WebCAE 101

LESSON 1 OF 19 · F01

A cantilever beam in twelve steps

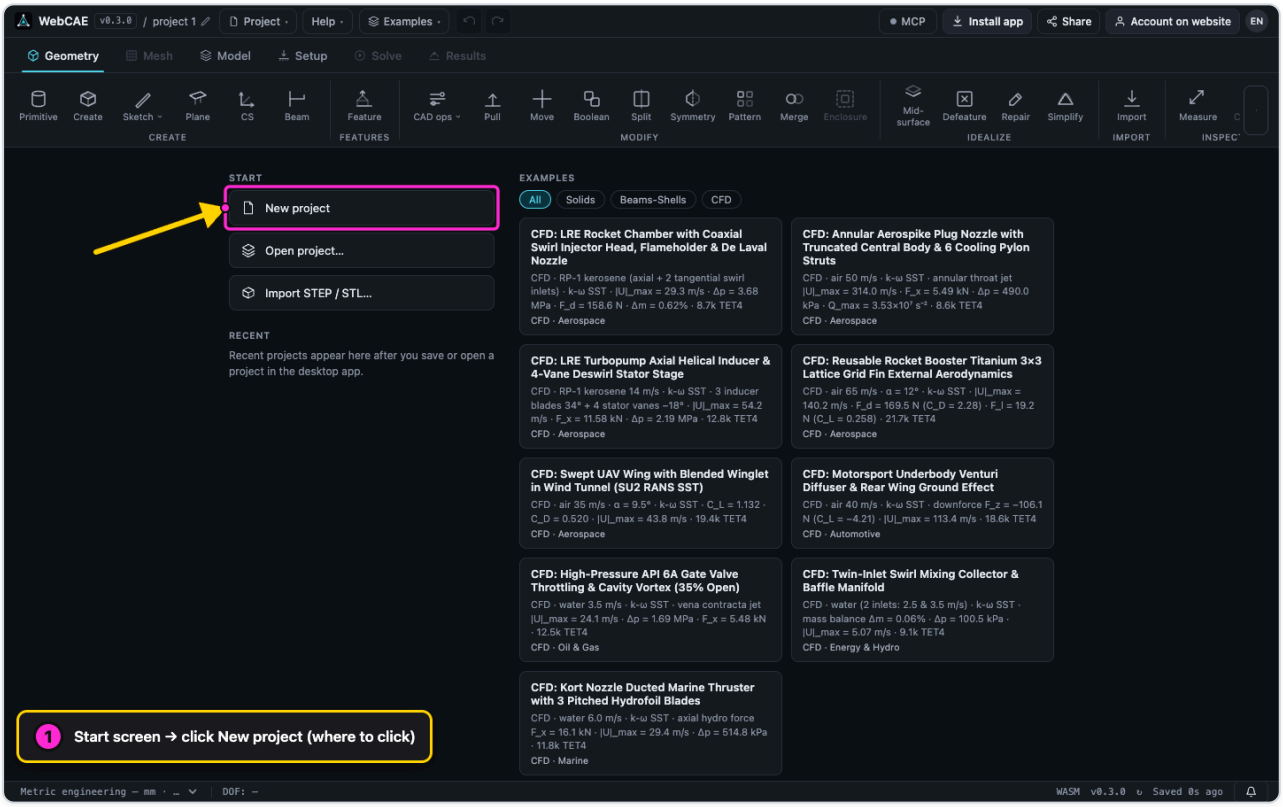
Goal Compute the tip deflection of a steel cantilever and check it against the textbook formula.

Outcome The tip deflection of a steel cantilever: 1.600 mm by hand, 1.595 mm in WebCAE (web) and 1.596 mm in WebCAE desktop.

6–7 min | web (WASM) | Run: 2026-11-04-32ffaed2 / F01

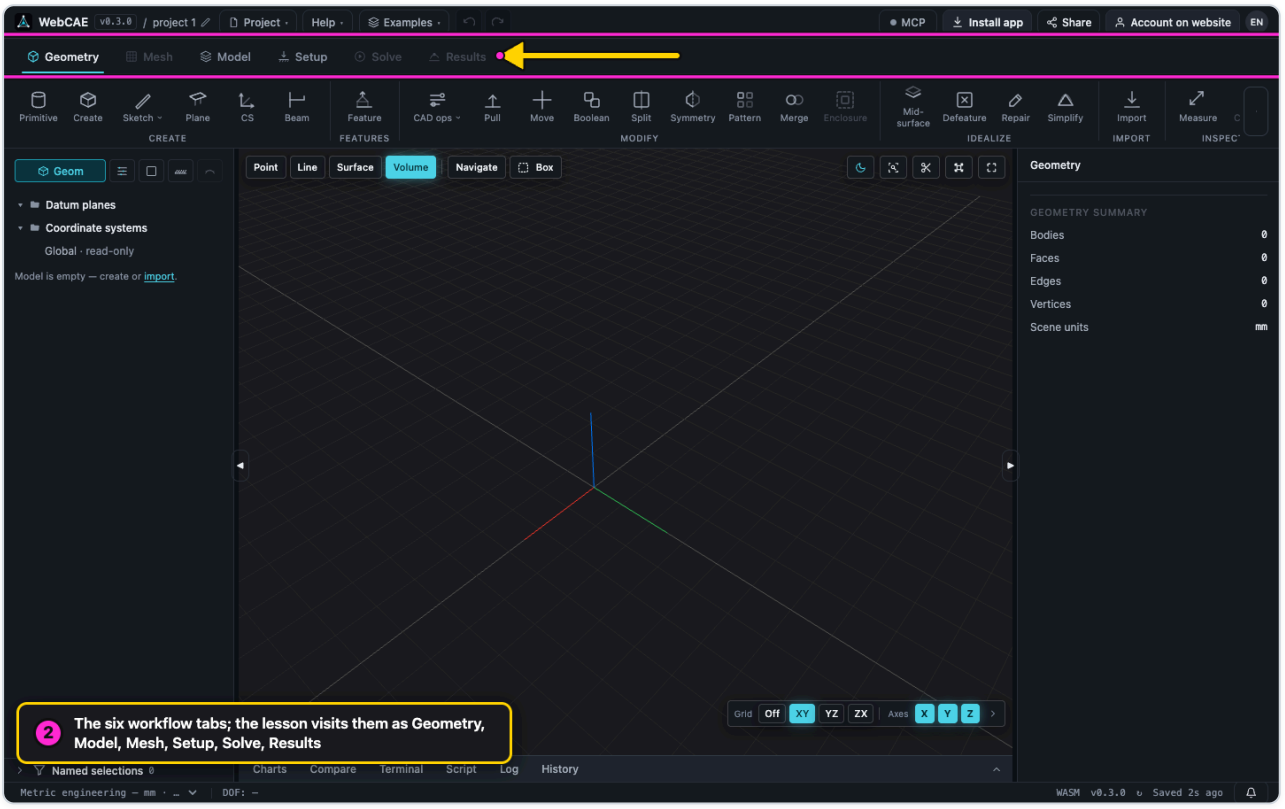
STEP 1 Start a new project

When WebCAE opens, it shows the **Start** screen: **New project**, **Open project...**, **Import STEP / STL...** and a gallery of examples. While it is on screen the workspace behind it is not usable (a ribbon button highlights, but its panel does not open). Click **New project**: the Start screen closes and an empty project opens.



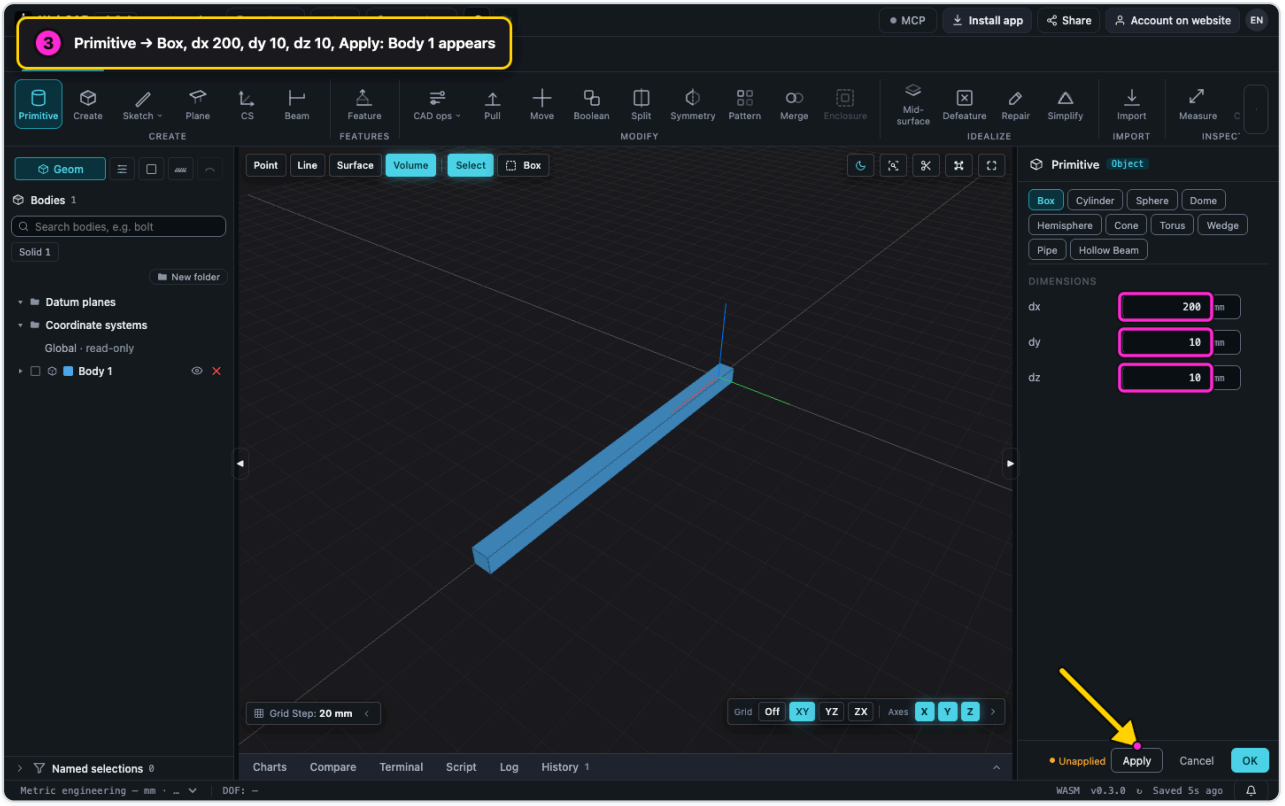
STEP 2 See the six workflow tabs

Look at the row of tabs under the title bar: **Geometry**, **Mesh**, **Model**, **Setup**, **Solve**, **Results**. Together they are the whole simulation workflow, but you will not visit them strictly left to right: the bar lists **Mesh** second, yet in this lesson you set the material in **Model** first and come back to **Mesh** afterwards. The order of the lesson is **Geometry → Model → Mesh → Setup → Solve → Results**. Start on **Geometry**; **Mesh**, **Solve** and **Results** stay greyed out until there is something to mesh, solve or show.



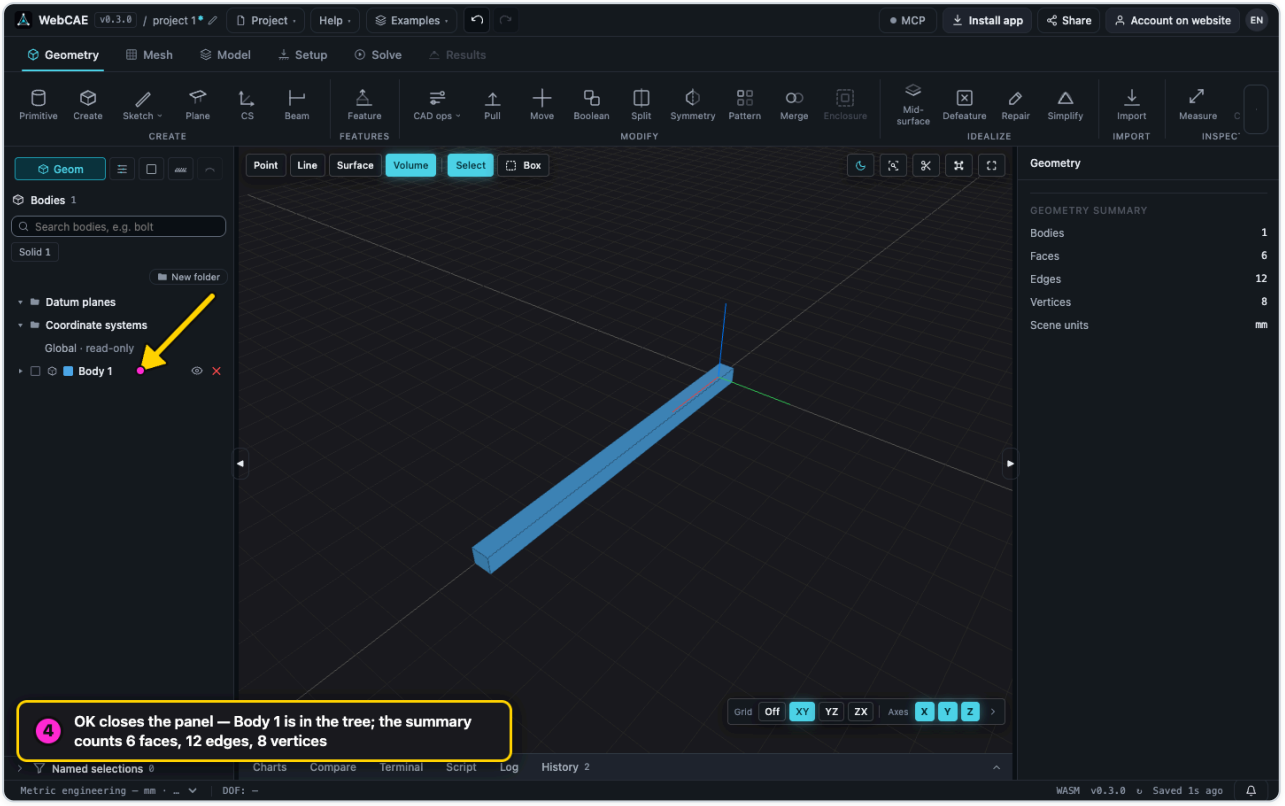
STEP 3 Create the 200 × 10 × 10 mm bar

In **Geometry**, click **Primitive** in the ribbon, make sure **Box** is selected, type `200` in **dx**, `10` in **dy**, `10` in **dz**, then click **Apply**. A thin bar appears in the viewport.



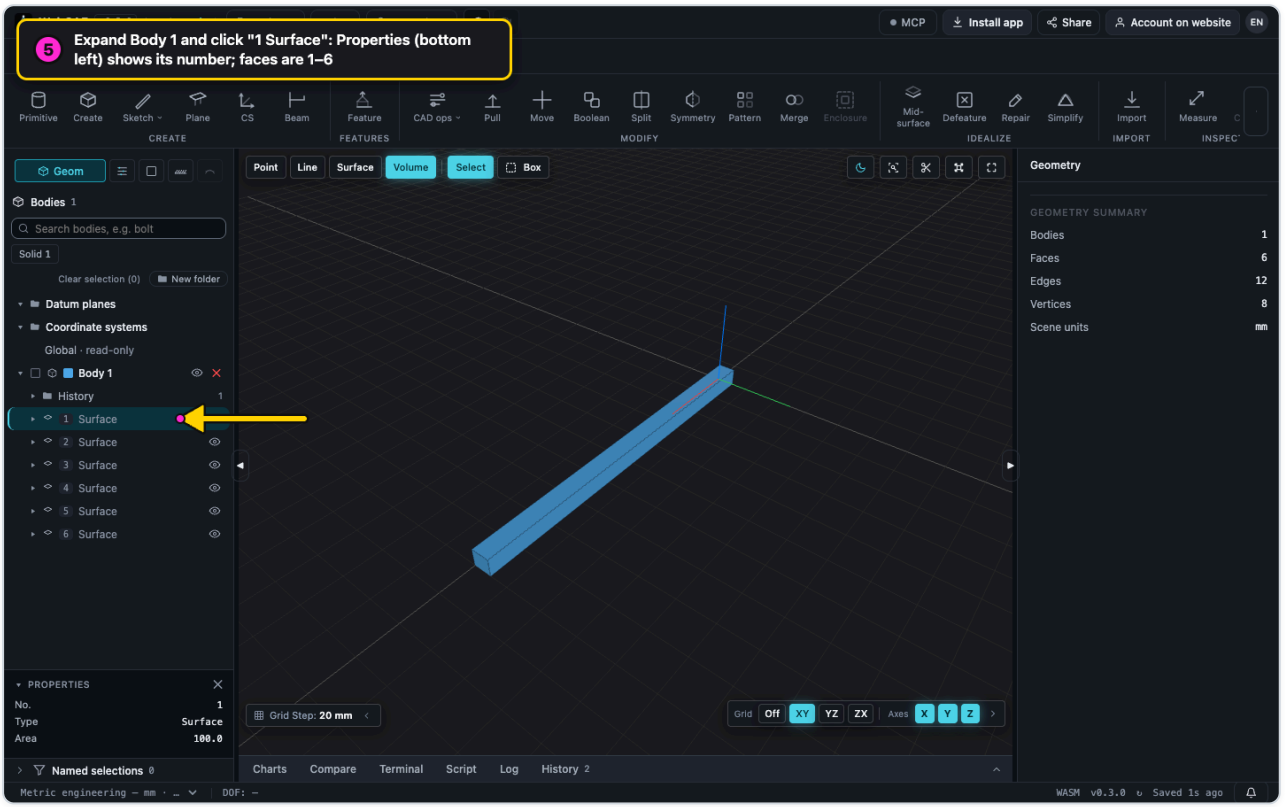
STEP 4 Close the panel

Click **OK**. The panel closes; **Body 1** is in the Geometry tree, and the **Geometry summary** on the right counts 1 body, 6 faces, 12 edges, 8 vertices — a box.



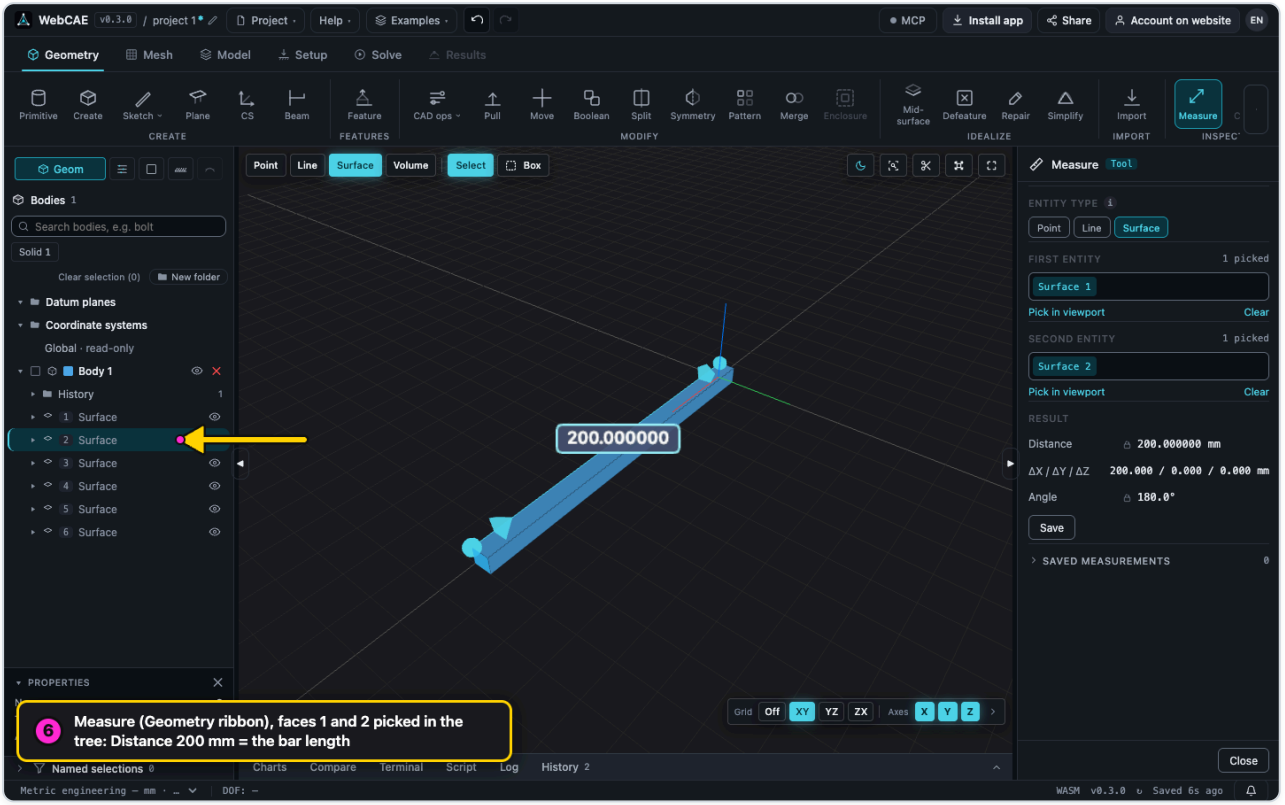
STEP 5 Find a face number

Expand **Body 1** (the arrow at its left) and click **1 Surface** in the tree. The **Properties** block at the bottom left shows **No. 1, Type Surface**. These tree numbers (faces 1–6) are what you will type later to say *where* a support or a load goes. Face 1 is one of the two small end faces — the one at the origin, where the bar will be clamped.



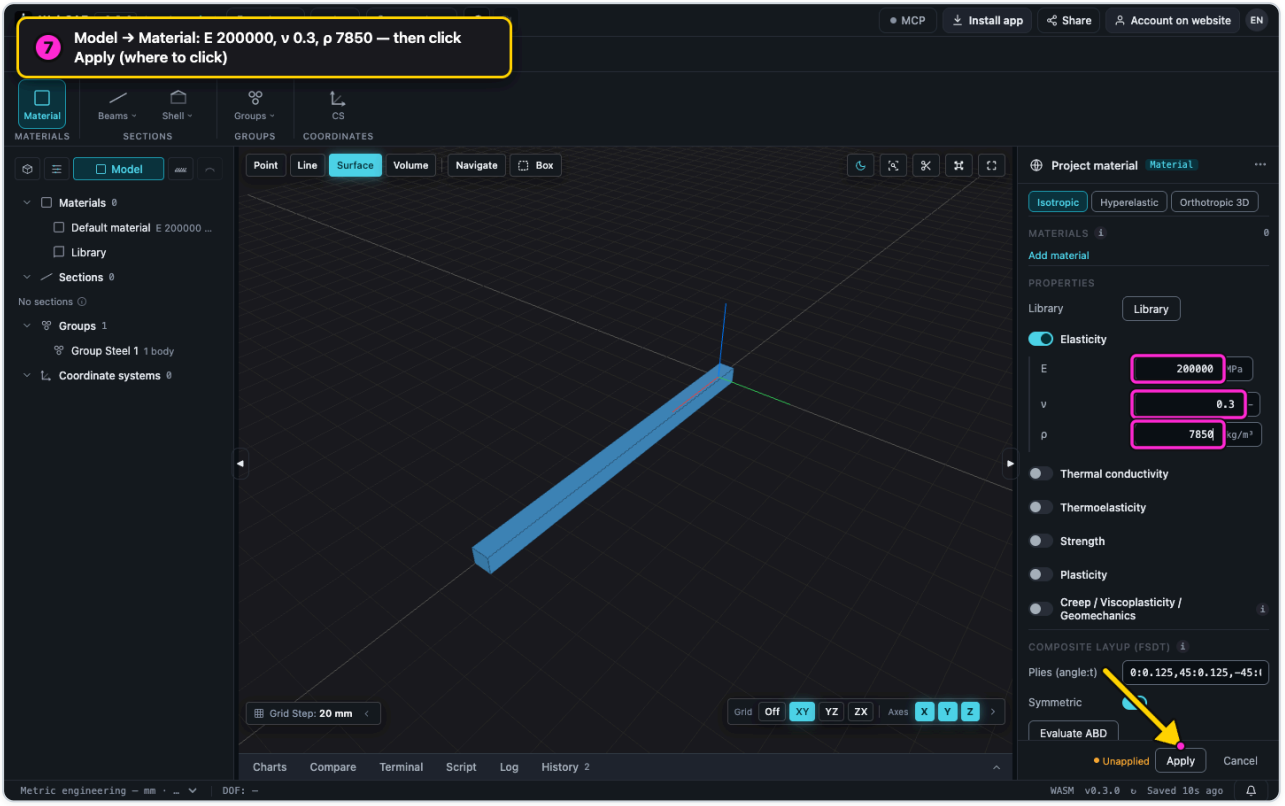
STEP 6 Check the length with Measure

Click **Measure** in the **Geometry** ribbon (Inspect group; the entity type is **Surface**), then click **1 Surface** and **2 Surface** in the tree. The panel's **Distance** reads **200.000000 mm** (ΔX / ΔY / ΔZ = 200 / 0 / 0): the two end faces are 200 mm apart, so face 2 is the free end. Click **Close** to leave Measure.



STEP 7 Assign structural steel properties

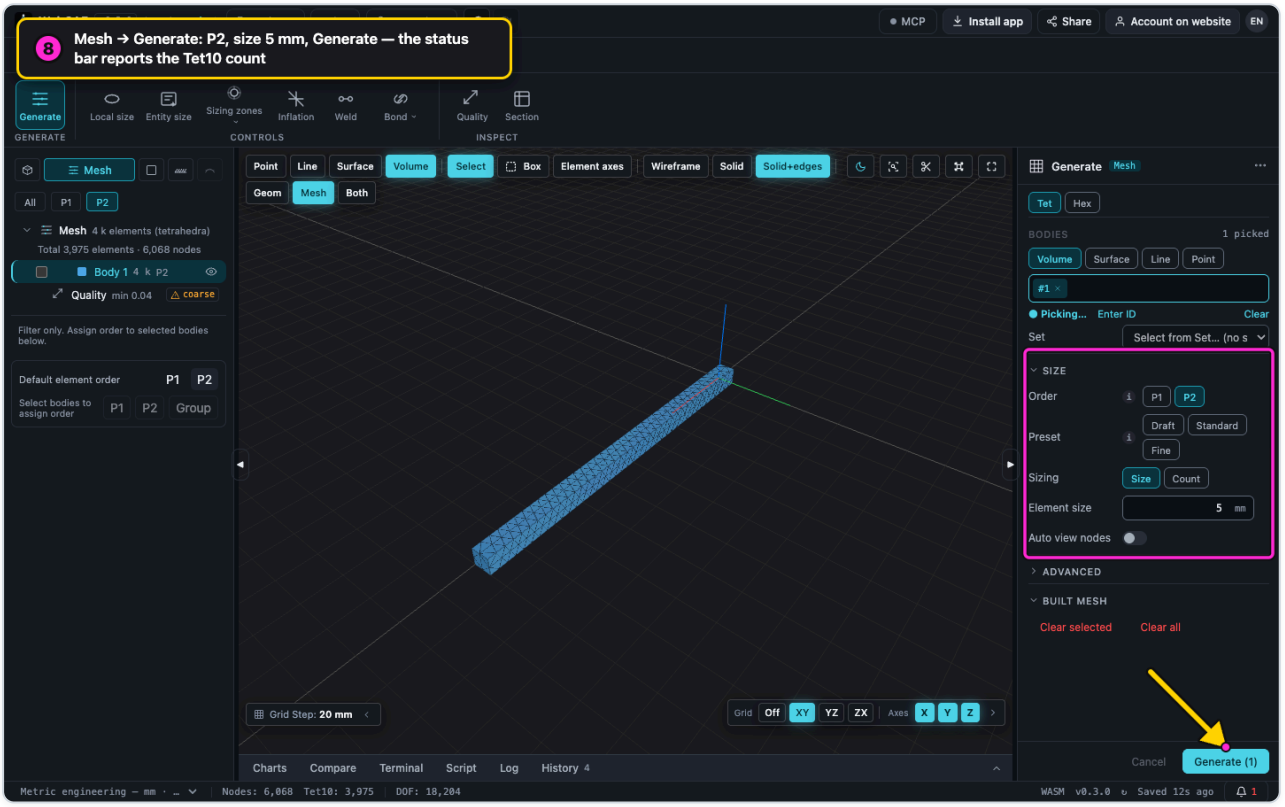
Open the **Model** tab and click **Material**. Enter Young's modulus **E** = `200000` MPa, Poisson's ratio **ν** = `0.3` and density **ρ** = `7850` kg/m³, then click **Apply**.



STEP 8 Mesh with quadratic tetrahedra

Open the **Mesh** tab, click the **Generate** tile in the ribbon, pick order **P2** (quadratic), set **Element size** to `5` mm and click **Generate (1)** at the bottom of the panel (the number in brackets is how many bodies are picked). When it finishes, the status bar at the bottom reports the node and **Tet10** counts (about 3,975 Tet10 in this run). The mesh tree may flag the mesh as *coarse* (quality min ≈ 0.04): that is expected at 5 mm for a 10 mm section and is accurate enough for this bending check.

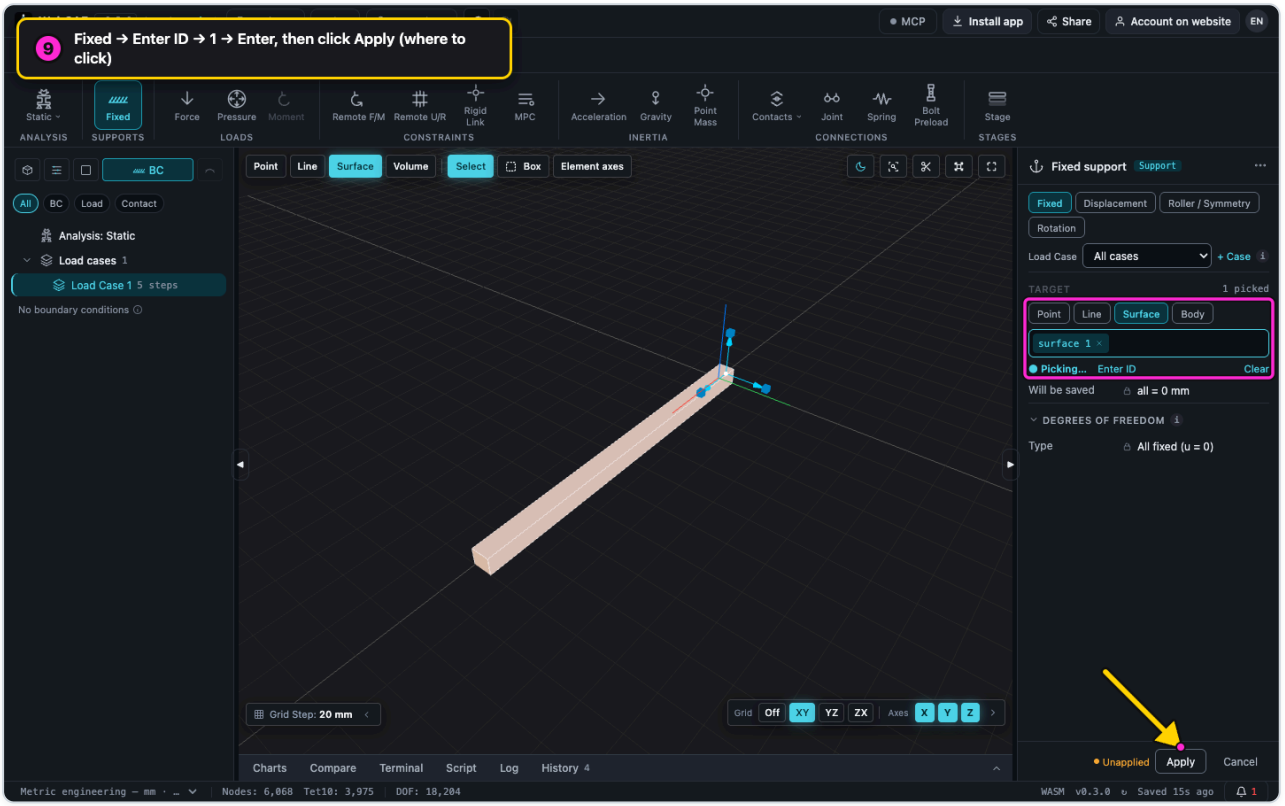
WHY Quadratic 10-node tetrahedra (Tet10) do not lock in bending, so a coarse mesh is enough.



STEP 9 Clamp the root face

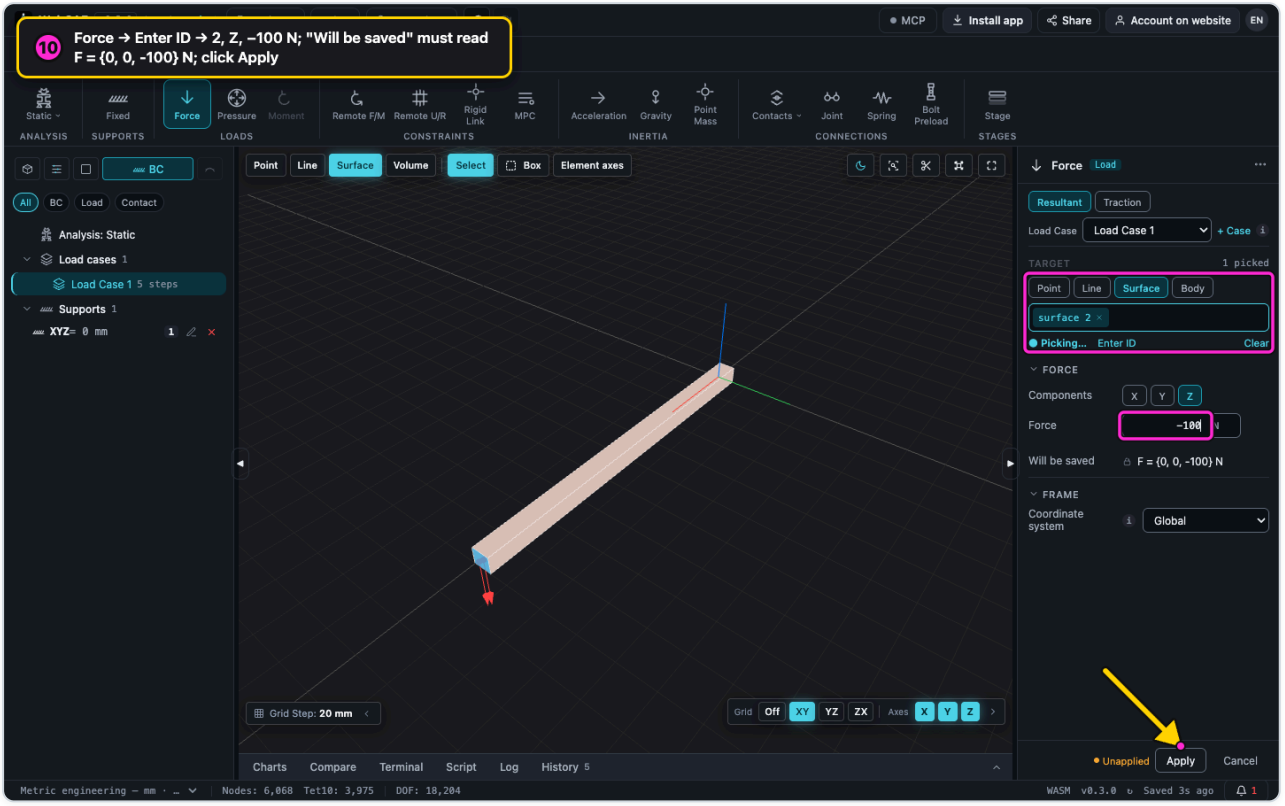
Open the **Setup** tab and click **Fixed**. Click **Enter ID**, type `1` (the root face from step 5), press **Enter**, then click **Apply**. The target field now shows *surface 1*.

WHY A fixed support removes all three translations of the root face (u = 0).



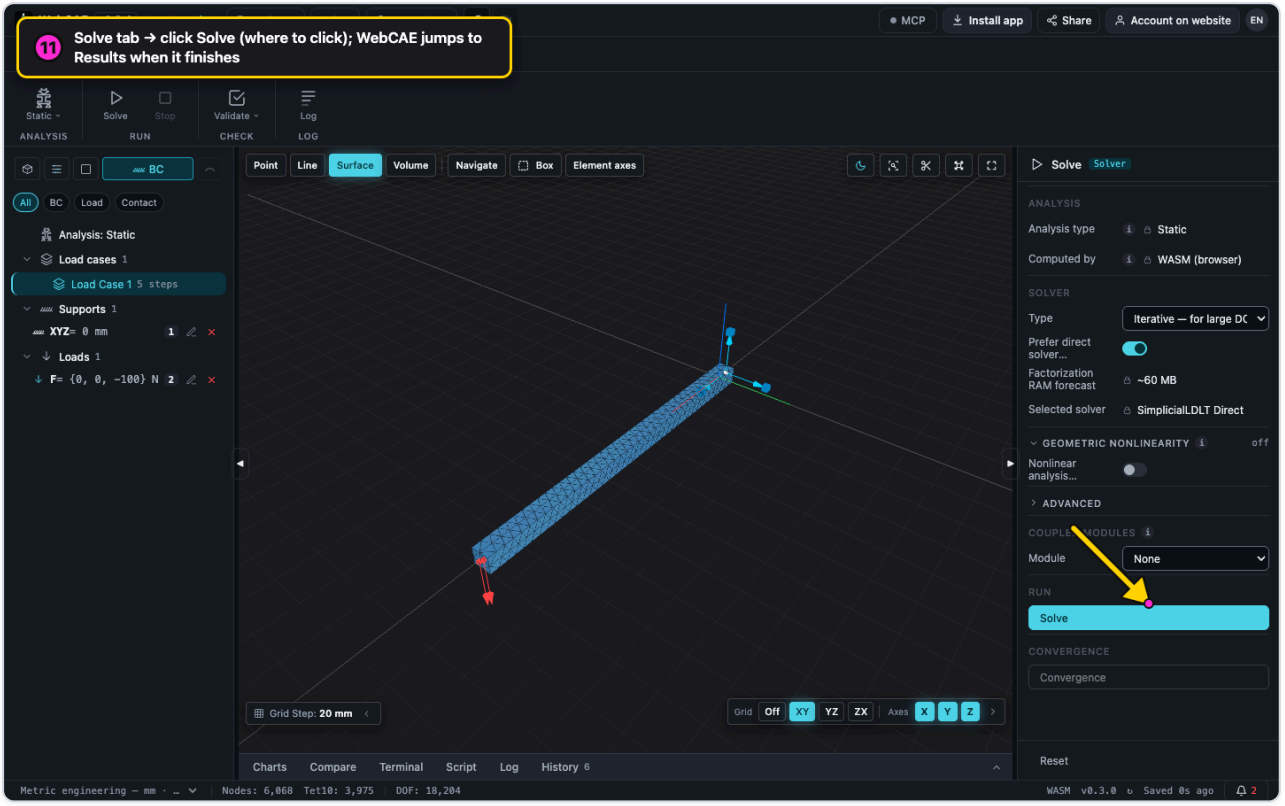
STEP 10 Apply the 100 N tip force

Click **Force**. Click **Enter ID**, type `2` (the free end from step 6), press **Enter**. Choose the **Z** component, type `−100` and check that the line **Will be saved** reads `F = {0, 0, −100} N`. Click **Apply**.



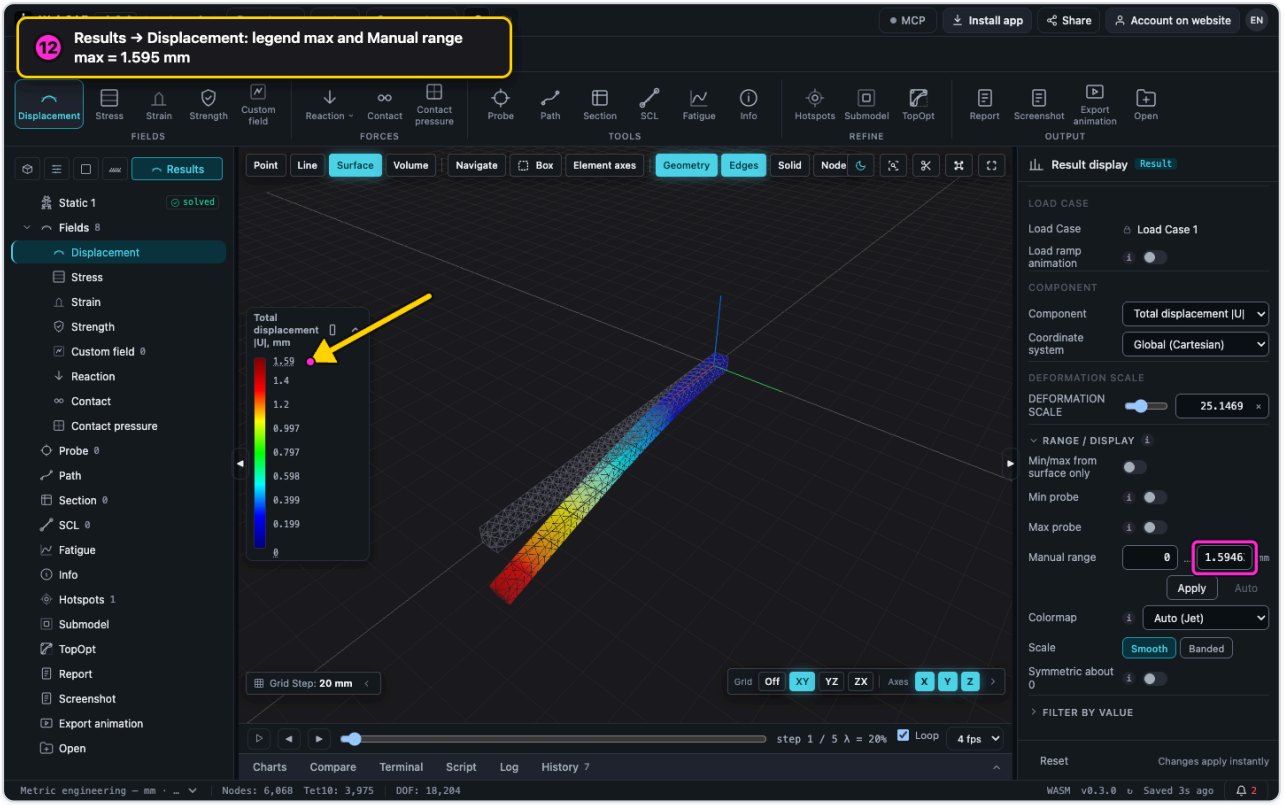
STEP 11 Run the static solver

Open the **Solve** tab and click **Solve**. Wait a few seconds: when the solver finishes, WebCAE switches to the **Results** tab by itself.



STEP 12 Read the tip deflection

You are already on the **Results** tab and the **Displacement** contour is already shown (its tile in the ribbon is selected; click it if yours is not). The legend's top value and the **Manual range** maximum show the largest displacement, **1.5946 mm** (legend: 1.59); it sits at the free end.



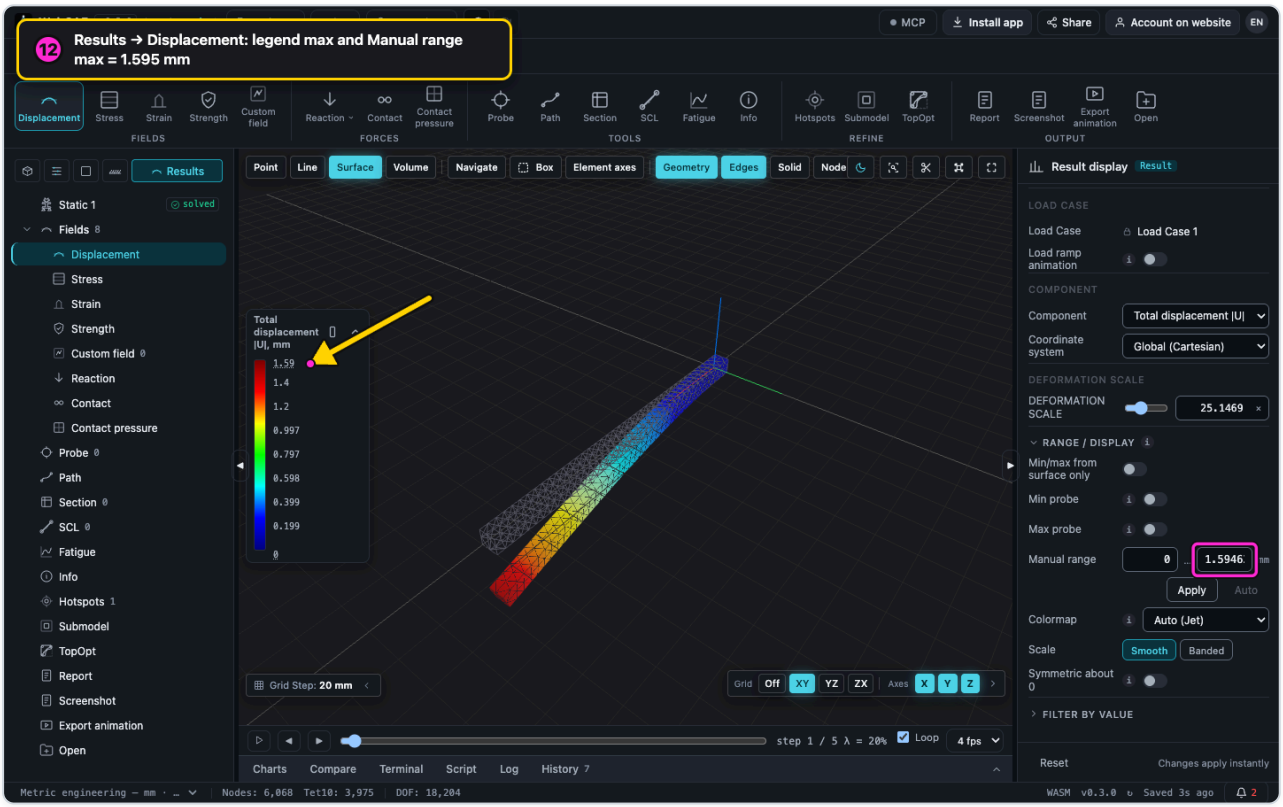
VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Error web / native
Tip deflection δ	$\frac{PL^3}{3EI} = \frac{100\times200^3}{3\times200000\times833.33} = 1.600$ mm	1.595 mm (Step 12, legend and Manual range)	1.596 mm (probe at the tip, from a separate desktop run; not on a frame)	0.34 % / 0.26 %

FORMULA

$\delta = \frac{PL^3}{3EI} = \frac{100\cdot200^3}{3\cdot200000\cdot833.33} = 1.600$ mm (Timoshenko shear adds 0.003 mm $\rightarrow 1.603$ mm). Both numbers are within 5 % of it.



TROUBLESHOOTING & NOTES

Common mistakes

- **Forgetting to select the force axis:** The Force panel opens with no pre-selected axis. Always click the specific component chip (such as **Z**) and check the `Will be saved: F = {0, 0, −100} N` preview before clicking Apply.
- **Wrong face numbers:** the numbers in this lesson come from the tree of a freshly created box. If your tree shows different numbers, find the end faces with Measure (step 6): the pair that is 200 mm apart.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#2 — A bracket by operations, and is it strong enough? (F02)** to build a part from boolean cuts, holes and fillets.