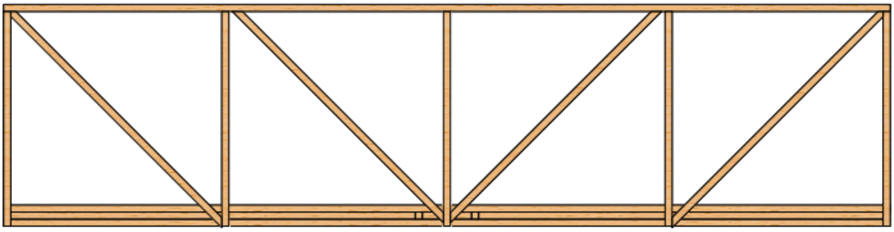


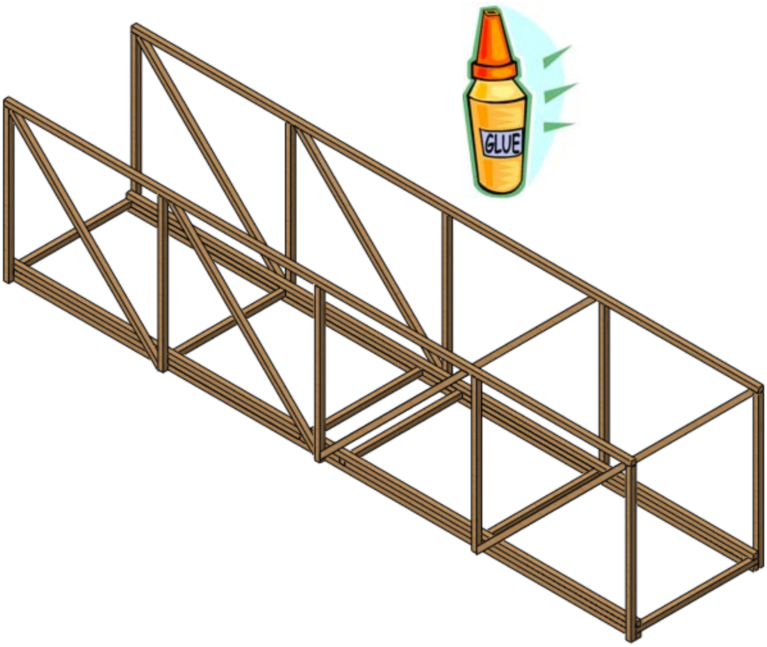


Tip: You might want to build a side and then cross brace it (step **6** on page 100) before starting on the opposite side.

6 Cross bracing.

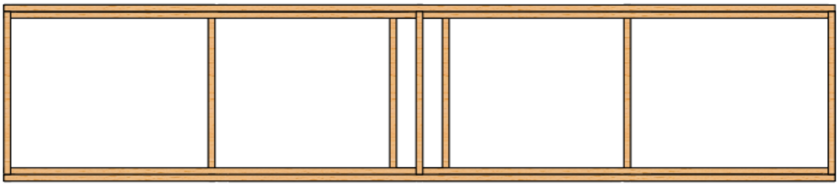
The brace members must all be cut (45 degrees) to fit within the existing framing. Glue all beams.

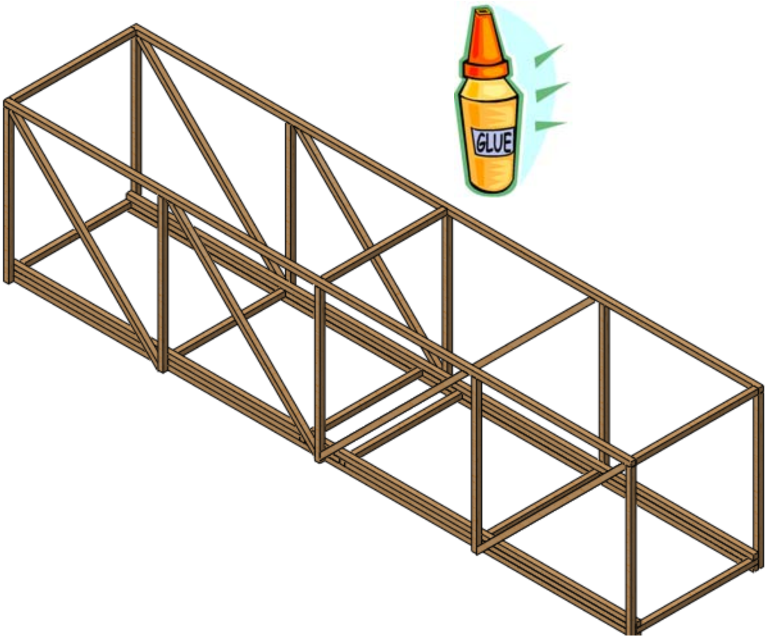




7 Top cross supports.

Glue all beams.





Testing the Structure

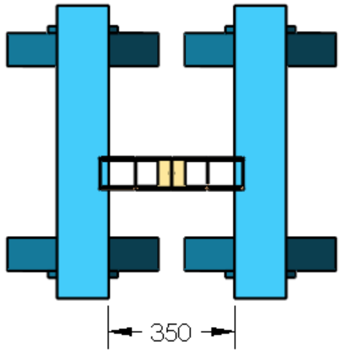
The structure can be tested by placing it across a gap and applying a load at the center of the bridge. See the following details for more information.

Creating the Span

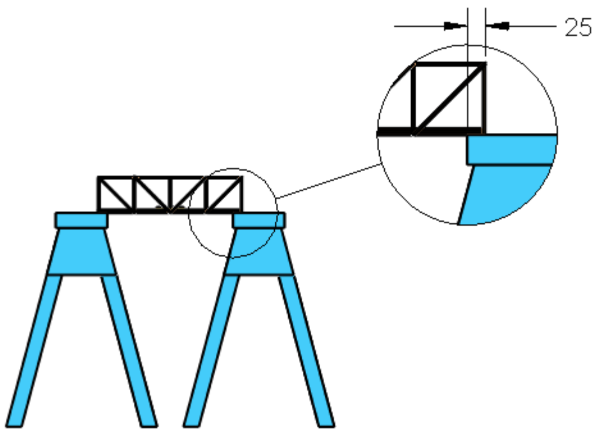
One way to create a span is to place two sawhorses a set distance apart as shown below. Placing the model so that it overlaps each sawhorse the same amount simulates the environment of the analysis.

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.

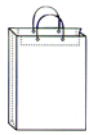
Using Common Objects with Known Weights

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 X 0.0245 = 1.225	1	$0.50
100	___X 0.0245 = _____	____	____
500	___X 0.0245 = _____	____	____
1000	___X 0.0245 = _____	____	____
5000	___X 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.

Glossary

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 it’s 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.

Term	Definition
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.
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 calculated in the 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.

Term	Definition
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.
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 newton 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.
restraint	The restraint describes that part of the model that cannot move in the analysis.

Term	Definition
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.
tension and compression	Internal forces in a beam that are caused by bending.
truss	A simple bridge structure usually used by railroads.
yield strength	The limit of a beam’s strength based on the stresses in the beam.