Spring Powered Ping Pong Launcher

Mechanical Design, Stress Analysis, and Component Verification

Project Overview

Designed and built a spring-powered ping pong launcher as part of a team-based machine design project. The system was modeled in SolidWorks and analyzed to verify critical components, including spring design from wire gauge selection, shaft strength under combined bending and torsion, and bearing life. A parametric Excel design tool has been developed to streamline spring design, other component verification, and rapid design iteration.

The final launcher was manufactured and assembled, and the team achieved **first place in a 14-team performance-based tournament**, launching ping pong balls into a cup from a distance of 8 feet. Even went on to outperform the professor in a Human vs Machine Competition.

Individual Contributions

- Led the design and verification of the compression spring, including stress, manufacturability, and buckling stability analysis
- Performed the shaft fatigue analysis under combined bending and torsion
- Developed the parametric Excel design and verification tool for springs, shafts, and bearings
- Supported CAD design and system testing

Final Design that won 1st Place in the class competition

CAD model illustrating launcher geometry and cocking mechanism; drive chain and servo shaft simplified for clarity.

Parametric Design & Verification Tool (Excel)

Purpose of the Tool

A parametric Excel-based design tool was developed to support rapid sizing and verification of critical launcher components. The tool enables design inputs, including material properties, geometry, and applied loads, to be modified. All while automatically updating governing stress, safety factors, and pass/fail checks for the spring, shaft, and bearings.

Tool Structure

- **Input Sheets:** User-defined geometry, material properties, and load cases
- **Spring Calculations**: Constraint-driven spring sizing from a wire gauge and required deflection (cock back distance)
- **Shaft and Bearing Calculations:** Combined stress, fatigue safety factor, and bearing life verification
- **Summary Sheet:** High-level design validation with clear pass/fail indicators

Inputs Sheet

	A	B	C	D	E	F
1	**Section 1: Spring Selection**			**Section 2: Loads and Use Conditions**		
2	Wire Diameter (from Guage)	0.115	Inches	Max Spring Force	29	lbf
3	Material	302 Stainless Steel	N/A	Fractional Overrun	0.15	N/A
4	Shear Yield Strength	115000	psi	End Type	Squared and Grounded	N/A
5	Desired Factor of Safety	1.5	N/A	End Condition	Fixed Ends	N/A
6	Shear Modulus (Material property)	11200000	psi			
7	Gauge	29	N/A			
8	Required Spring Defelection (Cocking stroke)	3	Inches			

G	H	I	J	K	L
Section 3: Shaft			**Section 4: Bearing**		
Shaft Diameter	0.5	inches	Type	Ball Bearings	N/A
Shaft Length	1	inches	Life Exponent	3	N/A
Apllied Torque	30.3	lbf-in	Dynamic Load Rating	320	lbf
Applied Radial Load (Servo/Motor Weight)	2.5	lbf	Operating Speed	500	rpm
Offset Distance (Moment arm)	0.039	inches	Desired Life	4000	Hours
Ultimate Tensile Strength	72.519	kpsi	External Radial Load (Resultant side load on shaft)	60	lbf
			Load Loaction from Bearing A	1.2	inches
			Distance Between Bearings	3	inches
			Axial Load (No Thrust Load Assumed)	0	lbf

User-defined inputs for material selection, geometry, and loading conditions that drive automated component verification

Spring Sizing & Stability Verification (Excel)

Constraint-driven design based on material limits and required deflection

Overview

The compression spring was sized and designed using a constraint-driven approach rather than verification post-design. From the inputs page of the Excel, the wire diameter from the Guage and material properties are selected for the design. For the calculations, allowable shear stress and manufacturability constraints are determined. Spring geometry, active coil count, and free length were then calculated to meet the required cocking stroke while satisfying stress limits and buckling stability criteria.

Spring Calculations			
Section 1: Material Limit	Allowable Shear Stress	76666.67	psi
Section 2: Load with Overrun	Load Stress Factor (β)	6424.81	psi
Section 3: Spring Index	Curvature Coeffiecent, C	10.59	N/A
	Check	Pass	
Section 4: Geometry from C	Diameter, D	1.22	inches
	Inside Diamter, ID	1.10	inches
	Outside Diameter, OD	1.33	inches
Section 5: Stress Verification	Wahl Correction Factor	1.13	N/A
(Stress sized to meet desired safety factor)	Max Shear Stress	76666.67	psi
	Computed Safety Factor	1.50	N/A
Section 6: Deflection & Active Coils	Number of Active Coils	14.03	N/A
	Calculated Spring Rate	9.67	lbf/in
Section 7: Free Length and Stability	Total Coils	16.03	N/A
(Based on Spring End Type)	Solid Length	1.84	inches
	Free Length	5.29	inches
	Free Length for Stability	6.40	inches
	Check	Pass	

Engineering Decisions

- Spring index verified within recommended manufacturability ($4 \leq C \leq 12$)
- Shear Stress sized to meet the desired factor of safety of 1.5
- Active coil count determined from the required cocking stroke
- Buckling stability verified for the calculated free length (Free length ≤ Free length for Stability)

Structural Finite Element Analysis – Flying Vehicle Frame

Static stress, deformation, and safety factor evaluation

Problem Statement

A static structural finite element analysis (FEA) was conducted on the flying car frame to evaluate stress distribution, deformation, and overall structural safety under a representative rider load. The objective of the analysis was to identify critical stress regions and verify that the frame met minimum strength and stiffness requirements before further design development.

Analysis Setup and Assumptions

- Frame modeled as a rigidly connected tubular structure
- Material properties are assigned based on structural steel
- A vertical load of 200 lbf is applied to the seat location to approximate the driver's weight
- Base mounting points constrained to represent ground support
- Linear static analysis is used to evaluate stresses and deflections

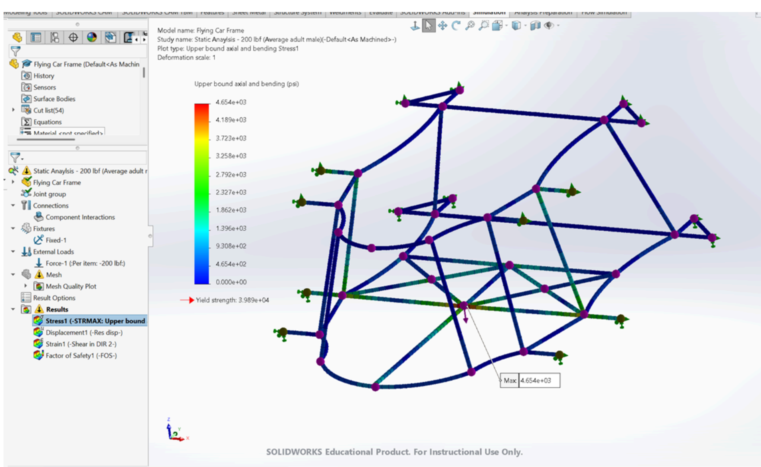


Figure 1: *Von Mises stress distribution under applied rider load.*

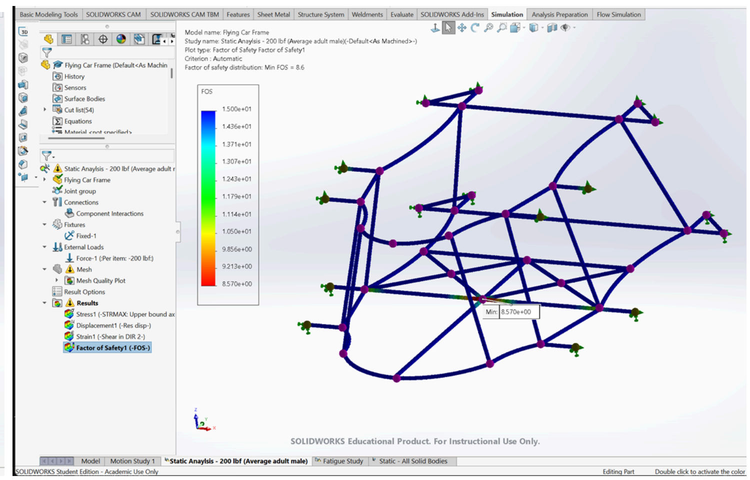


Figure 2: *Factor of safety distribution highlighting critical frame members*

Key Results

- Maximum stress accrued near joint intersections and load transfer points
- Overall deformation remained within the acceptable limits for structural integrity
- Minimum factor of safety exceeded the target design requirement
- No global structural instability observed under applied loading

Engineering Takeaways

- Load paths through the frame were validated, confirming effective force distribution
- Stress concentrations informed potential reinforcement locations in future iterations
- The elevated factor of safety reflects conservative loading assumptions and a stiffness-dominated frame geometry rather than over-stress
- Results provided confidence to proceed with fatigue and dynamic analyses

Fatigue Analysis – Frame Durability Assessment

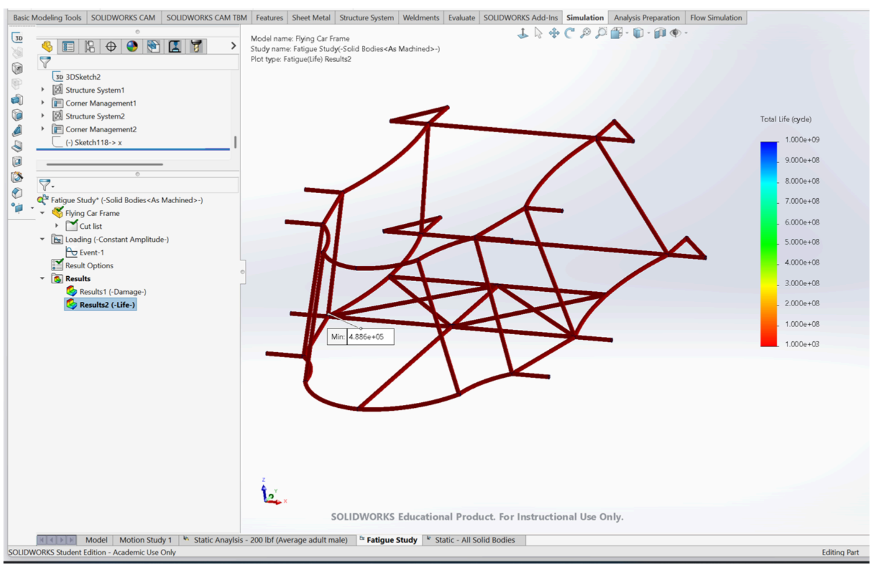
Life estimation under repeated loading

Purpose of the Fatigue Study

After performing the static structural analysis, a fatigue study was conducted on the frame to estimate the durability under repeated loading conditions. The objective was to identify potential fatigue-critical regions and verify that the structure achieved an acceptable operational life under cyclic service loads.

Fatigue Analysis Setup

- Stress results imported from the static structural analysis
- Material fatigue properties are defined using an S-N curve
- Loading is assumed to be repetitive and proportional to the static load case
- Linear cumulative damage model applied
- Analysis used to estimate relative life rather than exact failure cycles



Predicted fatigue life distribution under repeated loading conditions

Key Results

- Fatigue-critical regions aligned with high-stress areas identified in static analysis
- Predicted fatigue life exceeded operational usage
- No widespread fatigue damage observed across primary frame members

Engineering Takeaways

- Static stress analysis provided a conservative basis for fatigue evaluation
- Results indicated sufficient durability for intended operating conditions
- Fatigue analysis supported confidence in the structural robustness of the frame
- The extended predicted fatigue life reflects low cyclic stress amplitudes resulting from conservative loading assumptions and a stiffness-dominated frame design

CFD Flow Analysis – Propeller Concept (External & Internal Flow)

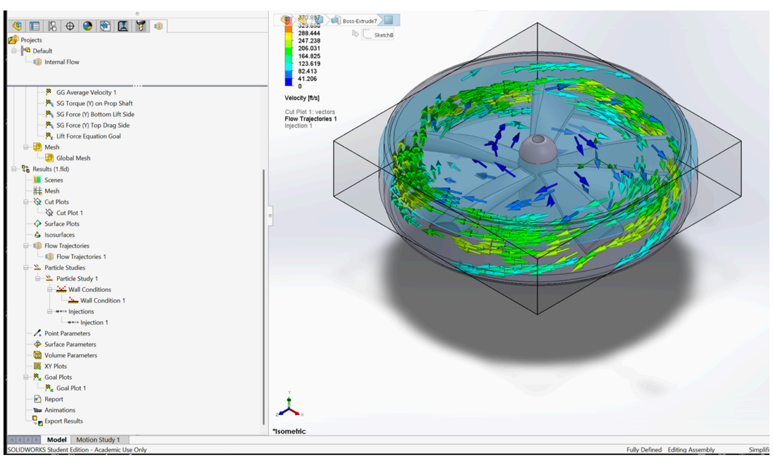
Qualitative airflow visualization and comparative performance indicators

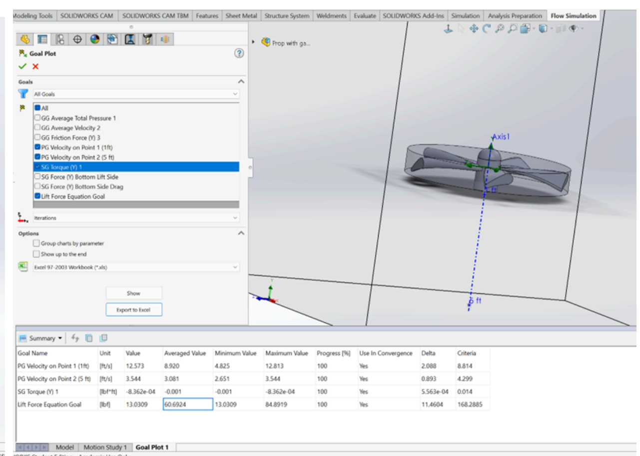
Purpose

External and internal flow simulations were conducted to visualize airflow behavior around the propeller concept and estimate relative effects on lift and shaft torque. The goal was to identify flow patterns and compare design configurations, rather than produce absolute performance predictions.

Setup

- 3D CFD used to evaluate external and internal flow behavior around the propeller/housing geometry
- Boundary conditions selected to represent a simplified operating environment
- Outputs reviewed: flow trajectories (qualitative) and lift/torque indicators (comparative)
- Results interpreted as design guidance, not test-validated performance





External flow trajectories illustrating induced airflow and wake structure around the propeller on the right. Computed lift and shaft torque used as comparative indicators between design configurations on the left

Key Takeaways

- Flow trajectories revealed non-uniform inflow and wake behavior, influencing efficiency
- CFD results were used to compare propeller behavior and guide design refinement
- Lift and torque outputs provided a quantitative reference for evaluating design changes