

ANSYS WB - Bike Crank - All Pages

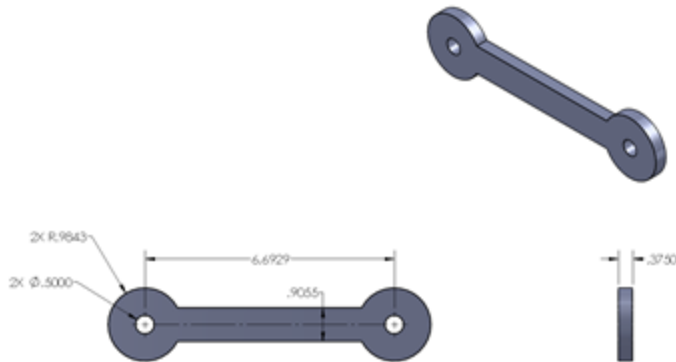
- [Problem Specification](#)
- [Pre-Analysis & Start-up](#)
 - [Hand Calculations](#)
 - [Open ANSYS Workbench](#)
 - [Material](#)
- [Geometry](#)
- [Mesh](#)
 - [Specify Material](#)
 - [Body Sizing](#)
- [Setup \(Physics\)](#)
 - [Force Loading](#)
 - [Fixed Support](#)
- [Solution](#)
 - [Deformation](#)
 - [Stress](#)
 - [Construction Geometry](#)
 - [Strain](#)
- [Results](#)
 - [Deformation](#)
 - [Strain](#)
 - [Stress](#)
- [Verification and Validation](#)

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)
1. [Pre-Analysis & Start-Up](#)
2. [Geometry](#)
3. [Mesh](#)
4. [Setup \(Physics\)](#)
5. [Solution](#)
6. [Results](#)
7. [Verification & Validation](#)

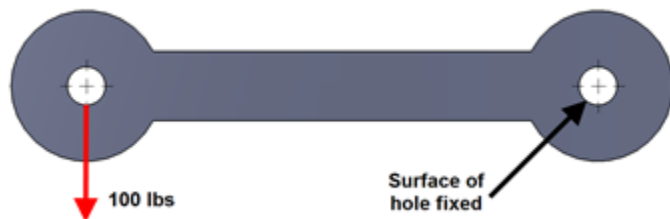
Problem Specification

A bicycle crank is fitted with a strain gauge and a 100 lb force is applied. The bicycle crank's material is aluminum 6061-t6, the Young's modulus is 10,000 ksi, and the Poisson's Ratio is .33. The dimensions (in inches) of the bike crank are shown below.



[Click here to enlarge the image](#)

The bicycle crank is loaded such that the fitting on the right is fixed while the fitting on the left is loaded with a 100 lb force.



Using hand calculations and ANSYS Workbench, determine the strain measured by a 5.64 mm strain gauge centered at (3.3748, -0.09409, .375) away from the force and aligned in the y-direction.

[Go to Step 1: Pre-Analysis & Start-Up](#)

[Go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Setup \(Physics\)](#)

[5. Solution](#)

[6. Results](#)

[7. Verification & Validation](#)

Pre-Analysis & Start-up

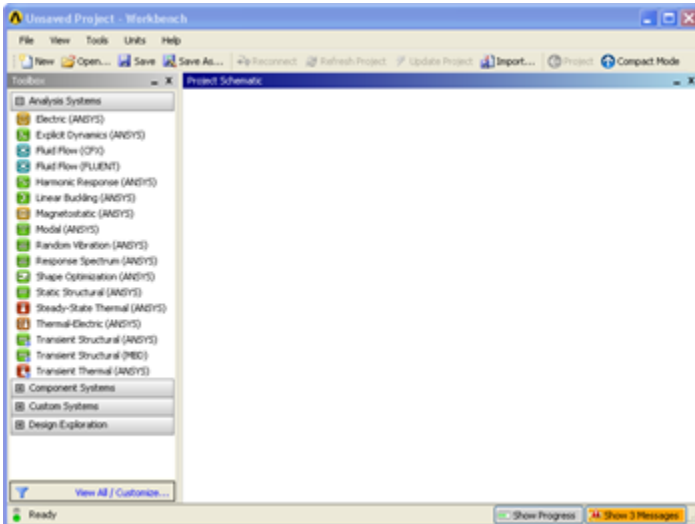
Hand Calculations

It is highly recommended that one perform some sort of hand calculation to predict expected results before attempting a simulation. We call this "Pre-Analysis". The hand calculation can later be used to check the results of the simulation.

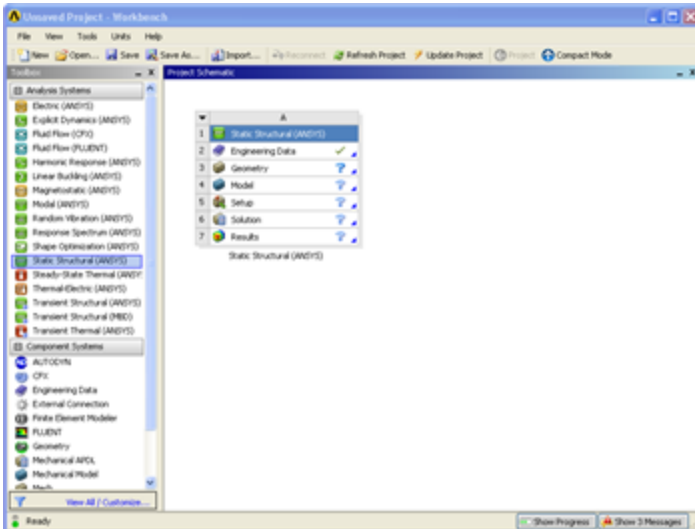
We use Euler-Bernoulli beam theory to calculate the expected strain values at the gage center. Click [here](#) to see the "Pre-Analysis" for the bike crank. Note that the hand calculation uses *tensor* shear strain while ANSYS reports *engineering* shear strain (which is two times the tensor shear strain).

Open ANSYS Workbench

Now that we have completed the pre-calculations, we are ready to do a simulation in ANSYS Workbench! Open ANSYS Workbench by going to **Start > ANSYS > Workbench**. This will open the start up screen seen as seen below



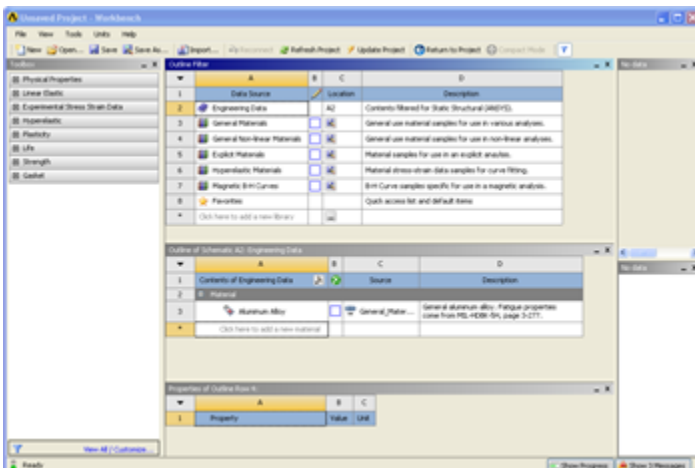
To begin, we need to tell ANSYS what kind of simulation we are doing. If you look to the left of the start up window, you will see the Toolbox Window. Take a look through the different selections. The bicycle crank is a static structural simulation. Load the **Static Structural** tool box by dragging and dropping it into the Project Schematic.



Name the Project Bike Crank .

Material

Now we need to specify what type of material we are working with. Double click **Engineering Data** and it will take you to the Engineering Data Menus.



If you look under the *Outline of Schematic A2: Engineering Data* Window, you will see that the default material is Structural Steel. The Problem Specification specifies the material's Modulus of Elasticity and Poisson ratio. To add a new material, click in an empty box labeled **Click here to add a new material** and give it a name. Our Material is A1 6061-T6. On the left hand side of the screen in the *Toolbox* window, expand **Linear Elastic** and double click **Isotropic Elasticity** to specify E and Unable to find DVI conversion log file.. In the *Properties of Outline Row 4: A1 6061-T6* window, Set the Elastic Modulus units to **psi**, set the magnitude as 1E7, and set the Poisson Ratio to 0.33.

Now that the Material has been specified, we are ready to import the geometry into ANSYS.

Continue to [Step 2: Geometry](#)

[Go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Setup \(Physics\)](#)

[5. Solution](#)

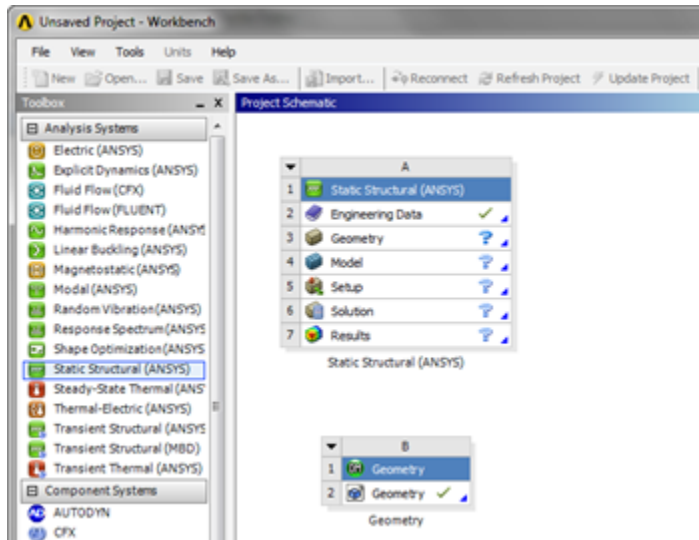
[6. Results](#)

[7. Verification & Validation](#)

Geometry

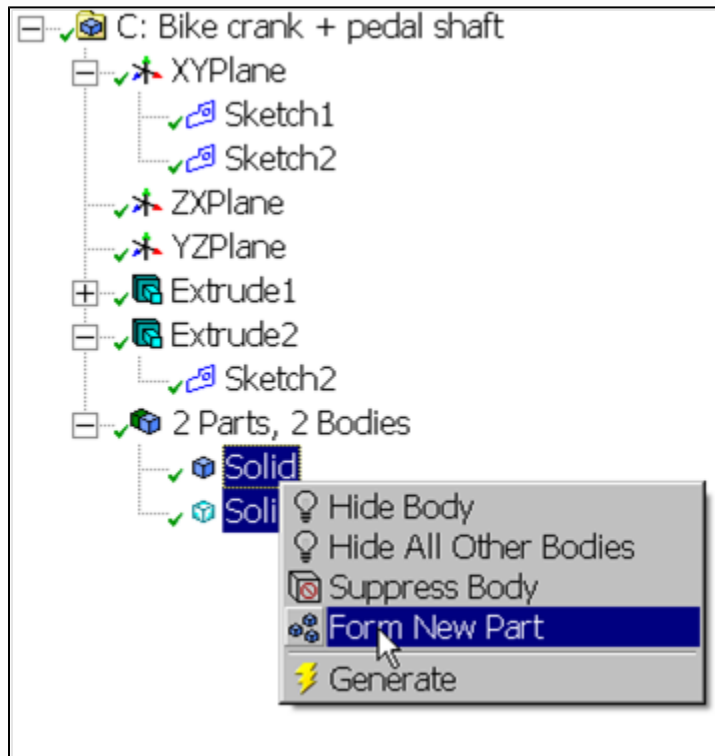
For this problem, we are going to import the geometry into ANSYS from a CAD package. Download the part file [here](#). Save the file somewhere convenient, like your desktop or your working directory.

To open the file in ANSYS, go to **File > Import**. Browse to the geometry location on your computer. If you do not see the file, make sure you are browsing for **geometry files** (the pull down menu at the bottom right of the browsing window for computers running Windows 7). Select the Geometry and click **Open**. This will import your geometry into ANSYS. Your project window should now include the main project, and the newly imported geometry (see below).



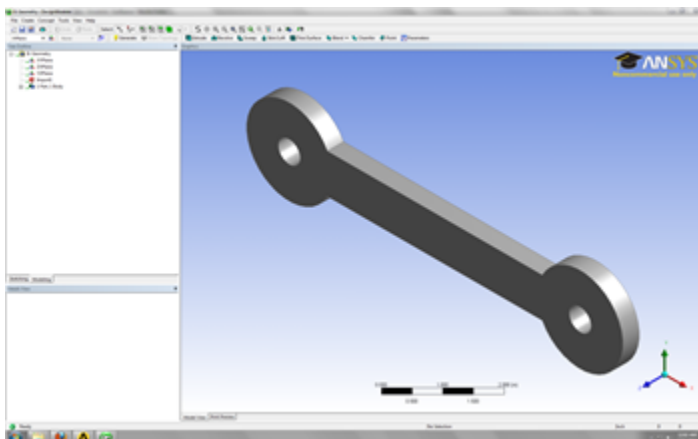
i Are you importing a pedal shaft with the crank arm?

If you are importing 2 parts at once, i.e. a crank and a pedal shaft, you need to connect the two geometries in ANSYS. To accomplish this, look to the *Outline* window. There should be a box labeled **2 Parts, 2 Bodies**. Expand the section, and there should be 2 solid parts underneath. To connect them, highlight both selections by holding down Ctrl and left clicking them. Right click, and select **Form New Part**. This will connect the two geometries and will allow you to carry on the simulation.

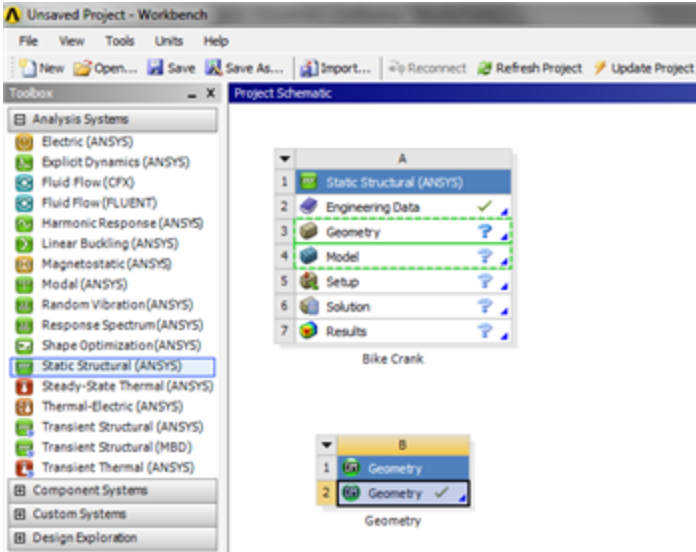



Now that the geometry has been imported, let's open the file and make sure everything is in order! Double click **2 Geometry**. This will open the design modeler. When you are prompted, select **Inch** as your standard unit of measurement. The first thing you should notice is that the geometry is not

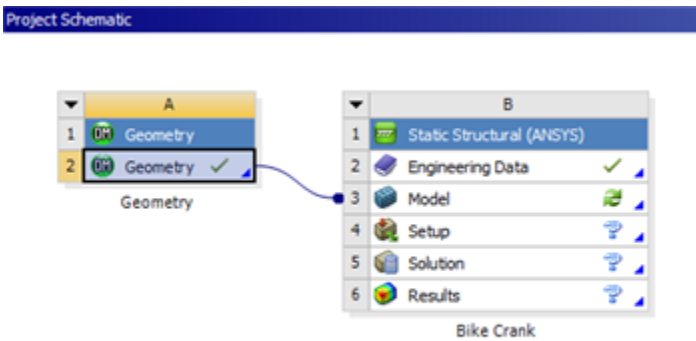
there, so click **Generate** to generate the geometry (be patient - it may take a couple of minutes). When the geometry finally generates, you should see the screen below.



Now that we have verified we are using the geometry we want, we need to tell ANSYS that this is the geometry we want to use. Close the Design Modeler and return to the project window. To connect the geometry to the project, click and drag the **2 Geometry**. As soon as you drag the box, ANSYS will highlight the geometry and model boxes in the main project.



Drag and drop the geometry box onto  Model



The geometry has been connected the project and we are ready for the next step.

Continue to [Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Setup \(Physics\)](#)

[5. Solution](#)

[6. Results](#)


[7. Verification & Validation](#)


Mesh

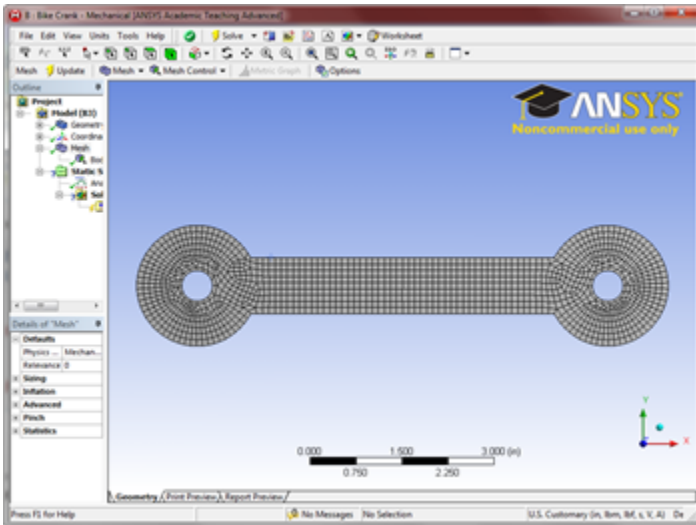
Specify Material

Open ANSYS Mechanical by double clicking **Model**. First, we will tell ANSYS which material we are using for the simulation. Expand **Geometry**, and click **Surface Body** in the *Outline* window. In the *Details* window, select **Material > Assignment > Al 6061-T6**. The material has now been specified.

Body Sizing

In the outline window, click on  **Mesh**. The only thing we will control for this geometry's mesh is the size of the elements. Go to **Mesh Control >**

Sizing to open up the Sizing menu. Select the entire geometry by clicking Body Selection Filter  then clicking on the model. Now click **Geometry > Apply**. Also, change the size of the mesh from default to **0.1 in**. (If the units are not in inches, change them by going to **Units > U.S. Customary (in, lbf, F, S, V, A)**). Now, generate the mesh by clicking **Mesh > Generate Mesh**. The final result should look something like this:



Now that the mesh has been created we can begin specifying the boundary conditions for the simulation.

Continue to [Step 4: Setup \(Physics\)](#)

[Go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Setup \(Physics\)](#)

[5. Solution](#)



[6. Results](#)


[7. Verification & Validation](#)

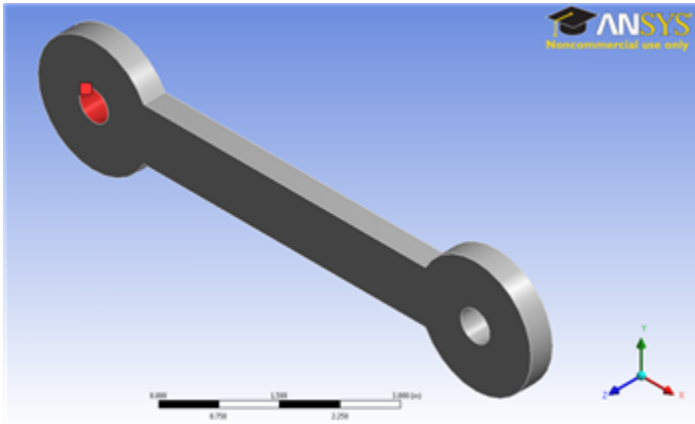
Setup (Physics)

Force Loading

Ok, now we can specify the loading and boundary conditions for the simulation. First, we will specify the 100 lb force on the bike crank at the origin (if you

don't know where the origin is, click on  **Coordinate Systems** to see it). Click on  **Static Structural (B4)** to open the loading tool bar, then select **Loads > Force**. We want to apply the force to the surface on the inside of the hole at the origin. So make sure you are selecting a surface by

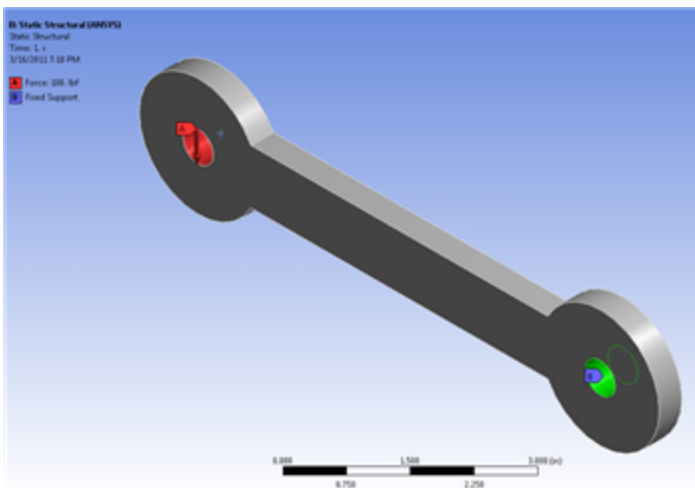
selecting . Next, select the inside of the hole at the origin, and select **Geometry > Apply**



Next, we need to specify the magnitude of the force. We want the 100 lb force to point in the negative y-direction. To do this, we need to select **Define By > Components**, then specify the **Y Component** of the force to be **-100 lbs**.

Fixed Support

Now that the force has been applied to the bike crank, we need to fix a part of the bar. To accomplish this task, select **Static Structural > Supports > Fixed Support**. Select the inside face of the other hole just like we selected the geometry for the force and click **Geometry > Apply**.



The setup is now complete; next, we will choose what we want to solve for and solve the simulation.

Continue to [Step 5: Solution](#)

[Go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Setup \(Physics\)](#)

[5. Solution](#)

[6. Results](#)

[7. Verification & Validation](#)

Solution

Deformation

Although we are only looking for the strain at the strain gauge location, we should also check the deformation of the crank to see if the solution makes intuitive sense. This will allow us to catch any obvious errors we have made setting up the simulation. To add deformation to the solution, first click




Solution (B5)

to add the solution sub menu to menu bar. Now in the solution sub menu click **Deformation > Total** to add the total deformation to the solution. It should appear in the outline tree.

Stress

We will also look at the stress of the bar to verify the simulation against the theory. In the solution sub menu, select **Stress > Normal**. In the details pane, ensure **Orientation** is set to **X Axis**, and **Geometry** is set to **All Bodies**. Rename the Stress to **StressXX**. Now, create another stress the same way, but change **Orientation** to **Y Axis** and rename it **StressYY**.

Construction Geometry

The next thing we are going to do is to set up a path where the strain gauge lies on the bike crank. In the outline window, click  **Model**. Next, go to **Construction Geometry > Path**. This will open up the path toolbox in the details window. Creating a path in Workbench is very easy: you just have to specify the starting and ending points of your path and workbench will create it. In the details window, enter **(3.3748, -.20512, 0.375)** as the starting point and **(3.3748, 0.016929, 0.375)** as the end point. Change the number of sampling points to **50**.

Details of "Path"	
[-] Definition	
Path Type	Two Points
Path Coordinate System	Global Coordinate System
Number of Sampling Points	50.
Suppressed	No
Show Mesh	No
[-] Start	
Coordinate System	Global Coordinate System
Start X Coordinate	3.3748 in
Start Y Coordinate	-0.2051 in
Start Z Coordinate	0.375 in
Location	Click to Change
[-] End	
Coordinate System	Global Coordinate System
End X Coordinate	3.3748 in
End Y Coordinate	1.69e-002 in
End Z Coordinate	0.375 in
Location	Click to Change

When the parameters of the path have been set, right click on the path you just created in the project tree in the outline window and select **Rename**. Rename the path, **Strain Gauge**.

If your default coordinate system is not centered on one of the holes in the crank, or you wish to learn how to define new coordinate systems inside the mechanical window, please refer to the following video:



Why Are We Creating a Path to Represent the Strain Gauge?

We are trying to find the average of the strain over the entire area of the strain gauge. However, lucky for us, the strain does not vary much in the x-direction of the bike crank. Therefore, we can neglect the strain changes in the x-direction, and assume strain only varies in the y-direction. This allows us to use a line as a representation of the strain gauge, making our analysis simpler and not harming our results.

An alternate, more difficult, and more accurate method than using paths is to use surface bodies to average the strain over the entire area covered by the strain gauge. The video below walks you through this process.

Strain

In the Solution sub menu select **Strain > Normal**. Now, we need to specify that we want the strain to be measured at the strain gauge in the y direction. To accomplish this, go to **Scoping Method > Path**, then select **Path > Strain Gauge** in the details window. To change the direction of the strain, select **Orientation > Y Axis**. The rest of the default options should suffice for our simulation.

Details of "Normal Elastic Strain"	
[-] Scope	
Scoping Method	Path
Path	Strain Gauge
Geometry	All Bodies
[-] Definition	
Type	Normal Elastic Strain
Orientation	Y Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
[-] Integration Point Results	
Display Option	Averaged
[-] Results	
<input type="checkbox"/> Minimum	-4.9324e-005 in/in
<input type="checkbox"/> Maximum	1.3965e-004 in/in
[-] Graph Controls	
X-Axis	S
[+] Information	

We are ready to solve the simulation. Press



Continue to [Step 6: Results](#)

[Go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

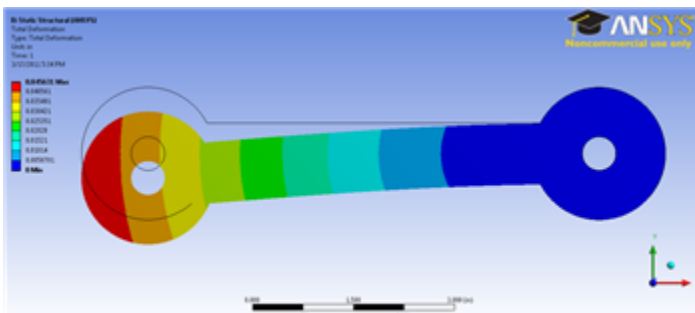
[Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Setup \(Physics\)](#)
- [5. Solution](#)
- [6. Results](#)
- [7. Verification & Validation](#)

Results

Deformation

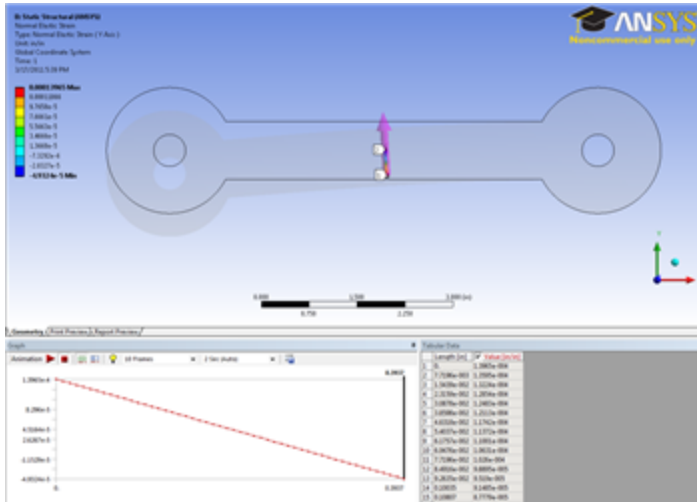
First, we will look at the deformation to see if the simulation makes intuitive sense. In the project outline window, select **Solution > Total Deformation**. Also, in the Result tool bar, select **Edges > Show Undeformed WireFrame**.



From the deformed view, we can see that the crank deformed basically as we would expect from this loading.

Strain

Now, let's examine the strain at the strain gauge. If we select **Solution > Normal Elastic Strain** we can see the strain distribution along the path we created. In the Tabular Data window, we can see the numerical values of the strain along the path.



While the strain along the entire strain gauge is useful, what we really want is the average along the strain gauge. Right click anywhere in tabular data window, and go to **Export**. Save the file as an .xls file and name it **Strain data**. Open the file, and average the data by selecting all of the strain data, then going to **Formulas > AutoSum > Average**. At the bottom of the data, we will see the calculated average of the strain gauge: 4.5167E-5 in/in.

Strain Gauge Technique Comparison

There are multiple ways to model a strain gauge in ANSYS

1. Creating a small area gauge of zero thickness and zero stiffness
2. Creating a path and averaging the strain values across it
3. Creating a path and calculating strain from the change in length
4. Finding the displacements of two nodes and using this to calculate strain

All four methods were used to measure strain in both the X and Y directions at the same position on the bicycle crank. Results are as follows:

Strain Values:

	Area Gauge	Line Strain	Strain from Line Displacement	Strain from Nodal Displacement
X-Strain (Micro-Strain)	787.17	758.93	785.71	770
Y-Strain (Micro-Strain)	-251.99	-251.65	-250.00	-260

Percent Differences using Area Gauge as the base-line:

	Line Strain vs. Area Gauge	Line Displacement vs. Area Gauge	Nodal Displacement vs. Area Gauge
X-Strain (Micro-Strain)	-3.59%	.185%	2.18%
Y-Strain (Micro-Strain)	.135%	.790%	3.18%

Theoretical and Experimental* Values at the Same Position:

	Experimental (ARAMIS)	Theory
--	-----------------------	--------

X-Strain (Micro-Strain)	932	789.13
Y-Strain (Micro-Strain)	-153	-260.41

*Experimental data was obtained from using digital image correlation using ARAMIS software. See Verification and Validation section for details.

There is no significant difference between the results of the different methods. All four techniques output acceptable results in both the X and Y directions with only a maximum of a three percent difference between using any of the methods. The Y direction has a slight discrepancy from the experimental values, but this can be explained from an unexpected difference in material properties (see Verification and Validation section).

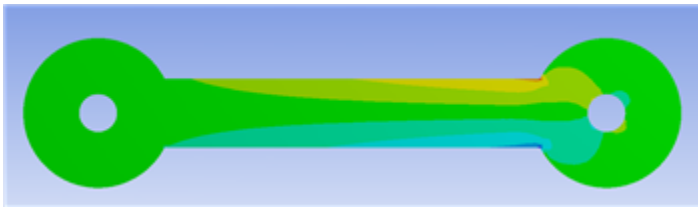
The procedure for using the area gauge is much more lengthy and complicated than the other three; however, the area gauge allows for strain measurements in cross-wise directions, not along the X, Y, or Z axes, and no further calculations are needed after ANSYS solves the model. The two methods which use paths are both fast and easy to set up, and the gauge can be modified while in the Mechanical workspace. The two path methods both involve calculations after ANSYS completes the solution, though, and the paths will only give solutions in the X, Y, or Z directions. The nodal displacement technique is the most unwieldy, and heavily depends on the mesh of the part. There need to be two nodes along a horizontal or vertical line which are in the region where strain is to be desired. The nodal displacement also requires calculations.


The area strain gauging technique allows for the greatest flexibility in the measurement of the strains. Any direction can be calculated. The line strain gauges are much easier to set up, though, and so if strain is only needed in one of the axis directions, as in this tutorial, these are a good substitute to the area gauge.

For a more details on the analysis of strain gauge techniques see [ANSYS Strain Gauge Techniques Report.doc](#)

Stress

Now, let's check the stress at the strain gauge. Click on **Solution > StressXX** in the Project Outline window. This will bring up the stress distribution over the entire crank.



Now, click on the probe tool  **Probe**. Hover the probe tool around where the strain gauge is located. We just want to compare the stresses we see to the calculated value of -1368.65 psi. If you hover the probe around that area, you should find that the stresses are about what we expect them to be based on the calculation. Repeat this step for **Solution > StressYY**



Here, we assumed that the stress was zero. If we use the probe tool to examine the middle section of the bar, we find that most of the stresses are on the order of 1×10^{-3} , meaning once again the simulation agrees with our calculations.

Continue to [Step 7: Verification & Validation](#)
[Go to all ANSYS Learning Modules](#)
[See and rate the complete Learning Module](#)

Author: Benjamin Mullen, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

Verification and Validation

Now that we have our results, it is important that we check to see that our computational simulation is accurate. One possible way of accomplishing this task is comparing to the pre-calculations, as we did in the results section. Another way to check our results is by refining the mesh further. The smaller the elements in the mesh, the more accurate our simulation will be, but the simulation will take longer. To refine the mesh, look to the outline tree and click **Mesh > Body Sizing** Change the element sizing to .05 (half the size of the mesh we originally tried). The new mesh looks like [this](#) . It has twice as many elements as the original.

Now hit solve. Compare the values for your stresses with those we found for the original mesh. Are the very different? Or do they seem to approach a limit? If the latter, the mesh is refined enough and if you modeled the problem correctly, you are done! Below are the values from our original mesh, followed by the values for our refined mesh.

	Strain
Theory Value	4.5165 x 10 ⁻⁵
Measured Value	~ 4.2 x 10 ⁻⁵
Original Mesh	4.5167 x 10 ⁻⁵
Refined Mesh	4.5165 x 10 ⁻⁵

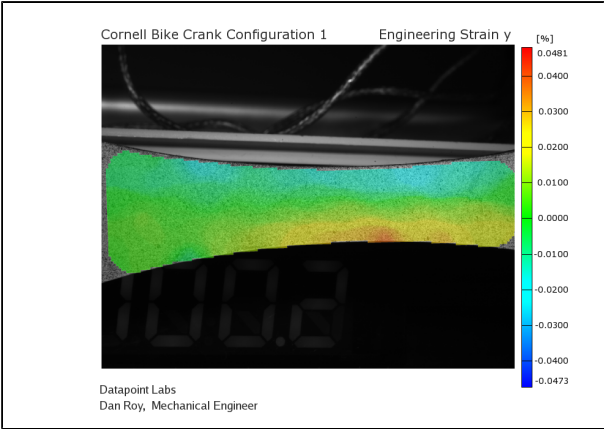
As one can see from the table above the results do not change greatly as the mesh is refined. This means we don't need to refine the mesh further.

Verification using Digital Image Correlation (DIC)

A more thorough and professional verification of the ANSYS bicycle crank simulation was done by setting up the crank under the same boundary conditions as in the tutorial and using digital image correlation technology with the software ARAMIS to find the strain field along the face of the crank. This was done to show how the verification and validation step can be accomplished in the industry using state of the art technology.

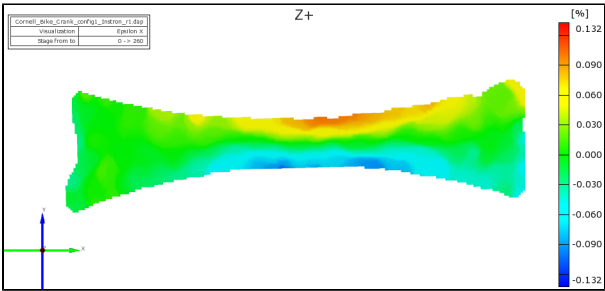
Digital image correlation is a technology that uses cameras to watch the movements of a mechanical part as it is being loaded. From the deformations of the part, ARAMIS is able to calculate a strain field across the entire part. This strain field can than be used to find local strains by modeling strain gauges within the software.

The ARAMIS works in a similar fashion to a finite element analysis. First the part to be loaded is painted with a speckled pattern, and a picture is taken at the zero load state. ARAMIS maps facets, essentially two dimensional elements, to the speckled pattern. Each facet contains a unique pattern of speckles within it, and ARAMIS tracks the movements of each of these patterns while the part is being loaded. The facet size can be adjusted, just as the element size in FEA software can. Multiple pictures are taken as the load is applied to the part, and at each stage the ARAMIS calculates strain fields in the X and Y directions, along with other results such as Von Mises strain and shear angle. The results are mapped to the face of the part as contours.

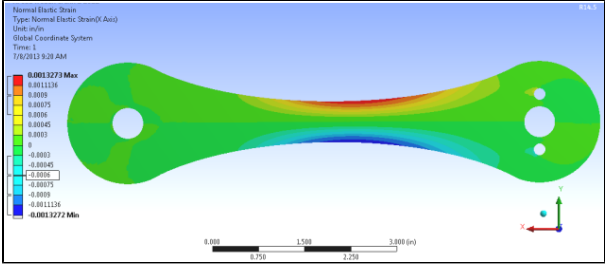


ARAMIS was used to obtain the X and Y strain fields for a bike crank made from Al 6061 T6, and an ANSYS model of the same geometry and loading conditions was created. The results of the strain fields output are shown below:

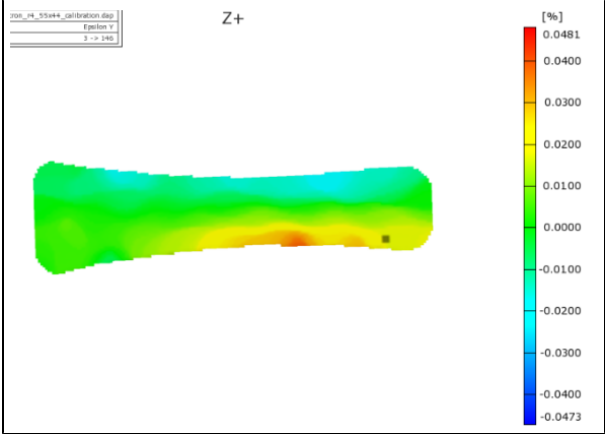
ARAMIS X-Strain Field:



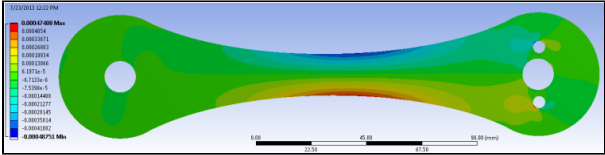
ANSYS X-Strain Field:



ARAMIS Y-Strain Field:



ANSYS Y-Strain Field:



The contours given by ANSYS match the ARAMIS contours of the strain fields very well. We can even see that ANSYS correctly matches the regions where the strain is most concentrated, which is not in the middle of the crank as one might first expect. ARAMIS shows that ANSYS is correct in having the strain concentrations slightly to the right of the center. The ARAMIS X-strain field does show a more spread out area of high strain magnitude on the bottom edge of the crank, but this discrepancy is most likely due to the fact that the experiment is not run under ideal conditions as the ANSYS model is.

The magnitudes of the range of the ARAMIS contours were scaled to match those given by ANSYS, so the colors on the ARAMIS strain fields represent the same magnitude of strain as the colors in the ANSYS fields. ANSYS shows slightly higher strain magnitudes along the top and bottom edges of the crank, but besides this discrepancy, the strain field's magnitudes match fairly well.

Strain gauges were also added to both the ARAMIS strain fields and the ANSYS model. One gauge was placed on the top half of the crank, the other on the bottom half. The gauges used were the same dimensions and placed in the same positions in both ARAMIS and ANSYS. X and Y strain values were obtained from both:

Strain Gauge Values:

Strain Gauge and Direction	ANSYS	ARAMIS	Theory
Top Gauge X	781.64	932	789.13
Top Gauge Y	-250.38	-153	-260.41
Bottom Gauge X	-883.35	-918	-893.42
Bottom Gauge Y	280.38	188	294.83

Percent Differences:

Strain Gauge and Direction	ANSYS vs. Theory	ARAMIS vs. Theory	ANSYS vs. ARAMIS
Top Gauge X	-.949%	18.1%	-16.1%
Top Gauge Y	3.85%	-41.2%	63.6%
Bottom Gauge X	-1.13%	2.75%	-3.77%
Bottom Gauge Y	-4.9%	35.7%	49.1%

The results of the strain gauge comparison easily show that the X-Strain values output by ANSYS are accurate according to both the theoretical values and the experimental values. The experimental ARAMIS values are slightly higher than both the theoretical and ANSYS values, but due to slight differences in geometry between the physical crank and the CAD model, and noise in the experiment, these differences are expected.

The strains in the Y direction are predicted by ANSYS to be higher by about one-hundred micro-strain than the values that ARAMIS reports. This discrepancy was explored in more detail with further gauging experiments. It appears that the Poisson's Ratio of the bike crank used is not the expected .34 but is in fact lower. As the Poisson's Ratio was decreased within ANSYS, the Y-strain gauge values began to match the ARAMIS values.

ANSYS accurately predicts the regions and magnitudes of the strain concentrations in the direction with the greatest strains. It also accurately reports X-strain gauge values. The X-direction is also of more interest because it has higher strains. Even though the Y-direction did not have as close of a correlation, the discrepancy is believed to be due to the material properties of the bicycle crank.

For more details on validation using digital image correlation, see [Validation of ANSYS Workbench with DIC.doc](#).

We're Done!

[Click here to go to all ANSYS Learning Modules](#)

[See and rate the complete Learning Module](#)