Bicycle Frame Design

Course Project

ME 460 001 — Dr. V. Prantil

Paul Gessler
Adam Lange

Mechanical Engineering Department
Milwaukee School of Engineering

February 5, 2012

Introduction

In this finite element analysis course project, a bicycle frame was designed. The dimensions of the frame are shown in Fig. 1. The frame is to be made out of aluminum tube with properties listed in Table 1. The ultimate goal of the project is to minimize the weight of the frame.

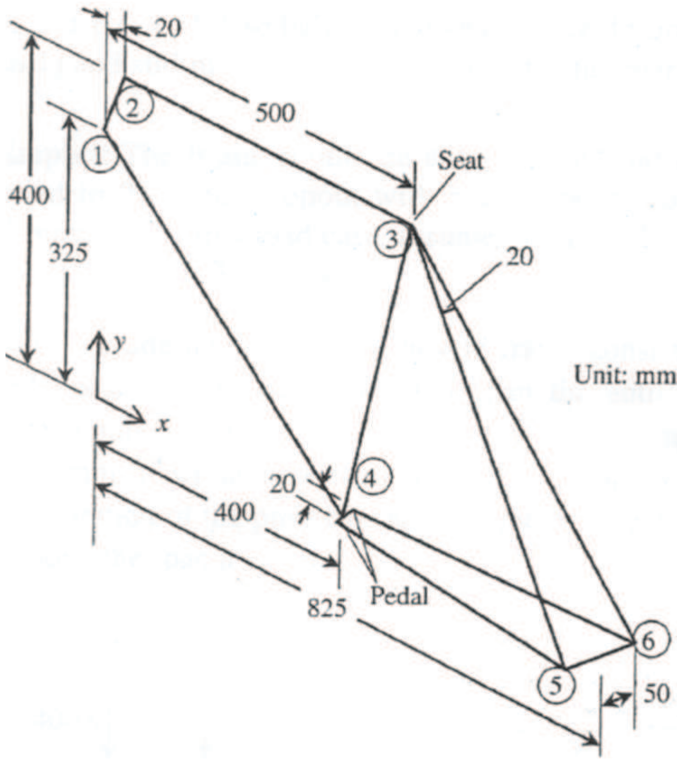

Figure 1: Bicycle Frame Dimensions [1]

Table 1: Aluminum Tubing Properties

Material Property	Symbol	Value	Units
Young's Modulus	E	70	GPa
Poisson's Ratio	ν	0.33	-
Density	ρ	2580	kg/m^3
Yield Strength	σ_y	210	MPa

Two load cases were considered in the design of the frame:

Vertical Load: When an adult rides the bike, the nominal load is estimated as a downward load of 600 N at the seat position and a load of 200 N at the crank location. A dynamic load factor, $G = 2$, was used.

Horizontal Impact Load: The frame was designed to withstand a horizontal impact load of 1,000 N applied to the front dropout with the rear dropouts constrained from any translational motion. A dynamic load factor of $G = 2$ was used for this load case as well.

The Von Mises Effective Stress failure criterion was used to predict factor of safety for the analyses throughout.

(1) Consider a crude model of your bicycle frame consisting of a single beam with two point loads. The crude model will span the entire length of the bicycle, have a tubular cross section with OD of 25 mm, and a wall thickness of 2 mm. If we assume the bike frame is a single straight beam, then the configuration of the problem can be represented in Fig. 2, where the lineal dimensions along the span are in meters.

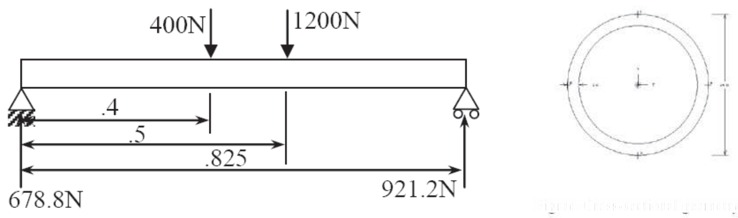

Figure 2: Simplified Model Loading and Cross Section

NOTE: Most of the analysis for Part (1) is taken from earlier coursework [2] with minor modifications.

Step 0: What should the solution look like?

Given the geometry and load case, the solution should exhibit the following characteristics:

- Reactions must sum to 1600 N
- Stresses on bottom should be tensile, top should be compressive
- Bending moment diagram should be piecewise-linear, zero at supports
- Shear force diagram should be piecewise-constant with 3 segments
- Maximum deflection should occur between the two loads
- Maximum bending stress should occur at the point of application of the 1200 N load

Step 1: Element Formulation

The '`BEAM3`' element was selected for the analysis since it has three degrees of freedom (DOF) at each endpoint, behaving like a classic Euler-Bernoulli beam. '`BEAM3`' has two nodes, each of which have the following degrees of freedom: x-displacement, y-displacement, and z-rotation. In this analysis, all elements have the same properties as defined in the tutorial, and the shear deflection constant (SHEARZ) was taken to be 0.

Step 2: Discretization

The choice of the '`BEAM3`' element predicates all loads and boundary conditions applied at element endpoints. Therefore, the coarsest mesh possible is one comprised of 3 elements, presented in Fig. 3. Note that element numbers are circled and node numbers are boxed.

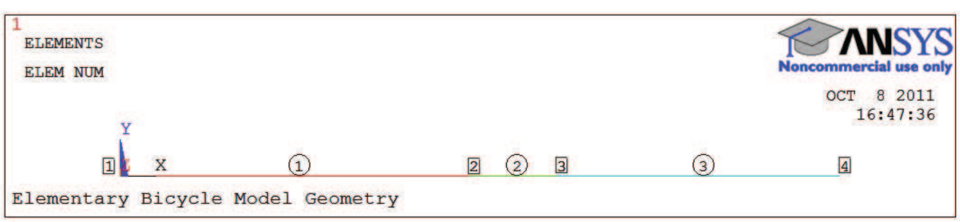

Figure 3: 3-Element Mesh Discretization

A finer mesh was also created with 33 elements. The nodes and elements in this mesh were also numbered in increasing order from the left end to the right end of the beam.

Steps 3-5: Constitutive Relationships and Matrix Assembly

Since ANSYS is a small-deformation code and the stress-strain relation is linear Hooke's law, the element-wise load-displacement relation is as in Eq. 1:

$$\left[K^{(e)}\right]\left\{\delta^{(e)}\right\} = \left\{f^{(e)}\right\} \tag{1}$$

ANSYS also handles assembling the global stiffness matrix $[K]$ from all individual $\left[K^{(e)}\right]$ matrices.

Step 6: Boundary Conditions

The boundary conditions required to produce a solution are listed for both meshes in Table 2. The application of these boundary conditions allows the system of equations to be solved in Step 7 below.

Table 2: Boundary Conditions

Boundary Condition	Value (3-Element Mesh)	Value (33-Element Mesh)
Left End Displacement	$U_{1,x} = U_{1,y} = 0$	$U_{1,x} = U_{1,y} = 0$
Right End Displacement	$U_{4,y} = 0$	$U_{34,y} = 0$
Applied Load 1	$F_{2,y} = 400\,\mathrm{N}$	$F_{17,y} = 400\,\mathrm{N}$
Applied Load 2	$F_{3,y} = 1200\,\mathrm{N}$	$F_{21,y} = 1200\,\mathrm{N}$

Steps 7-8: Solution and Results Calculation

This step was performed by the ANSYS APDL Software for both meshes. Both meshes had reactions of 678.79 N and 921.21 N at the left and right support, respectively. The deformed meshes for the coarse and fine mesh calculations are presented in Figs. 4 and 5, respectively.

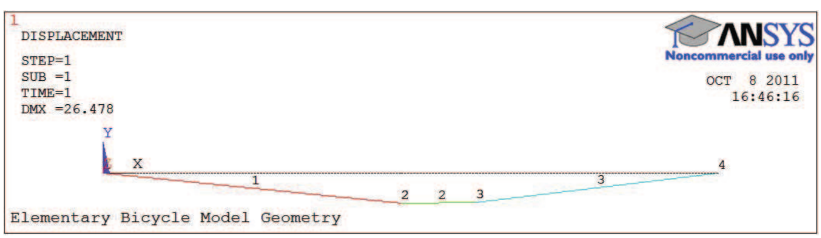

Figure 4: Coarse Deformed Mesh

Note that the deformed shapes look significantly different from each other, despite each analysis having the same loading and boundary conditions. This discrepancy results from the ANSYS plotting methodology, which does not use the shape functions to plot the shape of each element. Rather, it simply plots the endpoint displacements of each element and connects them with straight lines. Therefore, the fine mesh (Fig. 5) shows a much better approximation of the beam shape.

The results for the coarse and fine meshes are presented in Tables 3 and 4, respectively.

Bending moment diagrams for the coarse and fine meshes are presented in Figs. 6 and 7, respectively.

The ANSYS-reported shear force diagrams for each mesh are presented in Figs. 8 and 9. Note that the shear diagrams are inverted because of ANSYS's choice of convention.

Table 3: Coarse Mesh Results

Quantity	Node	Loc'n	Value	Units	Quantity	El.	Loc'n (mm)	Value	Units
Displacement	1	0	0.0000	mm	Maximum Bending Stress	1	200.0	352.50	N/mm^2
	2	400	$\boxed{-26.478}$	mm		2	450.0	$\boxed{388.69}$	N/mm^2
	3	500	-25.622	mm		3	662.5	$\boxed{388.69}$	N/mm^2
	4	825	0.0000	mm					

Table 4: Fine Mesh Results

Quantity	Node	Loc'n	Value	Units	Quantity	El.	Loc'n (mm)	Value	Units
Displacement	1	0	0	mm	Maximum Bending Stress	1	12.5	22.031	N/mm^2
	2	25	-1.0565	mm		2	37.5	44.062	N/mm^2
	3	50	-2.3237	mm		3	62.5	66.094	N/mm^2
	4	75	-4.6317	mm		4	87.5	88.125	N/mm^2
	5	100	-6.9082	mm		5	112.5	110.16	N/mm^2
	6	125	-9.1375	mm		6	137.5	132.19	N/mm^2
	7	150	-11.304	mm		7	162.5	154.22	N/mm^2
	8	175	-13.391	mm		8	187.5	176.25	N/mm^2
	9	200	-15.385	mm		9	212.5	198.28	N/mm^2
	10	225	-17.268	mm		10	237.5	220.31	N/mm^2
	11	250	-19.025	mm		11	262.5	242.34	N/mm^2
	12	275	-20.641	mm		12	287.5	264.37	N/mm^2
	13	300	-22.099	mm		13	312.5	286.41	N/mm^2
	14	325	-23.384	mm		14	337.5	308.44	N/mm^2
	15	350	-24.480	mm		15	362.5	330.47	N/mm^2
	16	375	-25.372	mm		16	387.5	352.50	N/mm^2
	17	400	-26.043	mm		17	412.5	361.55	N/mm^2
	18	425	-26.565	mm		18	437.5	370.60	N/mm^2
	19	450	$\boxed{-26.658}$	mm		19	462.5	379.65	N/mm^2
	20	475	-26.580	mm		20	487.5	$\boxed{388.69}$	N/mm^2
	21	500	-26.237	mm		21	512.5	$\boxed{388.69}$	N/mm^2
	22	525	-25.824	mm		22	537.5	358.79	N/mm^2
	23	550	-24.719	mm		23	562.5	328.89	N/mm^2
	24	575	-23.560	mm		24	587.5	299.00	N/mm^2
	25	600	-22.165	mm		25	612.5	269.10	N/mm^2
	26	625	-20.558	mm		26	637.5	239.20	N/mm^2
	27	650	-18.758	mm		27	662.5	209.30	N/mm^2
	28	675	-16.787	mm		28	687.5	179.40	N/mm^2
	29	700	-14.666	mm		29	712.5	149.50	N/mm^2
	30	725	-12.418	mm		30	737.5	119.60	N/mm^2
	31	750	-10.062	mm		31	762.5	89.699	N/mm^2
	32	775	-7.6214	mm		32	787.5	59.799	N/mm^2
	33	800	-5.1165	mm		33	812.5	29.900	N/mm^2
	34	825	0.0000	mm					

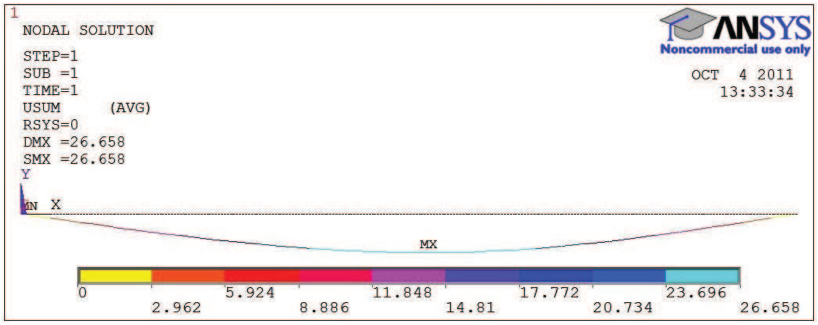

Figure 5: Fine Deformed Mesh

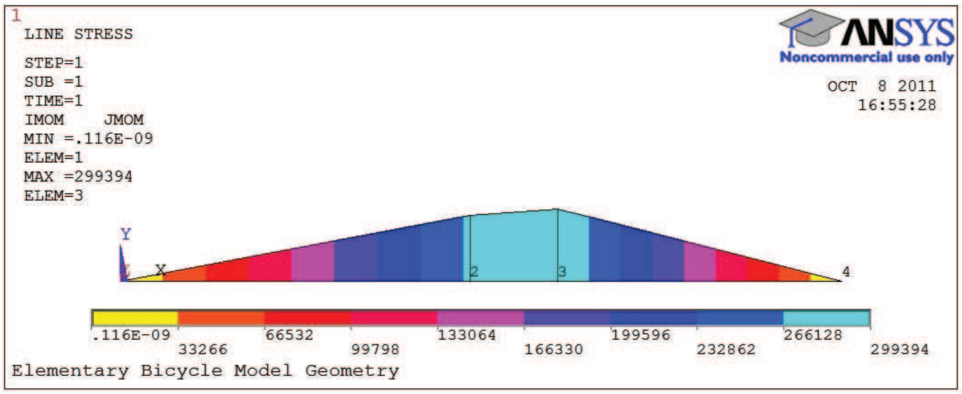

Figure 6: ANSYS Moment Diagram - Coarse Mesh

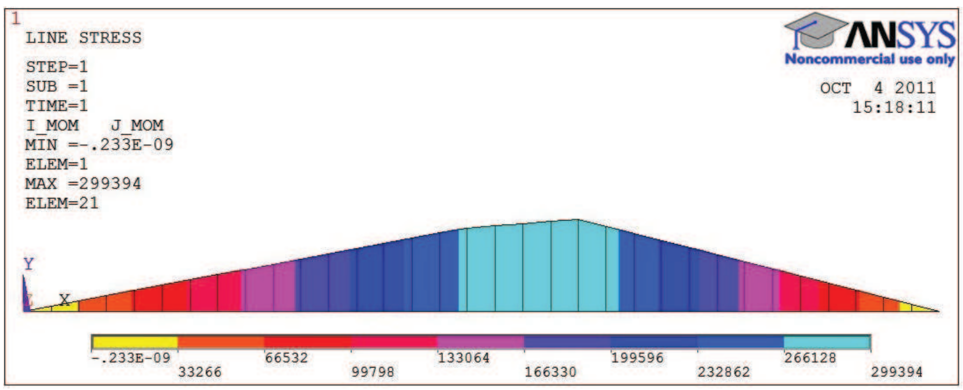

Figure 7: ANSYS Moment Diagram - Fine Mesh

Since the beam has only point loads applied, the moment varies linearly between loads and the shear is constant between loads. So in this case, the ANSYS plots show the correct distribution of moment and shear throughout the beam, unlike the plots of displacement previously discussed.

Step 9: Validation and Interpretation

The analytical solution to the beam loading presented in Fig. 2 was computed using the principle of superposition and beam deflection equations given by Levinson [3]. The analytical solution makes the following assumptions:

- Small deformations
- Linear elastic material

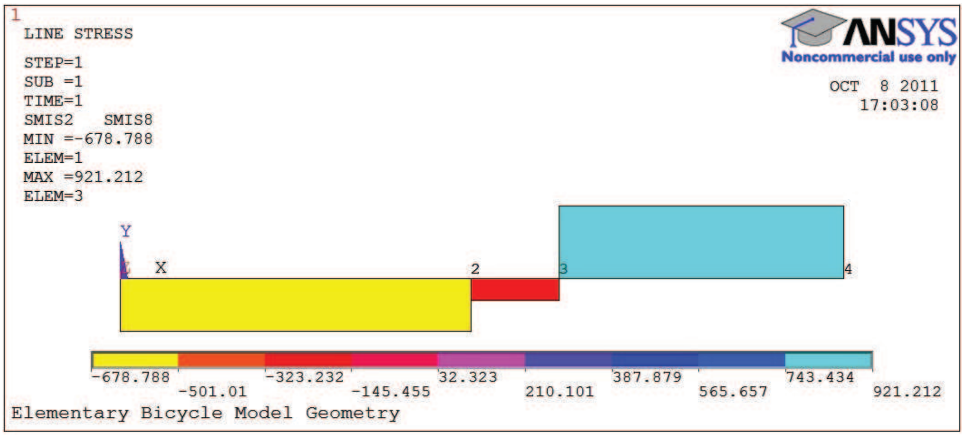

Figure 8: ANSYS Shear Force Diagram - Coarse Mesh

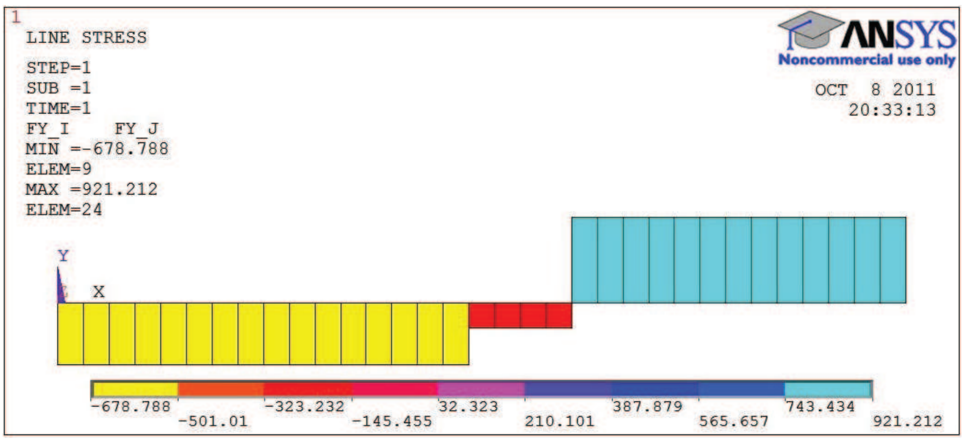

Figure 9: ANSYS Shear Force Diagram - Fine Mesh

- Plane sections remain plane
- Slender beam
  - Shear deflection is not taken into account in this theory, so for the solution to be accurate, the deflection caused by the bending must be dominant.

In addition, the Von Mises effective stress (σ_{eff}) was calculated using Eq. (2), given by Shigley [4].

$$\sigma_{\text{eff}} = \sqrt{{\sigma_{\text{x}}}^2 + 3{\tau_{\text{xy}}}^2}, \tag{2}$$

where σ_{x} is the normal stress due to bending and τ_{xy} is the direct shear stress.

First, the reactions were determined:

$$\sum M_1 = 0 = -(400\,\text{N})(400\,\text{mm}) - (1200\,\text{N})(500\,\text{mm}) + (R_2)(825\,\text{mm}) \tag{3}$$
$$R_2 = 921.212\,\text{N} \tag{4}$$

$$\sum F_y = 0 = R_1 - 400\,\text{N} - 1200\,\text{N} + 921.212\,\text{N} \tag{5}$$
$$R_1 = 678.788\,\text{N} \tag{6}$$

Then, to determine the deflections, each load is considered separately and must follow the form in Eq. 7:

$$\delta(x) = \begin{cases} \frac{Pbx}{6lEI}\left(l^2 - x^2 - b^2\right) & \text{for } x \le a, \\ \frac{Pb}{6lEI}\left[\frac{l}{b}\left(x-a\right)^3 + \left(l^2 - b^2\right)x - x^3\right] & \text{for } x > a, \end{cases} \tag{7}$$

where a is the point of load application, measured from the left end of the beam and b is the remaining length along the beam (to the right of the load).

For the 1200 N load, $a = 500\,\mathrm{mm}$ and $b = 325\,\mathrm{mm}$:

$$\delta_1(x) = \begin{cases} \frac{(1200\,\mathrm{N})(325\,\mathrm{mm})x}{6(825\,\mathrm{mm})(70000\,\mathrm{N/mm^2})(9628.2\,\mathrm{mm^4})} \\ \qquad \left((825\,\mathrm{mm})^2 - x^2 - (325\,\mathrm{mm})^2\right) & \text{for } x \le 500\,\mathrm{mm}, \\ \frac{(1200\,\mathrm{N})(325\,\mathrm{mm})}{6(825\,\mathrm{mm})(70000\,\mathrm{N/mm^2})(9628.2\,\mathrm{mm^4})} \\ \qquad \left[\frac{825\,\mathrm{mm}}{325\,\mathrm{mm}}\left(x - 500\,\mathrm{mm}\right)^3 + \left((825\,\mathrm{mm})^2 - (325\,\mathrm{mm})^2\right)x - x^3\right] & \text{for } x > 500\,\mathrm{mm}. \end{cases} \tag{8}$$

Similarly, for the 400 N load, $a = 400\,\mathrm{mm}$ and $b = 425\,\mathrm{mm}$:

$$\delta_2(x) = \begin{cases} \frac{(400\,\mathrm{N})(425\,\mathrm{mm})x}{6(825\,\mathrm{mm})(70000\,\mathrm{N/mm^2})(9628.2\,\mathrm{mm^4})} \\ \qquad \left((825\,\mathrm{mm})^2 - x^2 - (425\,\mathrm{mm})^2\right) & \text{for } x \le 400\,\mathrm{mm}, \\ \frac{(400\,\mathrm{N})(425\,\mathrm{mm})}{6(825\,\mathrm{mm})(70000\,\mathrm{N/mm^2})(9628.2\,\mathrm{mm^4})} \\ \qquad \left[\frac{825\,\mathrm{mm}}{425\,\mathrm{mm}}\left(x - 500\,\mathrm{mm}\right)^3 + \left((825\,\mathrm{mm})^2 - (425\,\mathrm{mm})^2\right)x - x^3\right] & \text{for } x > 400\,\mathrm{mm}. \end{cases} \tag{9}$$

Then, to obtain the complete deflection solution, sum and simplify the two equations, taking note the piecewise definitions of each segment:

$$\delta(x) = \begin{cases} \left(1.169 \times 10^{-7}\,\mathrm{mm^{-2}}\right)\left(575000\,\mathrm{mm^2} - x^2\right)x \\ \quad + \ \left(5.096 \times 10^{-8}\,\mathrm{mm^{-2}}\right)\left(500000\,\mathrm{mm^2} - x^2\right)x & \text{for } x \le 400\,\mathrm{mm}, \\ \left(9.892 \times 10^{-8}\,\mathrm{mm^{-2}}\right)\left((x - 500\,\mathrm{mm})^3 + \left(500000\,\mathrm{mm^2}\right)x - x^3\right) \\ \quad + \ \left(1.12553 \times 10^{-7}\,\mathrm{mm^{-2}}\right)\left(575000\,\mathrm{mm^2} - x^2\right)x & \text{for } 400\,\mathrm{mm} < x < 500\,\mathrm{mm}, \\ \left(9.892 \times 10^{-8}\,\mathrm{mm^{-2}}\right)\left((x - 500\,\mathrm{mm})^3 + \left(500000\,\mathrm{mm^2}\right)x - x^3\right) \\ \quad + \ \left(2.967 \times 10^{-7}\,\mathrm{mm^{-2}}\right)\left((x - 500\,\mathrm{mm})^3 + \left(575000\,\mathrm{mm^2}\right)x - x^3\right) & \text{for } x > 500\,\mathrm{mm}. \end{cases} \tag{10}$$

This solution was implemented in MATLAB to generate the displacement, maximum bending stress, and shear force at any point on the beam. The validation results are presented in Table 5. A plot comparing the displacements given by the two meshes to the theoretical deflection curve is displayed in Fig. 10. Plots comparing the bending stress and shear force are presented in Figs. 11 and 12, respectively.

Table 5: Validation Results

Result	v_{max} (mm)	Location of v_{max} (mm)	σ_{max} (N/mm^2)	Location of σ_{max} (mm)
Coarse Mesh	-26.478	400.00	388.69	500
Fine Mesh	-26.658	425.00	388.69	500
Theory	-26.662	427.43	388.69	500
Hand Calculation	-26.478	400.00	388.69	500

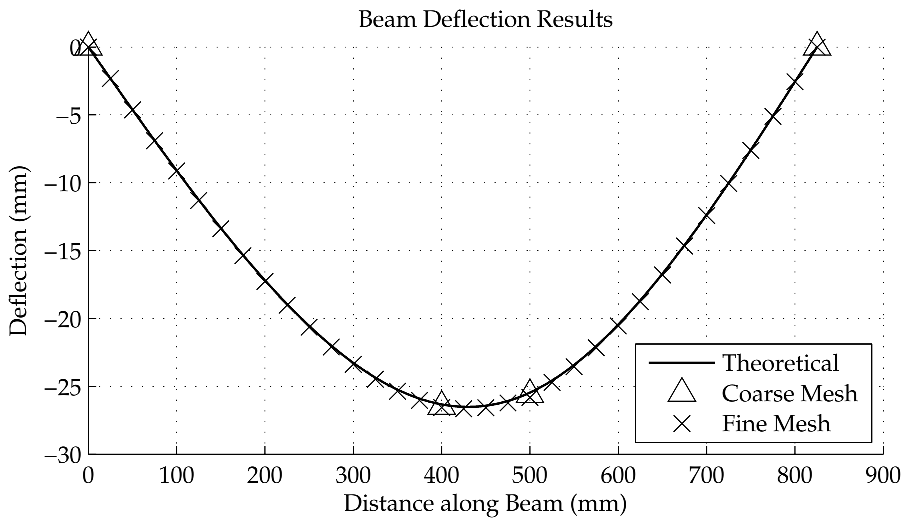

Figure 10: Beam Deflection Results

The small error in the maximum deflection (and its location) is a result of ANSYS's incorrect post-processing methods. This error becomes 20 times smaller with a tripling of the number of elements, and would approach zero as more elements are added. This might be interpreted as a need for more elements, however, the true solution is embedded in the shape functions that ANSYS decides to neglect when listing and plotting results!

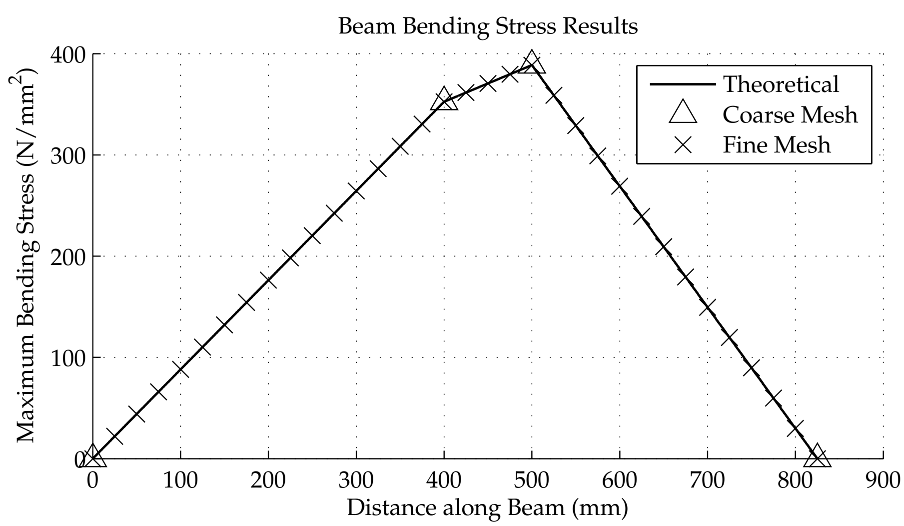

Figure 11: Beam Bending Results

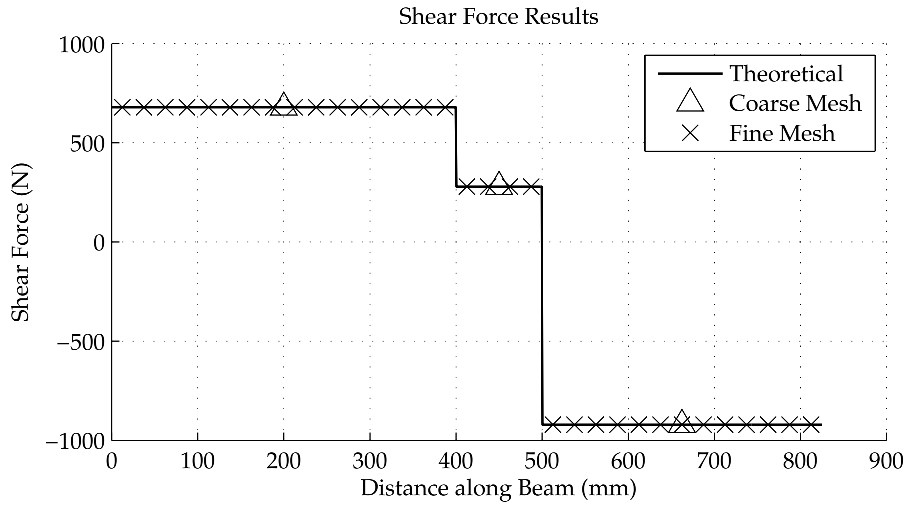

Figure 12: Shear Force Results

As expected, the results for bending stress and shear force are accurate to machine precision. This is because a beam with only point loads will exhibit piecewise linearly-varying moment and piecewise-stepped shear forces. In these cases, the ANSYS linear interpolation is sufficient to obtain the true solution between nodes, unlike with the cubic deflection solution.

Figure 13 shows the Von Mises effective stress calculated using Eq. (2) on the analytical solution results compared to the FEA-reported results accessed through an element table with the fine mesh. The factor of safety for the single beam under the given vertical load case is 0.54.

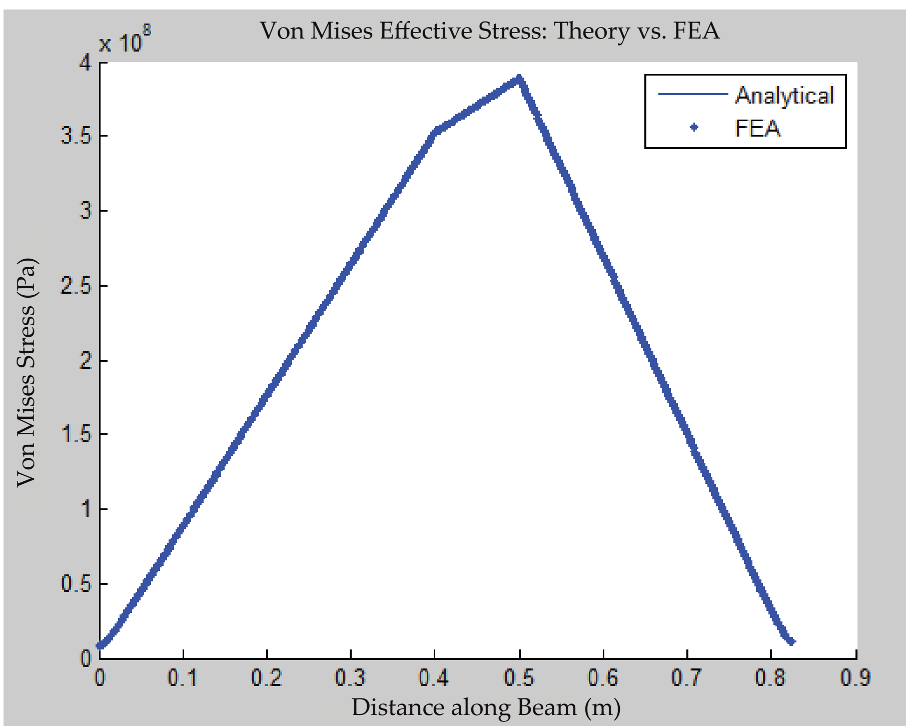

Figure 13: Single Beam Von Mises Stress

(2) Consider a crude model of your bicycle frame consisting of a single point loaded beam having the equivalent stiffness of two of the original tube beams separated vertically by a distance of 400 mm. The crude bike model will, again, span the entire length of the bicycle. If we assume the bike frame is a 'double' straight beam, then the configuration of the problem can be depicted as in Fig. 14.

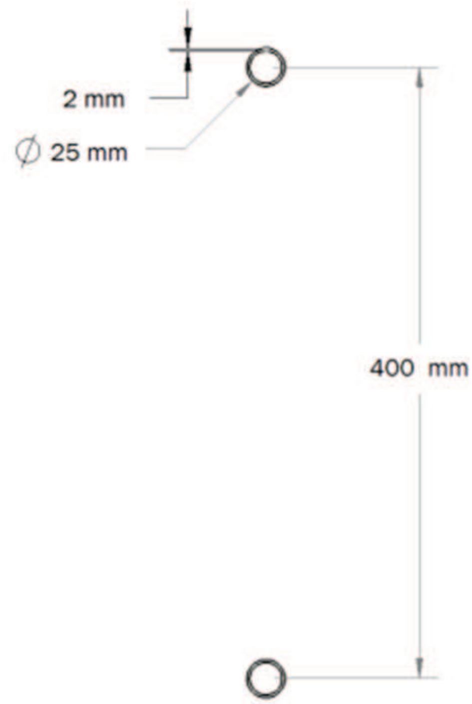

Figure 14: Double Beam Cross Section

The beam effectively acts stronger than the actual frame, thus representing a lower bound on the actual stresses. NOTE: When analyzing this approximation, consider loading the equivalent beam along its neutral axis.

Step 0: What should the solution look like?

This beam has the same length and loading as the beam in Problem 1. Both cases are modeled as single beam. The double beam differs from the single beam in that the moment of inertia is increased, and also in that fact shear deflection is more significant. The following should be true for the solution for the double beam:

- Reaction forces should sum to 1600 N
- Bending stresses should be piecewise-linear, zero at the supports
- Shear force diagram should be piecewise-constant with 3 segments
- Maximum deflection should be negative
- The solution should assume the same form as Problem (1), but all displacements will be lower since the double beam is stiffer than the single beam

Step 1: Element Formulation

The double beam was modeled as a single beam. '`BEAM3`' was used. Beam three has 2 nodes and 3 degrees of freedom at each node: x-displacement, y-displacement and z-rotation. The '`BEAM3`' element can calculate shear deflection using a SHEARZ constant. This constant was taken to be 2 throughout, since the beam is short and the effects of shear deflection cannot be neglected.

The equivalent moment of inertia for the double beam was calculated as follows:

$$I_{xx} = \frac{\pi(D^4 - d^4)}{64} = \frac{\pi\left((25\,\text{mm})^4 - (21\,\text{mm})^4\right)}{64} = 9628.2\,\text{mm}^4 \tag{11}$$

$$I_{xx'} = I_{xx} + Ad^2 = 9628.2\,\text{mm}^4 + 144.5133\,\text{mm}^2\,(200\,\text{mm})^2 = 49773\,\text{mm}^4 \tag{12}$$

$$I_{\text{double beam}} = 2I_{xx}2\left(49773\,\text{mm}^4\right) = \boxed{99466\,\text{mm}^4} \tag{13}$$

Steps 2-6: Discretization, Numerical Formulation, and Boundary Conditions

These steps were identical to the analysis in Part (1) except for the increased moment of inertia, calculated in Eq. (11), and a mesh resolution of one element per millimeter was used based on the results of the convergence study on the single beam. The mesh and boundary conditions are shown in Fig. 15.

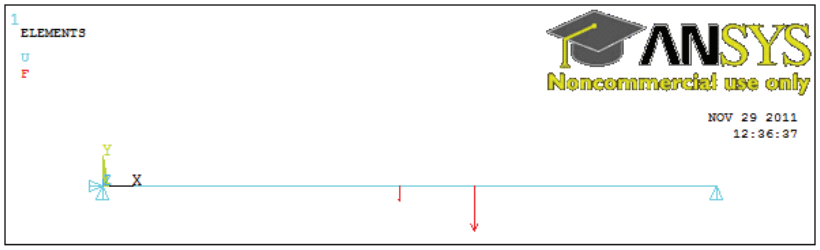

Figure 15: Double Beam Mesh and Boundary Conditions

Steps 7-8: Solution and Results Calculation

This step was again performed by the ANSYS APDL Software for a fine mesh with elements of length 1 mm (chosen based on results of the single-beam study of Part (1). ANSYS computed reactions of 678.79 N and 921.21 N at the left and right support, respectively. The deformed mesh for the double beam is presented in Fig. 16.

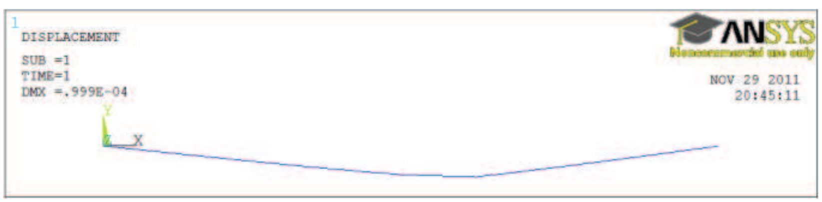

Figure 16: Deformed Mesh, Double Beam Study

Step 9: Validation and Interpretation

The analytical solution to the beam loading presented in Fig. 2 was computed using the principle of superposition and beam deflection equations given by Levinson [3], as derived in Part (1). Comparisons of FEA Results to the analytical solution of displacement, bending moment, direct shear stress, and Von Mises effective stress are shown in Figs. 17 to 20, respectively. Note that the displacement predicted by FEA is about 5 times larger than that predicted by the theory. This is because of the shear deflection constant applied to this analysis, which is not captured by the theory. If the shear deflection constant is set to 0 as in Part (1), the theoretical and FEA results agree for the double beam as expected.

The maximum values for the theory and FEA results are listed in Table 6.

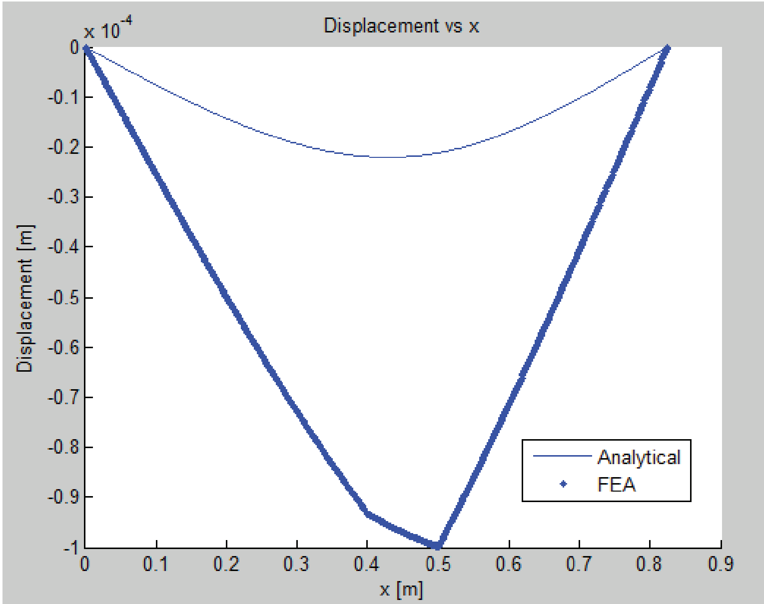

Figure 17: Double Beam Displacement

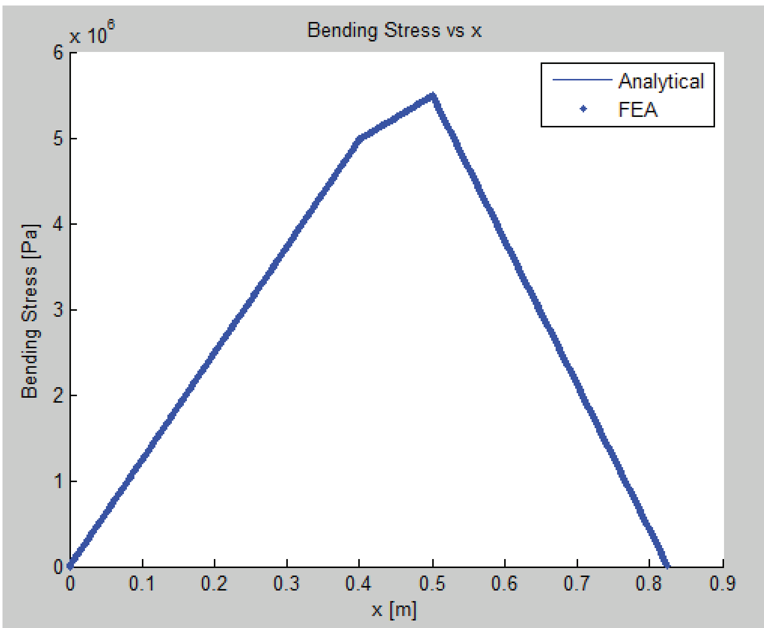

Figure 18: Double Beam Bending Moment

Table 6: Double Beam Maximum Results

Quantity	Value	X location
Deflection	-0.1 mm	0.500 m
Bending Stress	5.4941 MPa	0.500 m
Shear Stress	-3.1874 MPa	0.500 m $\Longrightarrow$ 0.825 m
Von Mises Stress	7.7886 MPa	0.5000 m
Factor of Safety	26.7	

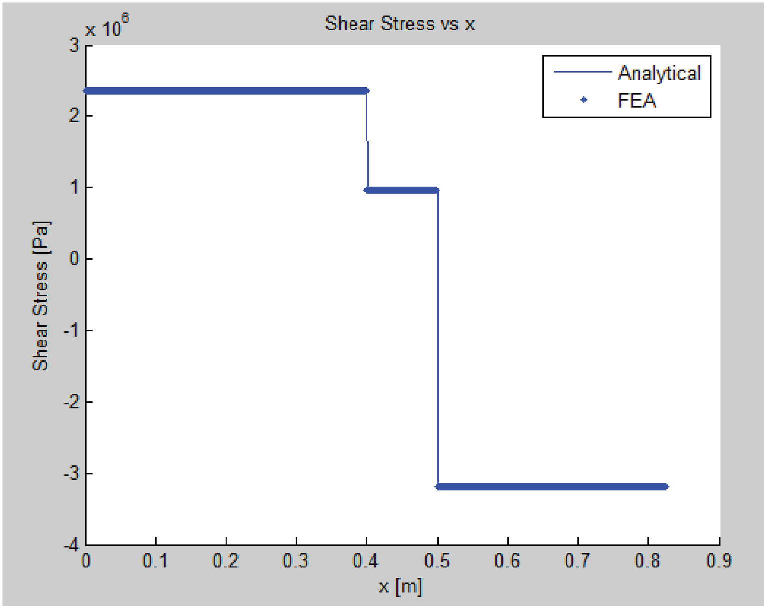

Figure 19: Double Beam Shear Stress

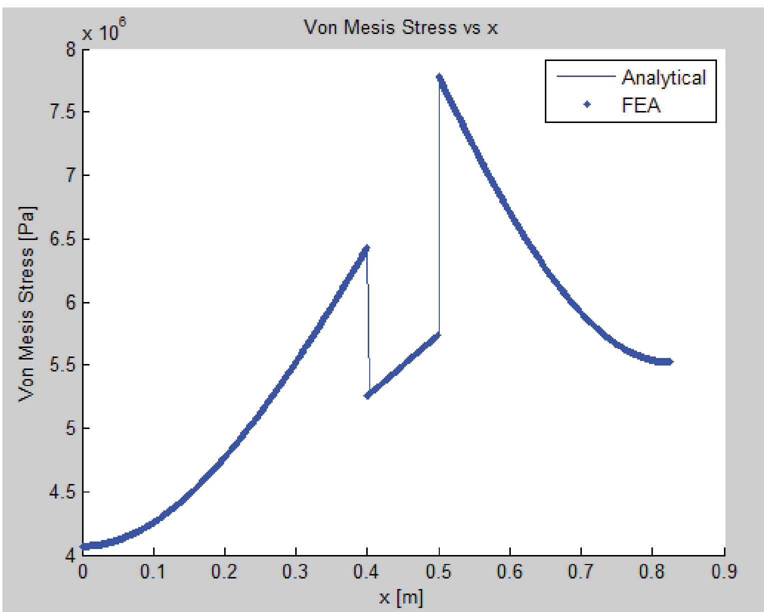

Figure 20: Double Beam Von Mises Effective Stress

(3) Perform design iterations on the actual frame with the goal being to minimize the frame's total weight.

To minimize the frame's weight, a MATLAB script was created to change tubing sizes, run the ANSYS analysis with the appropriate parameters, compare the maximum Von Mises stress to the yield stress to predict factor of safety, and iterate if tubing sizes can be reduced. This script is presented in Appendix A.

The frame was designed to a factor of safety of 2.85, computed using the method described by Pugsley [5] (Characteristics: Material and Workmanship Quality = Very Good, Control Over Loads = Fair, Accuracy of Modeling = Fair, Danger to Personnel = Very Serious, Economic Impact of Failure = Serious)

To simplify the optimization, each frame design was limited to two tube sizes: one for the main triangle and one for the rear triangle. These two sizes were adjusted to provide the lightest frame possible while meeting the required safety factor of 2.85.

Step 0: What should the solution look like?

The deformations and stresses should be bounded by the results of Parts (1) and (2). Reactions for the vertical load case should sum to 1600 N and be equivalent to the reactions calculated in Parts (1) and (2). Reactions for the horizontal load case should sum to 1000 N.

Step 1: Element Formulation

For this analysis, the `PIPE16` element was selected, since it is based on the 3D element `BEAM4` with additional simplifications including automatic calculation of section properties based on direct input of the outer diameter and wall thickness. This element has tension-compression, torsion, and bending capabilities. [6] '`PIPE16`' has two nodes, each of which have 6 degrees of freedom: displacement and rotation in all 3 axes.

Step 2: Discretization

Based on the convergence study results of Part (1), each tube of the frame was discretized with 100 equal-length segments. The frame is shown meshed in ANSYS in Fig. 21. Plain numbers denote keypoints and numbers preceded by '`L`' are lines. Each line was meshed sequentially, so elements 1-100 lie on `L1`, elements 101-200 lie on `L2`, and so on.

Steps 3-5: Constitutive Relationships and Matrix Assembly

As noted before, ANSYS is a small-deformation code, and the stress-strain relation is linear Hooke's law, so the element-wise load-displacement relation is as in Eq. 14:

$$\left[K^{(e)}\right]\left\{\delta^{(e)}\right\} = \left\{f^{(e)}\right\} \tag{14}$$

ANSYS again handles assembling the global stiffness matrix $[K]$ from all individual $\left[K^{(e)}\right]$ matrices.

Step 6: Boundary Conditions

In the full analysis of the bike frame, both load cases described in the introduction are analyzed. Each load case requires a different set of boundary conditions. Table 7 shows the boundary conditions for the horizontal impact load and Table 8 shows the boundary conditions for the vertical riding load. Note that in both load cases, the rear axle (connecting keypoints 5 and 6 in Fig. 21) is assumed to be rigid. The application of the listed boundary conditions allows the solution to be computed in Step 7 below.

The application of the correct boundary conditions was handled in batch mode using an `if` statement which would apply only the set of boundary conditions specified by MATLAB. Implementation details are listed in Appendices A and B.

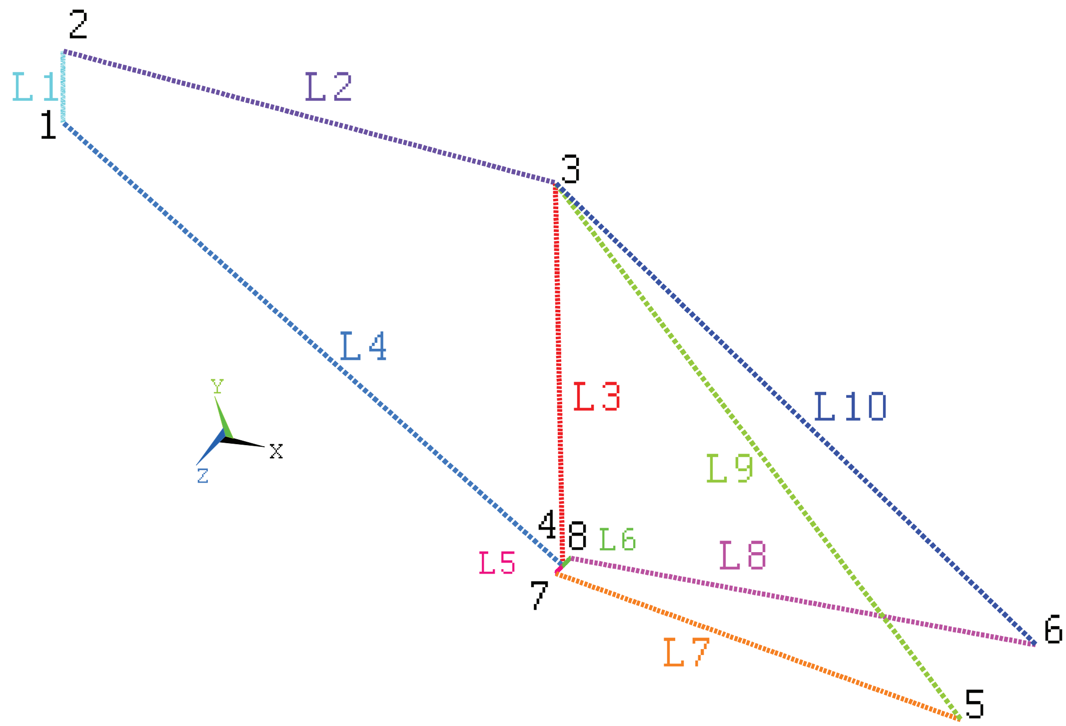

Figure 21: Full Bike Geometry Representation

Table 7: Boundary Conditions for Horizontal Impact Load Case

Boundary Condition	Value
Sliding Front Dropouts	$U_{1,y} = U_{1,z} = 0$
Rear Dropout Ball Joint	$U_{5,x} = U_{5,y} = U_{5,z} = U_{6,x} = U_{6,y} = U_{6,z} = 0$
Frontal Impact Load	$F_{1,x} = 2000\,\mathrm{N}$

Table 8: Boundary Conditions for Vertical Riding Load Case

Boundary Condition	Value
Front Dropout Ball Joint	$U_{1,x} = U_{1,y} = U_{1,z} = 0$
Sliding Rear Dropouts	$U_{5,y} = U_{5,z} = U_{6,y} = U_{6,z} = 0$
Seat Load	$F_{3,y} = -1200\,\mathrm{N}$
Crank Load	$F_{4,y} = -400\,\mathrm{N}$

Steps 7-8: Solution and Results Calculation

This step was performed by the ANSYS APDL Software, executed in batch mode from MATLAB. The initial guess of tube design parameters yielded the results presented in Figs. 22 and 23 for the vertical and horizontal load cases, respectively. For all plots, the deformation is shown in the upper half and the Von Mises effective stress is shown on the lower half. The locations of maxima and minima are annotated in all figures with ‘`MX`’ and ‘`MN`’, respectively. The maximum results for the initial design are summarized in Table 9.

Table 9: Initial Tube Size Results Summary

Quantity	Vertical Load Value	Horizontal Load Value	Units
Deflection	0.5237	0.6605	(mm)
Effective Stress	30.986	47.358	(N/mm^2)
Factor of Safety	6.777	4.434	(-)

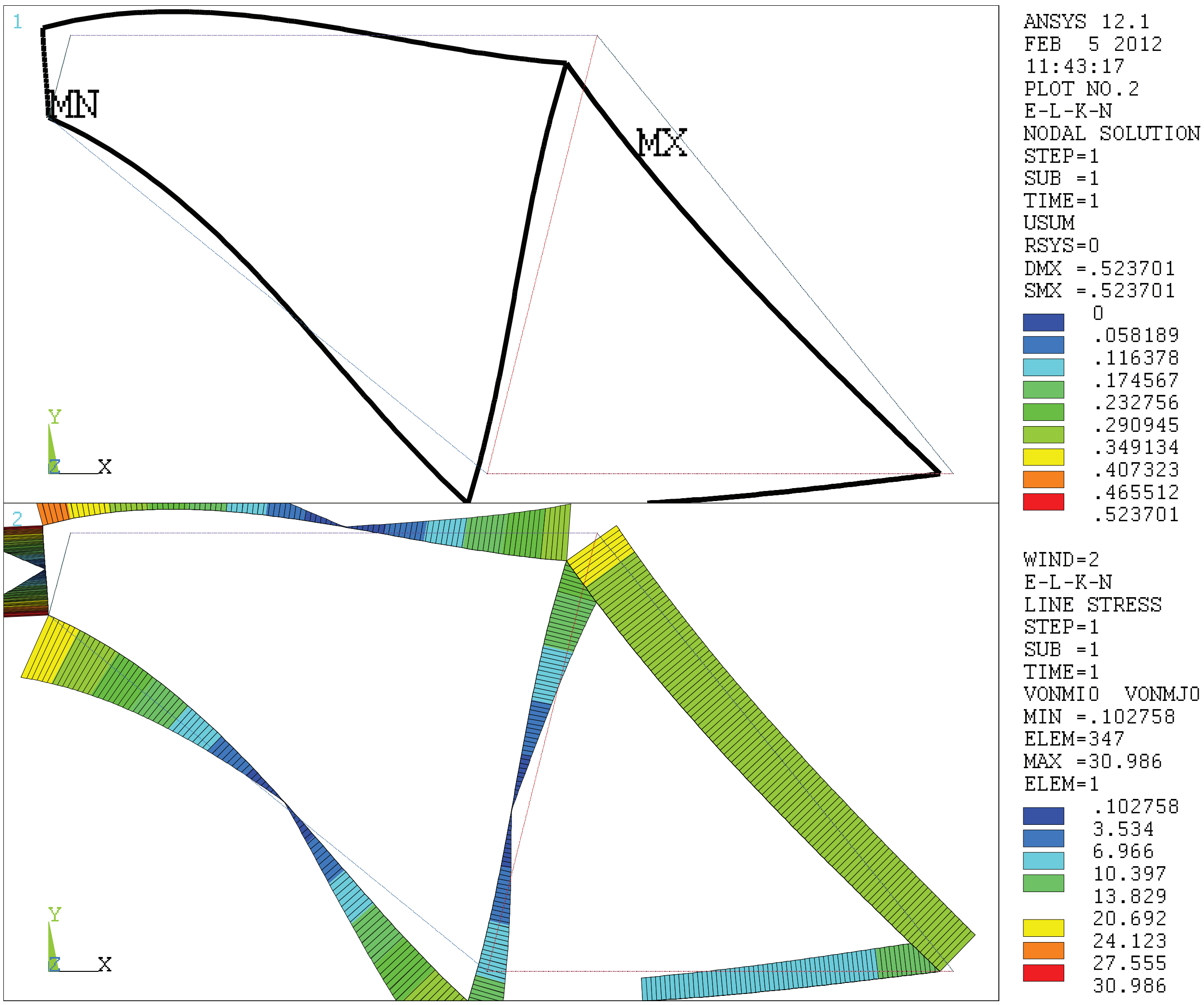

Figure 22: Initial Design Results, Vertical Load Case

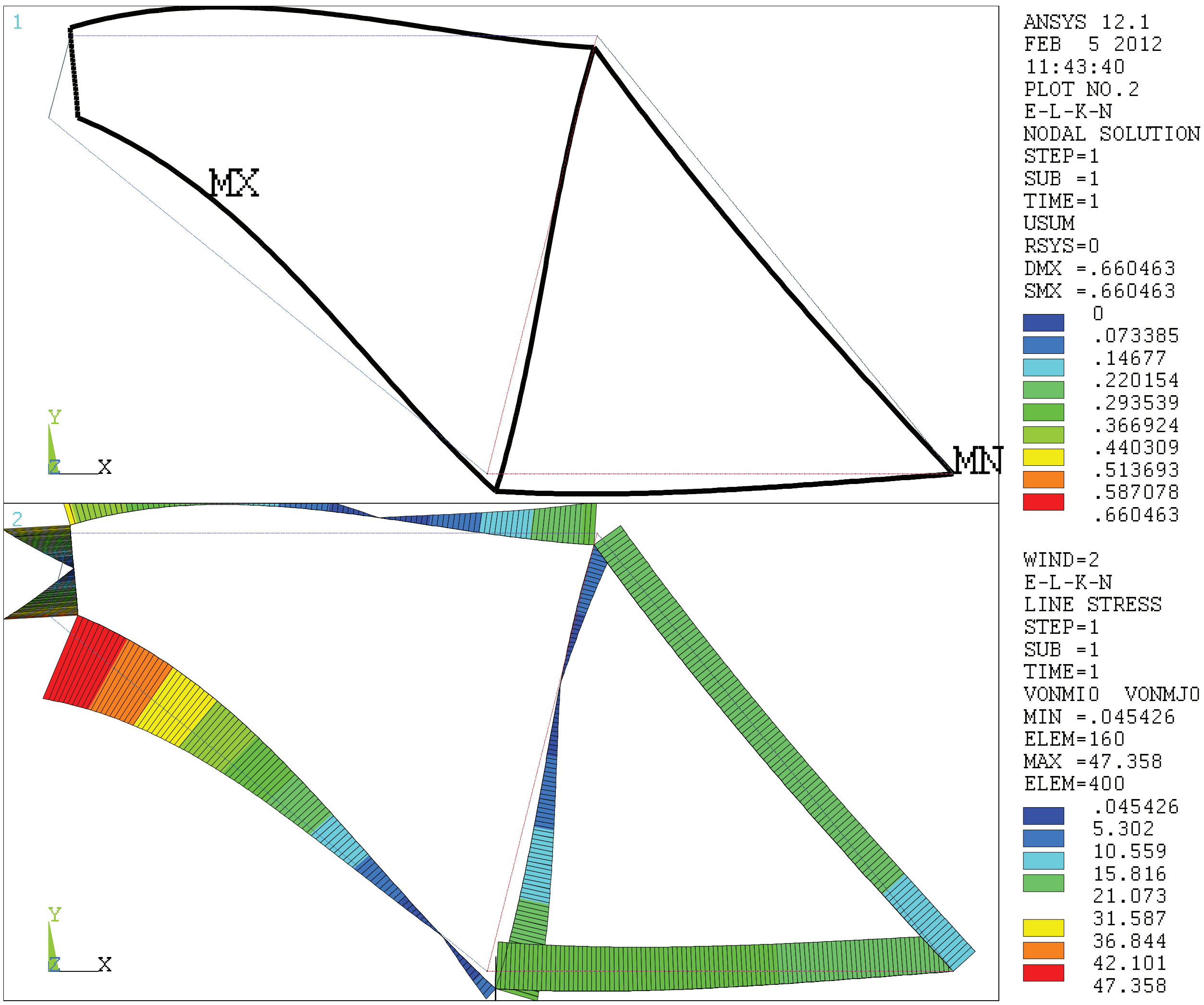

Figure 23: Initial Design Results, Horizontal Load Case

Step 9: Results Interpretation

To find the optimal design, an iterative process was followed, selecting new tube parameters after reviewing the previous results. Appendices A and B present the implementation details.

The iteration path is shown in Table 10. New parameters were chosen until a suitable design meeting all criteria was defined. Complete results for all designs are available from the authors; they have been omitted here for conciseness.

The final design chosen consists of:

- **30 mm dia., 0.85 mm wall tubing for the main triangle, and**
- **11 mm dia., 0.35 mm wall tubing for the rear triangle.**

The vertical load case results are shown in Fig. 24, with the horizontal impact case presented in Fig. 25. A summary of results is shown in Table 11.

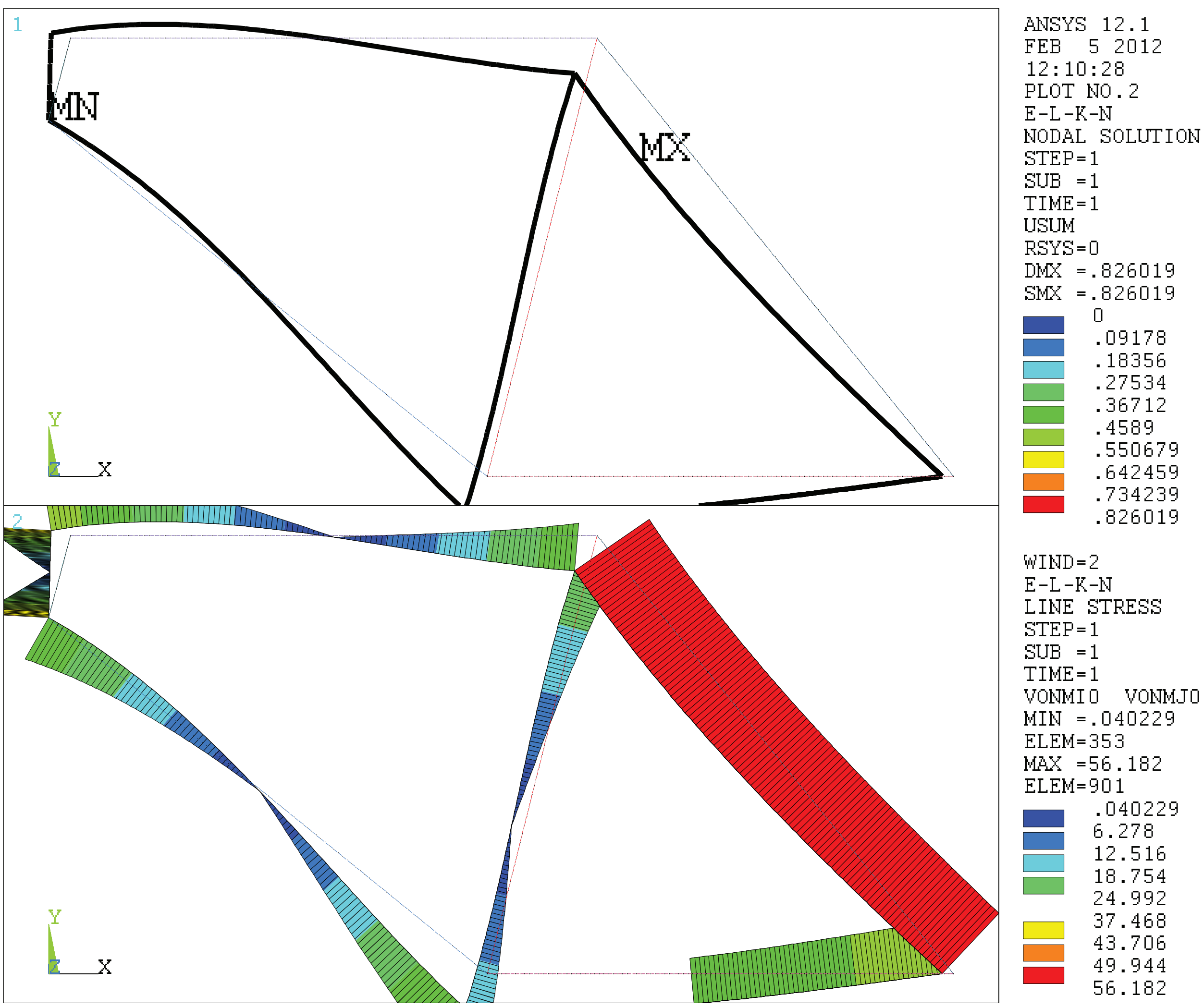

Figure 24: Final Design Results, Vertical Load Case

Table 10: Full Bike Frame Design Iterations

	Main Triangle		Rear Triangle		Maximum Von Mises Stress		Factor of Safety		
#	Outer Dia.	Wall Thickness	Outer Dia.	Wall Thickness	Vert. Load	Horiz. Load	Vert. Load	Horiz. Load	Frame Mass
(-)	(mm)	(mm)	(mm)	(mm)	(N/mm^2)	(N/mm^2)	(-)	(-)	(g)
1	25	2	12	1	30.986	47.358	6.777	4.434	2948.05
2	26	0.75	13	0.5	65.792	103.47	3.192	2.030	1319.03
3	26	1	12	0.5	50.779	79.495	4.136	2.642	1588.23
4	30	1	12	0.5	37.836	61.375	5.550	3.422	1786.04
5	30	0.75	12	0.5	49.230	80.177	4.266	2.619	1436.78
6	30	0.8	12	0.5	46.379	75.473	4.528	2.782	1507.13
7	30	0.85	11	0.5	43.869	71.313	4.787	2.945	1546.63
8	30	0.85	10	0.25	83.562	96.017	2.513	2.187	1374.50
9	30	0.85	11	0.35	56.182	71.311	3.738	2.945	1453.49

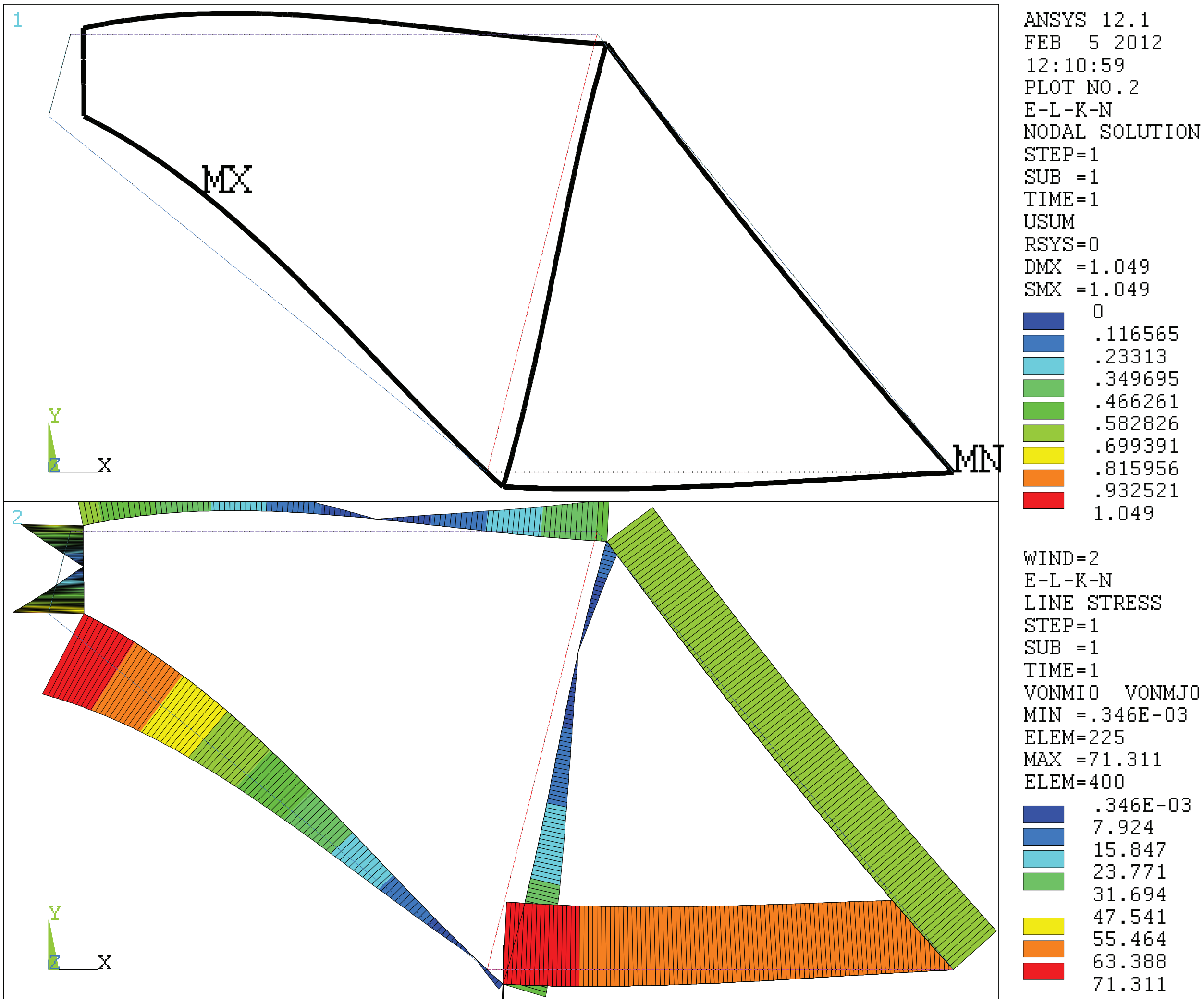

Figure 25: Final Design Results, Horizontal Load Case

Table 11: Final Design Results Summary

Quantity	Vertical Load Value	Horizontal Load Value	Units
Deflection	0.8260	1.049	(mm)
Effective Stress	56.182	71.311	(N/mm^2)
Factor of Safety	3.738	2.945	(-)

(4) Reconsider and revisit the single tube crude approximation from Part (1). Re-perform your analysis on a 3D finite element model using solid elements. Compare your results and comment fully on the similarities and differences between the 1D and 3D models. Comment fully on the differences in how you applied the relevant boundary conditions.

Step 0: What should the solution look like?

This solution to this case should be comparable to the analytical solution for the single 1D beam element. Like the analytical solution and the 'BEAM3' FEA solutions, in the 3D element solution point loads and point supports will be used. Near the applied point boundary conditions there will be very high localized stresses.

Step 1: Element Formulation

The element type 'SOLID187' was selected for this analysis. 'SOLID187' is a 10-noded tetrahedral element. The description of this element in ANSYS Element Reference [7] is as follows, with nodes and degrees of freedom as shown in Fig. 26:

> "The SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. . . "

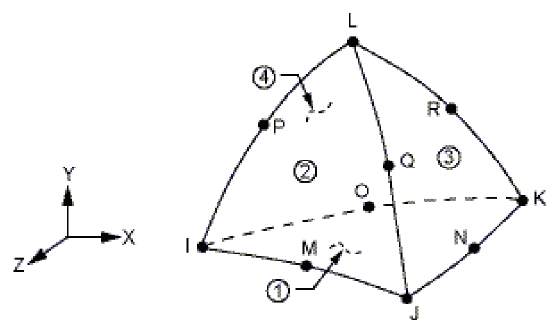

Figure 26: SOLID187 Element Configuration [7]

SOLID187 was the default element selected by ANSYS Workbench in the 'automatic' mode. The user can select tetrahedral or hexahedral elements.

Step 2: Discretization

The pipe geometry was created in Workbench. In anticipation that the node limit of the MSOE ANSYS license could be an issue, symmetry was exploited, with only half of the pipe modeled. The points that were to have point loads applied were specified prior to meshing to ensure that there would be a node at those points after meshing. Two meshes were created: one with a base element size of 10mm, as shown in Fig. 27 and another with a base element size of 4mm, shown in Fig. 28. Note that with a base element size of 3.5 mm, the node limit was exceeded.

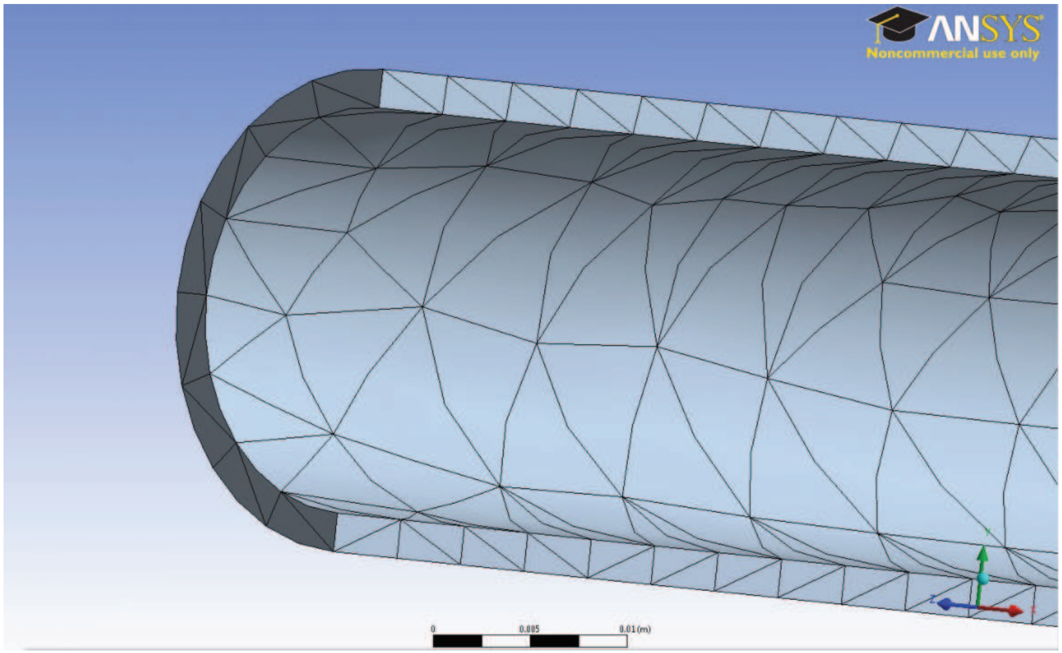

Figure 27: Solid Element Mesh with 10 mm Base Size (Note ≈3:1 Aspect Ratio)

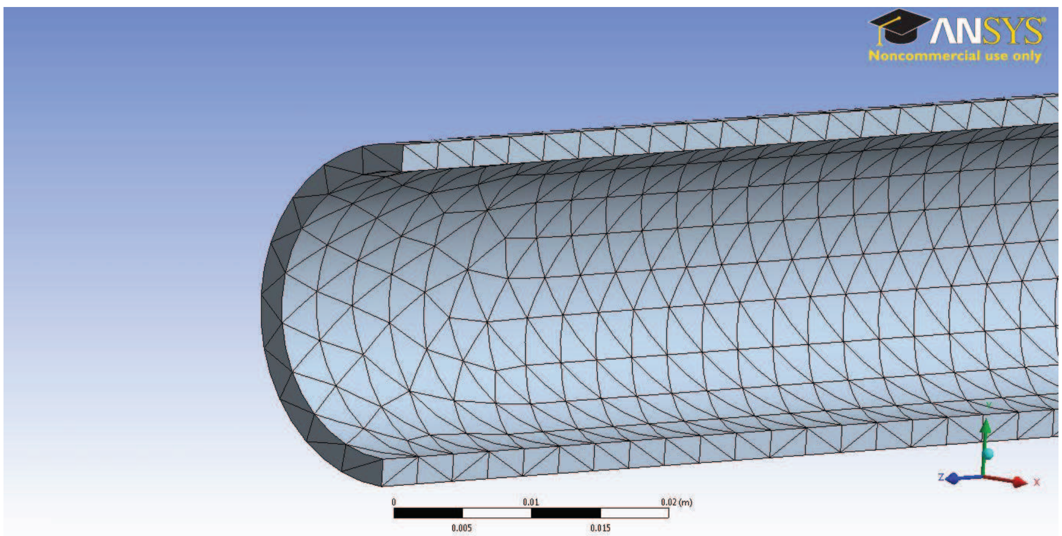

Figure 28: Solid Element Mesh with 4 mm Base Size

Steps 3-5: Constitutive Relationships and Matrix Assembly

As noted before, ANSYS is a small-deformation code at default settings, and the stress-strain relation is linear Hooke's law, so the element-wise load-displacement relation is as in Eq. 15:

$$\left[K^{(e)}\right]\left\{\delta^{(e)}\right\} = \left\{f^{(e)}\right\} \tag{15}$$

ANSYS again handles assembling the global stiffness matrix $[K]$ from all individual $\left[K^{(e)}\right]$ matrices.

Step 6: Boundary Conditions

Boundary conditions were applied to the pipe as shown in Fig. 2. The x-displacement on the symmetry plane was set to zero. The boundary conditions are shown in Fig. 29 and are explained in Table 12.

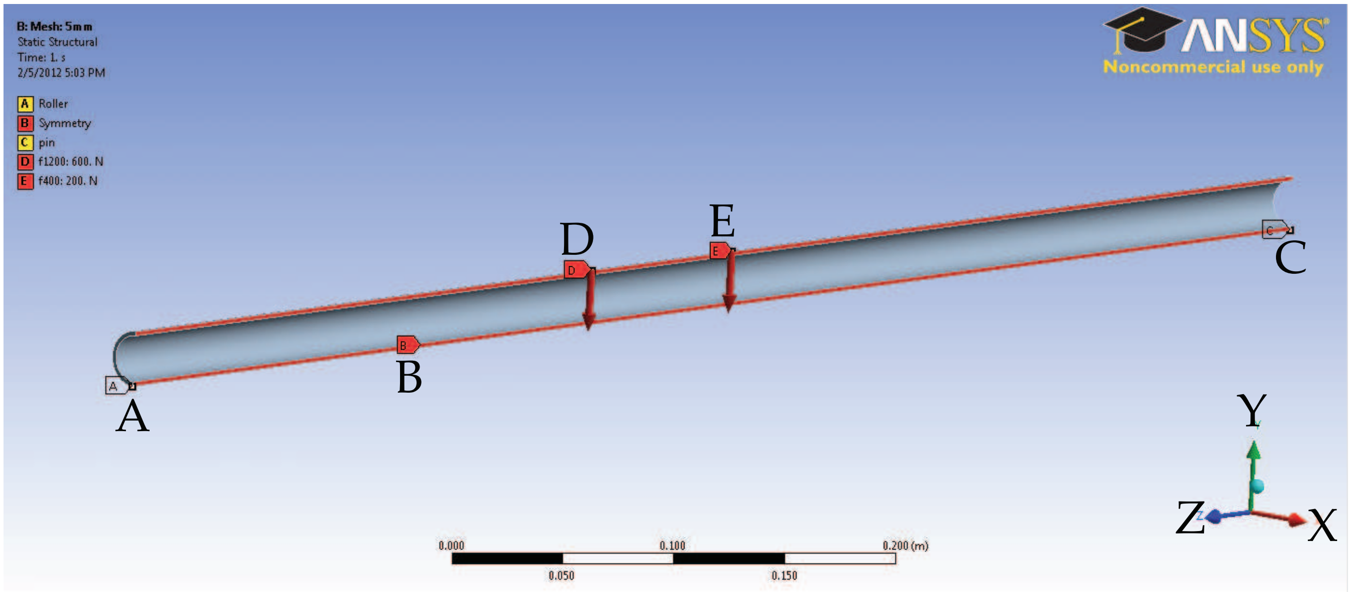

Figure 29: 3D Solid Element Boundary Conditions

Table 12: Boundary Conditions for 3D Solid Element Analysis

Location	Boundary Condition
A	Roller Boundary Condition $\delta_x = 0$ $\delta_y = 0$
B	Symmetry Boundary Condition $\delta_x = 0$
C	Pin Boundary Condition $\delta_x = 0$ $\delta_y = 0$ $\delta_z = 0$
D	Half of the 1200 N load applied at a single node $f_y = -600\,\mathrm{N}$
E	Half of the 400 N load applied at a single node $f_y = -200\,\mathrm{N}$

Steps 7-8: Solution and Results Calculation

The ANSYS Workbench results for both meshes are listed in Table 13. The Von Mises effective stress and

Table 13: 3D Solid Element Results

Quantity	Coarse Mesh	Fine Mesh
Maximum deflection	26.7mm	26.8 mm
Maximum von-Mises stress	587 MPa (at roller)	863 MPa (at roller)
Maximum Bending stress	414 MPa (near 1200 N load) 388 MPa (away from 1200N load)	456 MPa (near 1200N load) 388 MPa (away from 1200N load)

deformation for the coarse mesh are shown in Figs. 30 and 31, respectively.

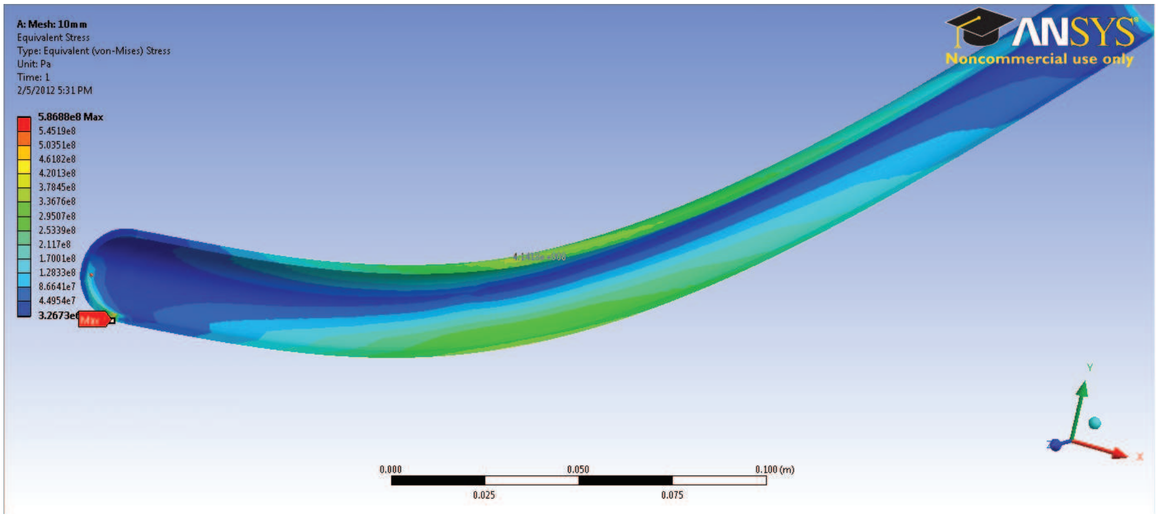

Figure 30: Coarse Mesh Von Mises Stress

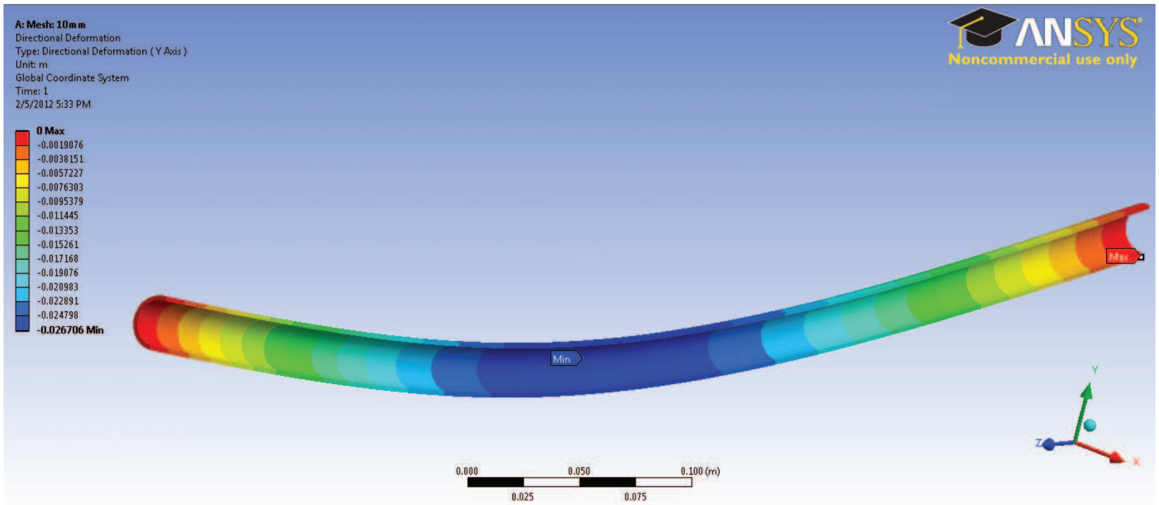

Figure 31: Coarse Mesh Displacement

The Von Mises effective stress and deformation for the fine mesh are shown in Figs. 32 and 33, respectively.

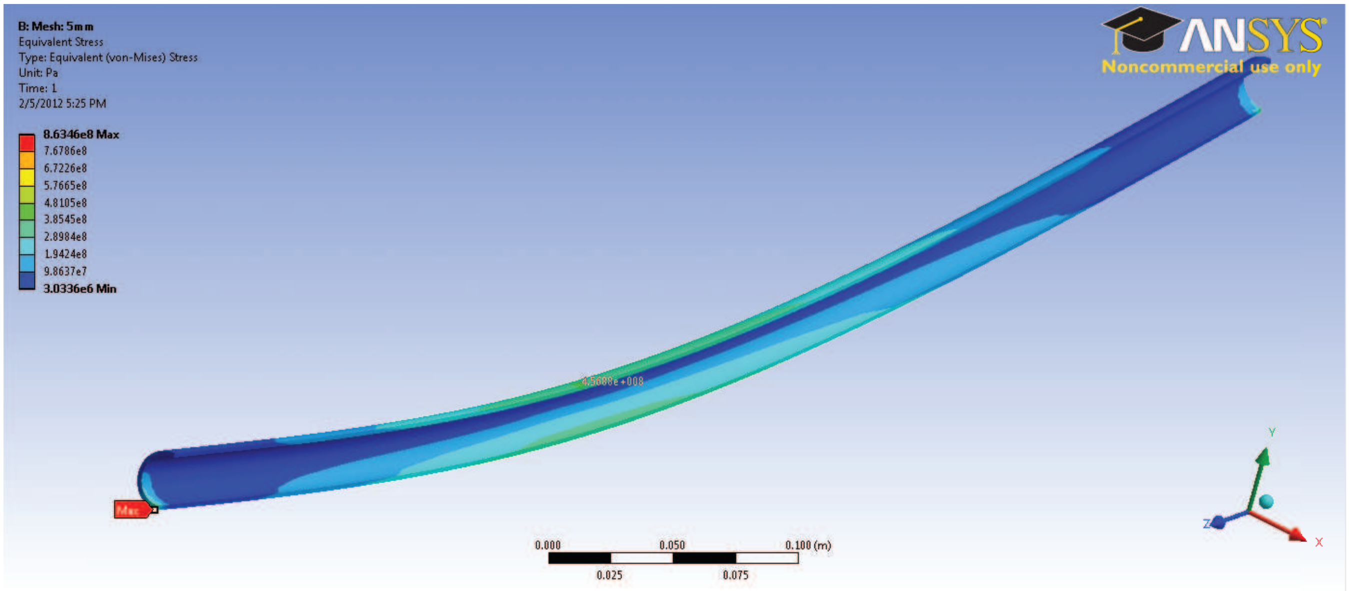

Figure 32: Fine Mesh Von Mises Stress

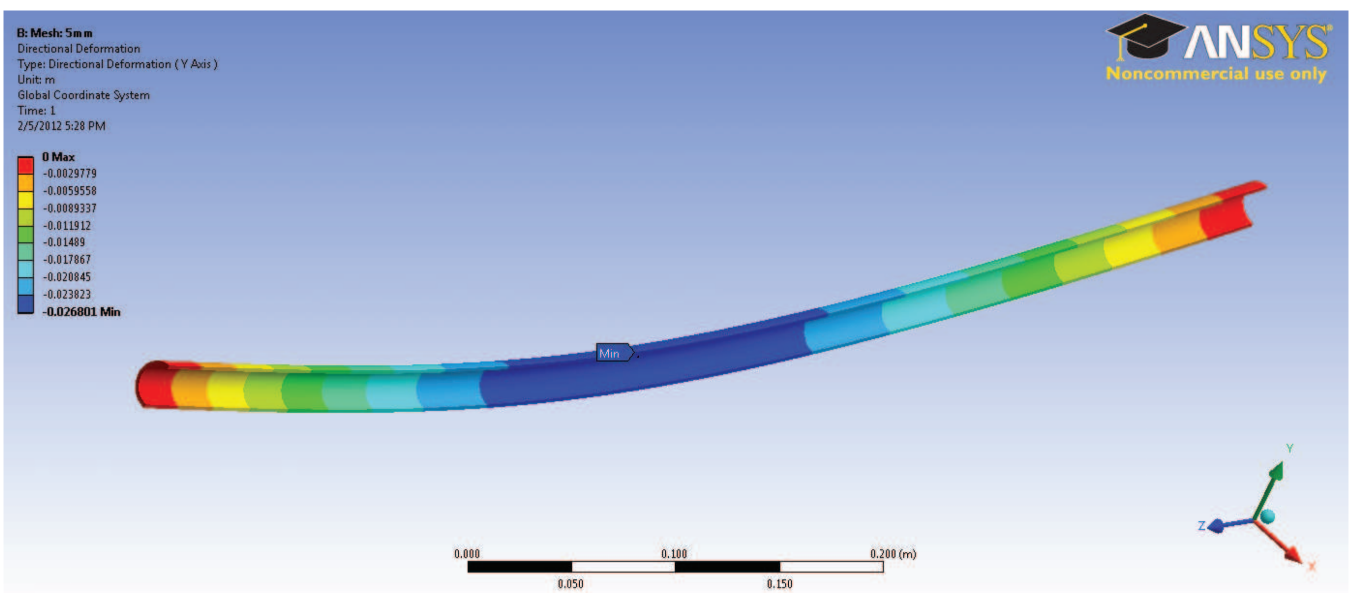

Figure 33: Fine Mesh Displacement

Step 9: Validation and Interpretation

The deflection values correspond with the analytical solution to within 2%. The maximum stress values found are much greater than those found analytically. This is becase of how the boundary conditions are applied. The pin and roller boundary conditions are displacement boundary conditions applied at single nodes. Similarly, the applied forces are applied as point loads on a single node. It was shown that as the mesh resolution increases, the stresses near these point loads increases. Away from the point loads, the stress solution agreed with the analytical solution to within 0.1%, consistent with Saint Venant's Principle.

References

[1] Prantil, V., 2011. ME460 Course Project.
[2] Gessler, P. D., and Lange, A. J., 2011. Beam Element Lab.
[3] Levinson, I. J., 1971. *Statics and Strength of Materials*, 16th ed. Prentice-Hall, Inc., Englewood Cliffs, NJ.
[4] Budynas, R., and Nisbett, J., 2011. *Shigley's Mechanical Engineering Design*, 9th ed. McGraw Hill, New York.
[5] Pugsley, A., 1966. *The Safety of Structures*. Edward Arnold, London.
[6] ANSYS, Inc., 2011. PIPE16: Elastic Straight Pipe Element Reference.
[7] ANSYS, Inc., 2011. SOLID187: 3-D 10-Node Tetrahedral Structural Solid Element Reference.

A MATLAB Script File

The following MATLAB m-script was used to command the parameters for each ANSYS simulation. The initial guess for tube diameters is located at the beginning of the file, and the user is prompted to enter a new set of parameters after reviewing the factors of safety of the previous analysis.

```
% --------------------------------|
% MATLAB Script for Bike Analysis |
% Paul Gessler, Adam Lange        |
% 2012-02-04                      |
% --------------------------------|

%% Cleanup from last execution
clear all; close all; clc

%% Set parameters
NSEG = 100;     % Number of elements per line (-)
% Initial guesses for tube sizing ...
OD1 = 25;       % Main Triangle Outer Diameter (mm)
WT1 = 2;        % Main Triangle Wall Thickness (mm)
OD2 = 12;       % Rear Triangle Outer Diameter (mm)
WT2 = 1;        % Rear Triangle Wall Thickness (mm)

N = 2.85;       % Design Factor of Safety (-)
SigY = 210;     % Yield Strength (N/mm^2)
rho = 0.00258;  % Density of Aluminum (g/mm^3)

frntL = 1525.32;% Length of tubes in front triangle (mm)
rearL = 1887.73;% Length of tubes in rear triangle (mm)

iterMAX = 1000;    % Maximum number of iterations (-)
Nmat = zeros(iterMAX,2);    % Factor of Safety (-)
frntCSA = zeros(iterMAX,1); % Front Triangle CSA (mm^2)
rearCSA = zeros(iterMAX,1); % Rear Triangle CSA (mm^2)
vol = zeros(iterMAX,1);     % Total Tube Volume (mm^3)
mass = zeros(iterMAX,1);    % Total Frame Mass (g)

%% Build system command string parts
ProgStr = 'C:\"Program Files"\"ANSYS Inc"\v121\ansys\bin\WINX64\ANSYS121.exe ';
OptStr  = ' -p aa_t_i -b -t6 -i FullBike.in -o FullBike.out';

%% Optimization Routine
for iter = 1:iterMAX            % Iterate to minimize tube weight
    for LOADCASE = 0:1          % Loop over horizontal and vertical loads
        CmdStr  = [ProgStr '-LOADCASE ' num2str(LOADCASE) ' -NSEG ' ...
            num2str(NSEG) ' -OD1 ' num2str(OD1) ' -WT1 ' num2str(WT1) ...
            ' -OD2 ' num2str(OD2) ' -WT2 ' num2str(WT2) OptStr];

        FAIL = system(CmdStr);  % Run ANSYS FEA

        if FAIL ~= 8            % Check for successful run completion
            fprintf('WARNING: The ANSYS solver did not complete as expected!')
            OD1, WT1, OD2, WT2, LOADCASE  % Dump parameters for diagnosis
            break;
        else                    % Successful run
            if LOADCASE == 0
```

```
                varStr = 'v';   % Vertical Load Case (Riding)
            else
                varStr = 'h';   % Horizontal Load Case (Impact)
            end
            % Rename output and image files with descriptive strings,
            % directory cleanup after successful analysis run.
            FnameStr = strcat('FullBike',varStr,'OD1-',num2str(OD1),...
                '-WT1-',num2str(WT1),'-OD2-',num2str(OD2),'-WT2-',...
                num2str(WT2));
            RenameCmd = ['rename FullBike001.eps ' FnameStr '.eps'];
            CopyCmd = ['copy FullBike.out ' FnameStr '.out'];
            dos(RenameCmd); dos(CopyCmd); dos('del FullBike000.eps');

            % Open output file for examination
            open([FnameStr '.out']);
            Seff = input('Input max Von Mises stress from output file:');

            Nmat(iter,LOADCASE+1) = SigY/Seff; % Calculate factor of safety
        end
    end

    % Output factors of safety for each load case
    fprintf(['The factor of safety (vertical loading) for iteration'...
        '%i was %f\n'],iter,Nmat(iter,1))
    fprintf(['The factor of safety (horizontal loading) for iteration'...
        '%i was %f\n'],iter,Nmat(iter,2))

    % Compute main and rear triangle tube CSAs (mm^2)
    frntCSA(iter) = pi()*(OD1^2-(OD1-2*WT1)^2);
    rearCSA(iter) = pi()*(OD2^2-(OD2-2*WT2)^2);

    % Compute total tube volume (mm^3)
    vol(iter) = frntL*frntCSA(iter) + rearL*rearCSA(iter);

    mass(iter) = rho*vol(iter); % Compute total frame mass (g)
    fprintf('The frame mass for iteration %i was %f g.\n',iter,mass(iter))

    if iter < iterMAX
        choice = input('Do you want to continue? (Enter y/n)')
        if strcmp(choice,'y') % Input new tube parameters (mm)
            OD1 = input('Select new value for OD1 (mm):');
            WT1 = input('Select new value for WT1 (mm):');
            OD2 = input('Select new value for OD2 (mm):');
            WT2 = input('Select new value for WT2 (mm):');
        else
            break;
        end
    end
end
```

B ANSYS APDL Input File

The following input file was used to run the full bike analysis. ANSYS was batch-executed via a MATLAB script (see Appendix A) which performed the optimization by setting several configurable parameters.

```
/batch               ! Tell ANSYS this is a batch file
! ---------------------------------------------------------------------------|
! Command File for Complete Bike Geometry                                    |
! Paul Gessler, Adam Lange                                                   |
! 2012-02-04                                                                 |
! Configurable parameters in this file: OD1, OD2, WT1, WT2, NSEG, LOADCASE|
! Set from MATLAB system call for optimization                               |
! Adapted from University of Alberta 3D Space Frame Example                  |
! URL: http://www.mece.ualberta.ca/tutorials/ansys/CL/CBT/Bike/Print.html |
! ---------------------------------------------------------------------------|

/filename,FullBike   ! Name the analysis
/prep7               ! Enter the pre-processor

! Define Keypoints
! k,key-point number,x-coord,y-coord,z-coord
K,1,   0, 325,   0   ! Fixed Geometry for the Frame
K,2,  20, 400,   0   ! Units: mm throughout
K,3, 500, 400,   0
K,4, 400,   0,   0
K,5, 825,   0,  50
K,6, 825,   0, -50
K,7, 400,   0,  10
K,8, 400,   0, -10

! Define Lines Linking Keypoints
! l,keypoint1,keypoint2
L,1,2                ! Head Tube                    \
L,2,3                ! Top Tube                      |
L,3,4                ! Seat Tube                     |_ Main Triangle (L1-L6)
L,4,1                ! Down Tube                     |
L,7,4                ! Bottom Bracket Shell L Half  |
L,4,8                ! Bottom Bracket Shell R Half /
L,7,5                ! L Chainstay                  \
L,8,6                ! R Chainstay                   |_ Rear Triangle (L7-L10)
L,3,5                ! L Seatstay                    |
L,3,6                ! R Seatstay                   /

! Define Element Type
ET,1,pipe16          ! PIPE16 Element
KEYOPT,1,6,1         ! Include Force and Moment Output

! Define Real Constants
! r,real set number,outside diameter,wall thickness
R,1,OD1,WT1          ! first set of real constants - for main triangle
R,2,OD2,WT2          ! second set of real constants - for rear triangle

! Define Material Properties
MP,EX,1,70000        ! mp,Young’s modulus,material number,value (N/mm^2)
MP,PRXY,1,0.33       ! mp,Poisson’s ratio,material number,value (-)
```

```
! Define the number of elements each line is to be divided into
! lesize,line number(all lines),size of element, arc size, number of divisions
LESIZE,ALL, , ,NSEG

! Line Meshing
REAL,1                 ! activate real property set #1
LMESH,1,6,1            ! mesh the Main Triangle
REAL,2                 ! activate real property set #2
LMESH,7,10,1           ! mesh the Rear Triangle

FINISH                 ! Finish pre-processing

/SOLU                  ! Enter the solution processor

ANTYPE,0               ! Analysis type,static

! Define Displacement Constraints on Keypoints   (dk command)
! dk,keypoint,direction,displacement,,,direction,direction
! fk,keypoint,direction,force

! Vertical Loads Case
*if,LOADCASE,EQ,0,then
DK,1,UX,0,,,UY,UZ      ! Ball Joint @ Front Dropout
DK,5,UY,0,,,UZ         ! Sliding BC @ Rear Dropout
DK,6,UY,0,,,UZ         ! NOTE: We assume a rigid axle
FK,3,FY,-1200          ! Seat Load with Dynamic Load Factor = 2
FK,4,FY,-400           ! Crank Load with Dynamic Load Factor = 2

! Horizontal Impact Case
*elseif,LOADCASE,EQ,1,then
DK,1,UY,0,,,UZ         ! Sliding BC @ Front Dropout
DK,5,UX,0,,,UY,UZ      ! Ball Joint @ Rear Dropouts
DK,6,UX,0,,,UY,UZ
FK,1,FX,2000           ! Impact Load with Dynamic Load Factor = 2
*endif

SOLVE                  ! Solve the problem

FINISH                 ! Finish the solution processor

SAVE                   ! Save your work to the database

/post1                 ! Enter the general post processor

PSCR,COLOR,2           ! Set to 256-color plots
PSCR,HIRES,1           ! Set high resolution output
PSCR,ROTATE,0          ! Set image rotation
PSCR,LWID,1            ! Set line weight
PSCR,PAPER,A4,Portrait ! Set paper size and orientation
/GFILE,2400            ! Set DPI of image output
/SHOW,PSCR,,0,         ! Output sequential plots

/WIND,ALL,OFF          ! Deactivate all windows
/WIND,1,TOP            ! Window 1 - Top of screen
/WIND,2,BOT            ! Window 2 - Bottom of screen
```

```
! Set up Element Table information
!   etable,arbitrary name,item name,data code number

! For the VonMises (or equivalent) stresses at angle 0 at both ends of the
! element (node i and node j);
etable,vonmi0,nmisc,5
etable,vonmj0,nmisc,45

GPLOT                               ! Open plots
/GCMD,1, PLNSOL,U,SUM,0,1           ! Plot displacement solution
/GCMD,2, PLLS,vonmi0,vonmj0,,1      ! Plot Von Mises Effective Stress Solution
/REPLOT                             ! Update plot files

PRNSOL,DOF,                         ! Output displacement solution
PRESOL,NMISC,5                      ! Output Von Mises Effective Stress
PRESOL,NMISC,45                     ! (for factor of safety calculation)
```