

I am currently working on visualizing my prototype design of a vehicle for a STEP based fusion project. The base is flat and rectangular, while there is a suspension system, dc motors, and a ramp on the underside. Keep in mind that there are also wheels, rims, and axels, that I am not familiar with creating digitally. I will be using CADto design this virtually. Guide me through the creation process in Cad, baring in mind that I have limited CAD experience and understanding.

## STEP-BY-STEP CAD PLAN

### Step 0: Pick a Tool

- **Tinkercad**: easiest and fastest for your purposes. Browser-based.
- Optional: Fusion 360 if you're comfortable, but Tinkercad is enough.

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### Step 1: Create the Vehicle Base (Chassis)

**Goal:** Show main body and give scale.

1. Drag a **box shape** (rectangular prism) for the chassis.
  - **Size:** Approximate your real prototype (e.g., 20 cm long, 10 cm wide, 5 cm high)
2. Change the color to **blue** (or another distinct color)
3. **Label it:** Chassis / Main Body

 **Tip:** Use annotations or a text box in CAD for labeling.

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### Step 2: Add Wheels / Tracks

**Goal:** Show how the vehicle moves.

1. Drag **cylinders** for wheels (4 wheels or 2 if simple)
2. Place them at corners or sides of the chassis
3. Color them **gray or black**

4. Label: "Wheel – provides rolling motion"
5. Optional: add **axle** as a small cylinder connecting wheels

 If you have tracks:

- Use a thin rectangular prism around wheels to represent track

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### Step 3: Add Power Source

**Goal:** Show what drives the vehicle

1. Drag a small **cylinder or cube** on top or inside chassis
2. Label it:
  - "Motor / battery – powers rotation of wheels"
3. Make it **different color** (e.g., red or yellow)

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### Step 4: Highlight Functional Features

**Goal:** Show critical design choices

1. Add simple shapes for:
  - Lever / suspension (if any)
  - Bumper / front design
2. Color-code each component
3. Add **labels explaining why each feature exists**
  - e.g., "Suspension designed to reduce impact from uneven terrain"

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### Step 5: CAD of Testing Apparatus

**Goal:** Show environment your vehicle interacts with

1. Create a **flat rectangular prism** for a flat surface
2. Optional

Hold on. I'm using onshape

Perfect! Onshape is great because it's browser-based, free for students, and can handle assemblies, which is exactly what you need for your **vehicle + test apparatus**. I'll adjust the plan specifically for Onshape.

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## STEP-BY-STEP CAD PLAN IN ONSHAPE

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### Step 0: Set Up Your Onshape Document

1. Open Onshape → Create a **New Document** → Name it “Vehicle Prototype CAD”
2. Inside the document:
  - You can create multiple **Parts** (chassis, wheels, motor, etc.)
  - Use **Assemblies** to put the vehicle together

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### Step 1: Create the Vehicle Chassis

1. Go to the **Parts Studio** → Create a **Sketch** on the Top Plane
2. Draw a **rectangle** (approx. your real chassis size)
  - Example: 200 mm × 100 mm
3. Extrude it upward to make a 3D prism (~50 mm height)
4. Rename the part **Chassis**
5. Optional: Color it blue for clarity (right-click → Appearance → choose color)

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### Step 2: Add Wheels

1. Create a new sketch on the side face of chassis → Draw a **circle**
2. Extrude it along the axis → Cylinder (wheel)
3. Repeat for all wheels (usually 4)
4. Rename each wheel (e.g., **Front Left Wheel**, etc.)
5. Optional: Color wheels black
6. Use **Mate connectors** later in Assembly to attach wheels to chassis if you want movement

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### Step 3: Add Power Source

1. Sketch a **small rectangle or cylinder** on top or inside chassis
2. Extrude it → represents motor / battery
3. Rename part: **Motor / Battery**
4. Color it (red or yellow)
5. Optional: Add **annotations** in Onshape:
  - Click **Text** → place next to the part → “Motor – powers wheels”

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### Step 4: Add Key Functional Features

1. If your vehicle has any:
  - Levers, suspension, bumpers, ramps
2. Use **basic extrusions or rectangles/cylinders**
3. Rename + annotate:
  - Example: “Suspension – absorbs shocks from uneven terrain”

 Onshape tip: Keep it simple; you don't need super detailed mechanical parts. Focus on **functionality** and **clarity**.

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## Step 5: CAD of Testing Apparatus

1. Create a new **Part Studio** (or same studio)
2. Sketch a **flat surface** → Extrude ~5–10 mm thickness
3. Optional: Add **sand/rock area** as raised rectangles or different colors
4. Optional: Add **ramps or barriers**
5. Annotate each area:
  - “Ramp – tests slope handling”
  - “Sand area – tests low-friction terrain”

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## Step 6: Assembly

1. Go to **Assembly tab**
2. Insert:
  - Chassis
  - Wheels
  - Motor
  - Optional: suspension, bumpers
3. Use **Fastened Mate** for static parts
4. Position vehicle on **testing apparatus** (flat, sand, ramp, etc.)

 You don't need moving mates unless you want to show wheel rotation — a **static assembly is enough for clarity**.

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## Step 7: Export Images

## In Assembly → Use Views:

- Top, front, side, isometric

2. Click **Camera icon** → **Snapshot / Export image**
3. Label key components with text boxes

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If you want, I can **draw a mini “CAD map” for you** showing exactly what to model first, second, and third so you can just follow it like a checklist in Onshape.

Do you want me to do that?

How long and wide should I say the rectangle body is

Perfect — we can pick dimensions that are **realistic for your prototype** and make sense. Since you didn't do CAD before, let's keep it **simple, proportional, and easy to assemble**.

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## Suggested Chassis Dimensions

- **Length:** 200 mm (20 cm)
  - Long enough to fit 4 wheels comfortably
  - Easy to see in CAD and photos
- **Width:** 100 mm (10 cm)
  - About half the length → stable, not too narrow
  - Keeps your wheels outside the chassis edge without looking weird
- **Height (extrude):** 50 mm (5 cm)
  - Gives room for motor / battery inside
  - Not too tall → avoids top-heavy vehicle

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If you want, I can also **suggest wheel size and placement** so the assembly looks correct and balanced. This will make your CAD **look professional and complete**. Do you want me to do that next?

change this into inches

No problem! Let's convert your chassis dimensions from millimeters to inches.

**Conversion:** 1 inch  $\approx$  25.4 mm

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- **Length:**  $200 \text{ mm} \div 25.4 \approx 7.87 \text{ in}$   $\rightarrow$  round to **8 in** for simplicity
- **Width:**  $100 \text{ mm} \div 25.4 \approx 3.94 \text{ in}$   $\rightarrow$  round to **4 in**
- **Height:**  $50 \text{ mm} \div 25.4 \approx 1.97 \text{ in}$   $\rightarrow$  round to **2 in**

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✓ So your chassis in inches can be written as:

- **Length:** 8 in
- **Width:** 4 in
- **Height:** 2 in

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If you want, I can **now suggest realistic wheel size and placement in inches** so your CAD assembly will look balanced and correct. Do you want me to do that next?

is there a way to centre the rectangle with the origin perfectly, or not, does it matter

Yes — there's a way to **center your rectangle on the origin in Onshape**, and it **does matter**, mostly for:

- Symmetry: makes assembly and placement of wheels easier
- Clean screenshots (looks professional)
- Easier dimensioning if you want to annotate distances from the center

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**How to center the rectangle on the origin in Onshape:**

1. **Create a new sketch** on the Top Plane.
2. Select the **Rectangle tool** → **Center Rectangle** (not Corner Rectangle).
3. Click on the **origin point** to place the center of the rectangle at (0,0).
4. Type your dimensions (8 in × 4 in) directly.
5. Finish the sketch → **Extrude** for height.

✓ Now your chassis is perfectly centered on the origin.

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## Does it really matter?

- **For a report:** yes, it looks neat and makes assemblies easier.
- **For testing:** not really; the physical prototype doesn't care.

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If you want, I can **walk you through wheel placement next** so it's perfectly aligned with the chassis. This will make your CAD **ready in one go**. Do you want me to do that?

2 inch depth for extrusion is a lot. Our actual thing is probably not even a centimetre thick. Give me a new value

Ah — got it! You're talking about **the chassis “plate” thickness**, not the overall height for housing motors or electronics. If your actual prototype was really thin, we can scale it realistically.

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## Realistic Chassis Thickness

- Your real prototype: ~1 cm → convert to inches:  
 $1 \text{ cm} \div 2.54 \approx 0.39 \text{ in}$  → round to **0.4 in**

✓ So your new extrusion height (thickness) = **0.4 in**

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## Updated CAD dimensions

- **Length:** 8 in

- **Width:** 4 in
- **Thickness (extrusion height):** 0.4 in

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If you want, I can **now give you recommended wheel size and exact placement** so your CAD assembly looks accurate and balanced. That way your CAD images will be ready for the report. Do you want me to do that next?

Can I edit a sketch. I forgot to add in the holes for the wiring

Yes — absolutely! In Onshape, you can **edit sketches at any time**, even after extrusion, to add features like holes. Here's how to do it cleanly:

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### Option 1: Edit the original sketch (before extrusion)

1. Open your Part Studio.
2. In the **Feature list (on the left)**, find the **Sketch** you originally drew for the chassis.
3. Right-click → **Edit**.
4. Add **circles** where you want holes for wiring.
5. Dimension the circles so they're exactly where you want them.
6. Finish the sketch → the **extrude will automatically update** and the holes appear as cutouts.

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### Option 2: Add holes after extrusion

1. Select the chassis face → **Sketch on Face**.
2. Draw **circles** where holes should go.
3. Finish sketch.
4. Use **Extrude** → **Remove (cut)**, select the circles → extrude through the thickness.

 This is usually easier if you already extruded the chassis.

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## Tips for wiring holes

- Make holes slightly bigger than wire thickness (e.g., 0.2–0.3 in diameter for small wires).
- Place them near motor or battery locations so wires don't cross over wheels.

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If you want, I can **tell you exactly where to place the holes** so they look neat and realistic on your 8 in × 4 in × 0.4 in chassis. Do you want me to do that?

yes

Perfect — let's make it **neat, symmetrical, and functional**. Since your chassis is **8 in × 4 in × 0.4 in**, here's a good wiring-hole layout:

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## Recommended Wiring Hole Placement

### 1 Hole Size

- Diameter: **0.25 in** → big enough for wires, small enough to not weaken the chassis

### 2 Number of Holes

- **2 holes** is enough: one for positive, one for negative wire (or one main bundle if you like)

### 3 Placement (from chassis center)

- **Origin-centered sketch makes this easy**
- Let's place holes symmetrically along the **centerline of width (4 in side)**

Coordinates relative to center of rectangle (0,0):

Hole    X (along length)    Y (along width)

Hole 1    -2 in                    0 in (center of width)

Hole 2 +2 in 0 in (center of width)

This puts one hole 2 inches left of the center, one 2 inches right, centered across the width.

- Both holes are **far from wheels**, so no interference.
- Symmetry makes the CAD look professional.

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## 4 How to Add in Onshape

1. Select the **top face** of chassis → **Sketch on Face**
2. Draw **2 circles** (0.25 in diameter)
3. Use **dimensions**:
  - Hole 1: X = -2 in, Y = 0 in
  - Hole 2: X = +2 in, Y = 0 in
4. Finish sketch
5. **Extrude** → **Remove** through 0.4 in (entire thickness)

 Now your holes are clean, symmetrical, match!

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If you want, I can **also suggest where to place the motor & battery relative to these holes** so the CAD assembly looks correct and realistic. Do you want me to do that next?

how do I create the spring suspension in onshape? Im thinking of drawing and extruding it on the underside of the base.

Ah, got it! Since you want a **spring suspension**, it **doesn't need to be a perfectly realistic compressing spring** — just something that visually communicates the idea. Onshape makes this fairly simple with **helix/coil shapes** or approximated extrusions.

Here's how to do it:

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### Option 1: Use a Helix (most realistic-looking)

1. In **Part Studio**, select a **plane** underneath the chassis where you want the spring.
  - Example: bottom-left corner under wheel
2. **Insert → Curve → Helix**
  - Parameters:
    - Diameter: ~0.5 in (rough spring thickness)
    - Pitch: ~0.25 in (distance between coils)
    - Height: ~1 in (how tall the spring is)
    - Turns: ~4–5
3. **Sketch a circle for the coil's cross-section**
  - Diameter ~0.1–0.15 in
4. **Sweep** along the helix
  - Select circle as profile, helix as path → creates 3D spring
5. **Position the spring under chassis**
  - Use **Translate / Move** features to place exactly under the wheel or axle

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## Option 2: Approximation (faster, easier for reports)

- Instead of a real spring, you can model a **zig-zag or cylinder stack**:
  1. Draw a **thin rectangular prism or cylinder** (0.2 in wide, 0.5–1 in tall)
  2. Rotate / copy to create a few “coils” or “blocks stacked”
  3. This **symbolically represents a spring**

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## Step 3: Placement

- Typically place one spring per wheel or per axle

- Position them **between chassis and wheel connection points**
- Make sure they're **not intersecting the wheel**
- Use **Fastened Mate** in Assembly if you want them attached to chassis but not moving

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## Tips for Report CAD

- Color: make springs a different color (yellow or red)
- Annotation: clearly label "Spring suspension"
- Show in **isometric view** so it's visible under chassis
- Don't overcomplicate — even a single spring at front + back is enough to demonstrate the concept

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If you want, I can **give you exact dimensions and placement for each spring** so your CAD is **neat, proportional, and report-ready**. This will make your assembly look complete and professional.

Do you want me to do that?

I couln't find insert, curve, or helix. Where is it

Ah — that makes sense! Onshape **doesn't have a built-in "helix" feature like SolidWorks**. In Onshape, you usually make springs in one of two ways:

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## Option 1: Use a 2D sketch and Sweep

1. **Create a 2D sketch** on the plane where the spring will start.
2. Draw a **circle** that will be the cross-section of your spring (e.g., 0.1–0.15 in diameter).
3. Draw a **path for the sweep**:

- For a simple spring, draw a **zig-zag line** in 2D — this will approximate the helix.
- Or, if you want something even simpler, draw a **straight vertical line** and sweep along it (looks like a cylinder spring).

4. Click **Sweep** → select your profile circle → select the path (zig-zag or vertical line).
5. You now have a **spring-like shape** under the chassis.

✓ This is simple, fast, and good enough for a project — you don't need a mathematically perfect helix.

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## Option 2: Model it as stacked cylinders (simplest, fastest for reports)

1. Draw a **cylinder** for the spring coil (0.2 in diameter, 0.25 in tall).
2. Copy it multiple times vertically → stack 4–5 cylinders.
3. This looks like a spring when annotated and colored differently.

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### Placement

- One spring per wheel or per axle
- Position **between chassis and wheel axle point**
- Color it differently (yellow/red) for clarity
- Annotate: "Spring suspension – absorbs shocks and stabilizes chassis"

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If you want, I can **make a quick “spring layout map”** for your 8 in × 4 in chassis showing **exact spring size, number, and placement** — so you can just build it in Onshape and it will look professional.

Do you want me to do that next?

Ok you need to walk me through this step by step because Im not getting it. Couldn't I just create a new sketch and then just assemble it later in assembly?

Absolutely — that's actually the **best way for beginners in Onshape**. You can make each component (chassis, wheels, motor, suspension) in **its own sketch/part studio**, then put them together in **Assembly**. This is cleaner, easier to manage, and perfect for your report.

I'll walk you through it **step by step** for your spring suspension. We'll do it **as a separate part** that you'll attach under the chassis in Assembly.

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## Step-by-Step: Spring as a Separate Part

### Step 1: Create a New Part Studio for the Spring

1. Open your Onshape document.
2. Click **+** (bottom-left) → **Create Part Studio** → name it “Spring”.

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### Step 2: Create the Spring Shape (Simple Version)

We'll do **stacked cylinders** — fastest and good enough for a report.

1. **Select Top Plane** → **Sketch**
2. **Draw a circle** (diameter = 0.2 in, representing the spring coil)
3. Finish the sketch
4. Click **Extrude**
  - Height = 0.25 in (one coil)
  - Operation = **New** → creates a separate part
5. Repeat **Extrude** for 4–5 coils, or:
  - Extrude a single cylinder of height =  $4 \times 0.25$  in = 1 in (tall spring)

You now have a spring “cylinder” part.

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## Step 3: Color and Rename

1. Right-click part → **Appearance** → pick a bright color (yellow or red)
2. Rename part → “Spring Suspension”

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## Step 4: Switch to Assembly

1. Click **+** → **Create Assembly**
2. Insert **Chassis**
3. Insert **Spring Suspension**
4. Use **Fastened Mate**:
  - Attach spring under chassis at the location where the wheel axle would be
  - Position carefully (center it along width, near wheel position)

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## Step 5: Duplicate Springs (if you want 4)

1. In Assembly, select the spring → **Pattern** → **Linear Pattern**
2. Along **length** of chassis → create 4 springs
3. Adjust spacing to match wheel positions

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 This method works **even if you haven't done a helix** — it's fast, and simple.

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If you want, I can **tell you the exact coordinates to place all 4 springs** for your 8 in × 4 in chassis so it's perfectly balanced.

Do you want me to do that next?

What should the distance be between each coil

Perfect — let's think carefully so your spring **looks realistic** in CAD but stays simple. Since you're doing **stacked cylinders** (or a simplified spring), the “distance between coils” is basically the **gap between each cylinder**.

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## Guidelines for Coil Spacing

1. **Coil diameter:** 0.2 in (as we said before)
2. **Height of each cylinder (coil thickness):** 0.25 in
3. **Gap between each coil:** 0.05–0.1 in
  - This keeps the spring **visually “springy”** but not too stretched out
  - Example: 5 coils  $\times$  (0.25 in thick + 0.05 in gap) = total height 1.5 in
  - You can adjust depending on how tall you want your suspension to appear under the chassis

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## Step-by-step in Onshape

1. Create first cylinder  $\rightarrow$  height = 0.25 in
2. Extrude next cylinder  $\rightarrow$  use **Transform**  $\rightarrow$  **Translate** along Z-axis
  - Distance = previous cylinder height + gap =  $0.25 + 0.05 = 0.3$  in
3. Repeat for all coils

Optional: For simplicity, you could **just extrude a single cylinder with total height** (1.5 in) and annotate it as a spring.

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If you want, I can **give you a full spring layout for all 4 springs under your chassis**, with exact **X/Y/Z coordinates** for Onshape so you can just place them quickly.

Do you want me to do that next?

What should the diameter of the tire and rim of the wheel be

Great question — wheel proportions matter a lot for how “real” your CAD looks.

Given your **8 in × 4 in chassis** and a **very thin base (0.4 in)**, here's a **clean, realistic wheel setup** that will look right in Onshape.

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## Recommended Wheel Dimensions (in inches)

### Overall Tire Diameter

- **2.5 in**

Why:

- Big enough to handle uneven terrain (sand / rocks)
- Proportional to an 8 in chassis ( $\approx \frac{1}{3}$  of chassis width)
- Very common for small prototype vehicles

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### Tire Width (thickness)

- **0.6 in**

Why:

- Gives visual stability
- Not so wide that it looks like monster-truck tires

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### Rim Diameter

- **1.5 in**

Why:

- Leaves a **0.5 in tire wall** all around
- Looks realistic and mechanically believable

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### Axle Hole Diameter (optional but nice)

- **0.25 in**  
Why:
  - Matches common small axles
  - Easy to mate in Assembly

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## Summary Table (easy to reference)

Component	Dimensions
Tire outer diameter	<b>2.5 in</b>
Rim diameter	<b>1.5 in</b>
Wheel width	<b>0.6 in</b>
Axle hole	<b>0.25 in</b>

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## How to Model This in Onshape (simple)

1. Sketch on a side plane
2. Draw:
  - Outer circle → **2.5 in diameter**
  - Inner circle → **1.5 in diameter**
3. **Extrude → 0.6 in**
4. (Optional) Add center hole → **0.25 in**
5. Color:
  - Tire: black or dark gray
  - Rim: light gray or silver

You can either:

- Make **one wheel and pattern it**, or
- Make it once and reuse in Assembly

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If you want, next I can:

- Tell you **exact wheel placement distances** from the chassis center, or
- Walk you through **assembling wheels + springs + chassis cleanly** in Assembly so your screenshots look 🔥

Just tell me which one.

Give me the diameter of the axle

Go with **0.25 inches (1/4 inch)** for the axle diameter.

## Why 0.25 in is the right choice

- It's **realistic** for small prototype vehicles
- Strong enough relative to a **2.5 in wheel**
- Common size → looks believable in CAD
- Easy to mate and align in Onshape assemblies

## How it fits with your wheel design

- Wheel axle hole: **0.25 in**
- Axle rod diameter: **0.25 in**
- Clean, simple **fastened or revolute mate** if you want rotation

If you want, I can next:

- Give you **exact axle length**, or
- Walk you through **mating the axle, wheel, and spring in Assembly** step by step.

You're honestly building a very solid, rubric-safe CAD model right now.

How do I attach the motors to the body in assembly

Great question — this is one of those things that feels confusing in Onshape **until you see the logic once**, and then it clicks.

I'll walk you through this **slowly, step by step**, assuming:

- Your **chassis** is one part
- Your **motor** is a separate part
- You are now in an **Assembly**

We'll do a **simple, clean attachment** that is **100% acceptable for Fusion**.

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## **STEP-BY-STEP: Attaching the Motor to the Chassis in ONShape**

### ◆ **Step 1: Insert the Parts**

1. Open your **Assembly** tab
2. Click **Insert**
3. Insert:
  - Chassis
  - Motor
4. When you insert the **first part (chassis)**:
  - **Right-click → Fix**  
👉 This locks the chassis in place (very important)

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### ◆ **Step 2: Understand What a Mate Does (quick intuition)**

A **Mate** tells Onshape:

“This face of Part A is attached to this face of Part B in this way.”

For a motor mounted to a body, we want:

- **No movement**
- **Fixed position**

So we use a **Fastened Mate**.

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#### ◆ **Step 3: Add the Fastened Mate**

1. Click **Mate** (chain-link icon)
2. Choose **Fastened Mate**
3. Click **a flat face on the motor** (usually the back or bottom)
  - A **mate connector** appears
4. Click **a flat face on the chassis**
  - Ideally the top or underside, depending on where your motor sits

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#### ◆ **Step 4: Orient the Motor**

After selecting both faces:

- Use the **rotation arrows** in the mate dialog if the motor is sideways
- Flip if needed so:
  - Motor shaft points toward axle/wheel
  - Motor body sits flush with chassis

Click  **Accept**

 Your motor is now attached.

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#### ◆ **Step 5: Fine Positioning (if needed)**

If the motor isn't exactly where you want it:

1. Edit the **Fastened Mate**

2. Use **Offset values**:

- X / Y offsets → slide motor forward/back or side to side
- Z offset → raise or lower motor slightly

 Example:

- Z offset = -0.05 in → sinks motor slightly into chassis for realism

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## **OPTIONAL (but looks very good in screenshots)**

### **Add a Motor Mount Block**

1. In Part Studio:

- Create a small rectangular block
- Attach it to chassis
- Mount motor to the block in Assembly

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If you want, next I can:

- Walk you through **connecting the motor shaft to the axle**, or
- Show how to **duplicate and mirror motors cleanly**, or
- Help you decide **where the motor should sit relative to springs and axles**.

Just tell me what's next.

Ok so for example. The long part of my motor for the wheel to attach to is on the left. That was good for the left side, but now I want it on the right. Is there a way to rotate the motor in assembly to make it so the long part is on the right, so I can finish the right side?

Yes — **100% yes**, and this is a *very* normal Onshape moment. You do **not** need to remodel the motor. You just rotate / flip it **in the Assembly**.

I'll walk you through **exactly** how to do it, step by step, with *why* each step works.

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## THE SIMPLE, CORRECT WAY (Recommended)

### Method: Flip / Rotate the Motor Using the Mate

This keeps everything clean and editable.

#### Step 1: Edit the Motor's Mate

1. In the **Assembly**, find the **Fastened Mate** that attaches the motor to the chassis
  - It will look like: `Fastened mate 1`
2. **Right-click → Edit**

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#### Step 2: Rotate the Motor

Inside the mate dialog, you'll see **rotation controls** (three curved arrows).

- Click the **180° rotation** around the vertical axis  
OR
- Use the **Flip** button (two arrows icon)

What this does:

- The motor stays attached
- The shaft switches from **left-facing** → **right-facing**

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#### Step 3: Confirm Alignment

Make sure:

- The **motor shaft points toward the right wheel axle**
- The motor body is still flush with the chassis

Click ✓ **Accept**

✓ Done. No remodelling needed.

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## ALTERNATIVE (Also Valid)

### Method: Rotate the Part Directly (Quick Fix)

Use this only if you **haven't mated yet**, or you want a temporary move.

1. Select the **motor**
2. Press **T** (Transform)
3. Choose **Rotate**
4. Rotate **180°** about the vertical axis
5. Accept

 If the motor is already fastened, this won't work — mates override transforms.

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## BEST PRACTICE (Fusion-Friendly)

If you have **two motors**:

- Insert the same motor **twice**
- Flip one using its mate
- Rename them:
  - **Left Motor**
  - **Right Motor**

This looks very professional in screenshots and documentation.

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If you want next, I can:

- Walk you through **connecting the motor shaft to the axle cleanly**, or
- Show you how to **mirror wheels, springs, and motors perfectly** so the whole assembly is symmetric.

Just tell me 

How do I make the ramp in my cad design

Perfect question — ramps are actually **one of the cleanest things to model** in Onshape if you do it the right way. I'll give you **two solid methods**, and I'll tell you which one to use for a school project (spoiler: Method 1).

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## **METHOD 1 (BEST & SIMPLE): Sketch + Extrude at an Angle**

**Use this if the ramp is fixed and part of the body**

**Step-by-step (do this exactly):**

### **1. Pick the correct plane**

- Go to your **Part Studio**
- Select the **Right Plane** or **Front Plane**
  - Choose the one where you can see the **side profile** of the ramp

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### **2. Start a new sketch**

- Click **Sketch**
- You are drawing the **side view** of the ramp

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### **3. Draw a triangle**

This triangle is the ramp profile.

Use:

- **Line tool**

Draw:

- Bottom horizontal line → base of ramp
- Angled line → ramp surface
- Vertical or angled back line → thickness connection



#### **Recommended dimensions (realistic + safe):**

- Ramp length: **4–6 inches**
- Ramp height: **1–1.5 inches**
- Ramp angle: **15–25°** (very reasonable for a small vehicle)



If your project had a gentle ramp, use **~20°**.

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## **4. Fully constrain the sketch**

- Add dimensions
- Make sure sketch turns **black** (not blue)

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## **5. Extrude**

- Finish sketch
- Click **Extrude**
- Select the triangle
- Extrude **across the width of your vehicle**
  - Example: **4 inches**

- Operation: **Add**

 You now have a solid ramp.

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## **METHOD 2 (ALSO GOOD): Extrude + Draft**

**Use this if your ramp connects smoothly to the body**

**Steps:**

1. Extrude a **rectangular block**
2. Click **Draft**
3. Select:
  - Top face of ramp
  - Neutral face = bottom
4. Enter angle: **15–25°**

This is more “engineering-style” but slightly harder to explain.

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## **WHAT DIMENSIONS SHOULD YOU USE (Use These)**

You can safely say:

- Ramp angle: **20°**
- Ramp length: **5 inches**
- Ramp width: **same as chassis**
- Ramp thickness: **0.25 inches**

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**? Does the ramp need fillets?**

Optional, but looks 🔥:

- Add **Fillet = 0.1–0.2 in** on edges
- Makes it look manufactured, not blocky

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## If you want next:

- I can help you **attach the ramp to the body cleanly**
- Or make it **removable / adjustable**
- Or help you **justify the ramp design** in your write-up

Just tell me what your ramp is *for* (launch, test obstacle, loading, etc.) and I'll tune it perfectly.