WebCAE
CLOUD & DESKTOP ENGINEERING SIMULATION & CAD PLATFORM

WEBCAE 101 · COMPLETE PRACTICAL GUIDE

WebCAE Simulation Handbook

30 Step-by-Step Hands-on Engineering Tutorials: From Direct CAD Modeling, Sheet Metal, Assemblies, and Defeaturing to Finite Element Meshing, Shells, Beams, Thermal and Multiphysics. Every number is reproduced by a product run.

Table of Contents & Tutorial Index

30 Lessons · Page Index

F01: A cantilever beam in twelve steps p. 2
F02: A bracket by operations, and is it strong enough? p. 16
F03: Meshing: when is the mesh good enough? p. 34
F04: Natural frequencies in WebCAE p. 52
F05: Column buckling in ten steps p. 63
F06: Steady heat conduction in a cooling fin p. 75
F07: CAD import and multi-surface boundary conditions p. 88
F08: Two bodies, one load path: assembly contact p. 100
F09: Load cases: two loadings on one model p. 114
F10: From clicks to code: the DSL p. 127
F11: Sketch on Face, Project 3D and Extrude p. 139
F12: Thin-walled housing, direct modeling offset and measure p. 154
F13: Hub with a Bore and Four Pins (Circular Pattern) p. 166
F14: Split Body and a Half-Model Cantilever p. 177
F15: Defeaturing and Midsurface p. 191
F16: Interference Resolve and a CFD Enclosure around the Whole Assembly p. 202
F17: Spur Gear Pair: Static Tooth Contact Under Driving Torque p. 214
F18: Planetary Gear Stage: Static Tooth Contact Under Sun Torque p. 229
F19: Register the official WebCAE MCP Server in Claude Code CLI (one-time setup): p. 243
F20: ISO 7206-4 Total Hip Arthroplasty Implant (Ti-6Al-4V + CoCrMo) p. 257
F21: Assembly Mates on Real Bodies: Ground, 14 Mates, Degrees of Freedom and BOM Mass p. 271
F22: Two Flanges on a Preloaded Bolt: Bolt Connector, Preload and Load Sharing p. 283
F23: 3D Portal Frame with 1D Beams: Profiles, Orientation, and N-Q-M Diagrams p. 296
F24: Thin-Walled Pressure Vessel with Shell Elements: Membrane vs Bending Stress p. 308
F25: Stiffened Aerospace Panel: Coupled Shell + Eccentric Beam Stringers p. 320
F26: Support Reactions: A Plate-and-Lever Shell Model and a Solid Bracket with a Remote Load p. 331
F27: Unilateral Contact: Hertz Line Contact of a Cylinder on a Block, Contact Pressure p. 344
F28: Snap-Arm Release Force: Prescribed Displacement and Support Reactions p. 357
F29: Steady and Transient Heat Transfer: Heat Flux, Convection, and Thermal Time Constant p. 367
F30: Coupled Thermo-Elasticity: Restrained Thermal Expansion and Bimetallic Strip Bending p. 377

WebCAE Engineering Documentation · Complete Edition · 30 Lessons
Run ID: 2026-11-04-32ffaed2 · 30-Lesson Curriculum

WebCAE 101

LESSON 1 · F01

A cantilever beam in twelve steps

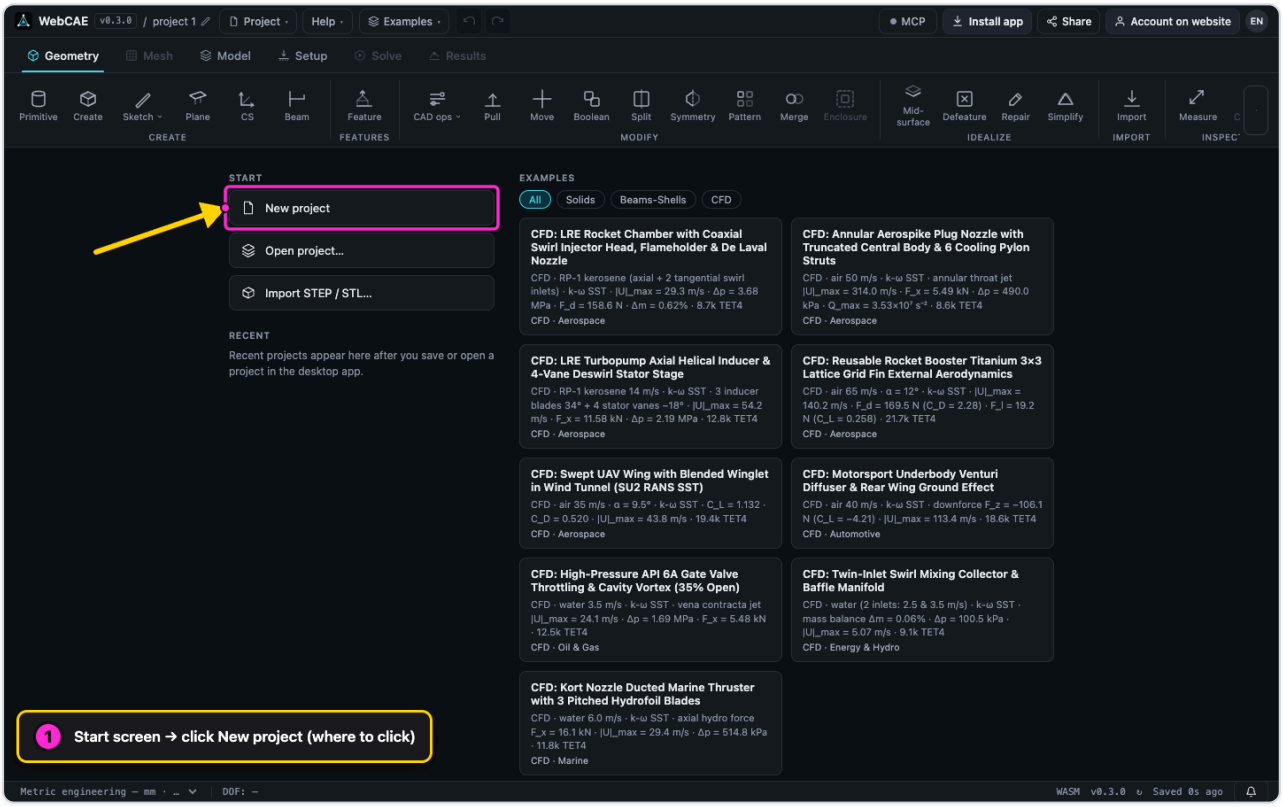
Goal Compute the tip deflection of a steel cantilever and check it against the textbook formula.

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

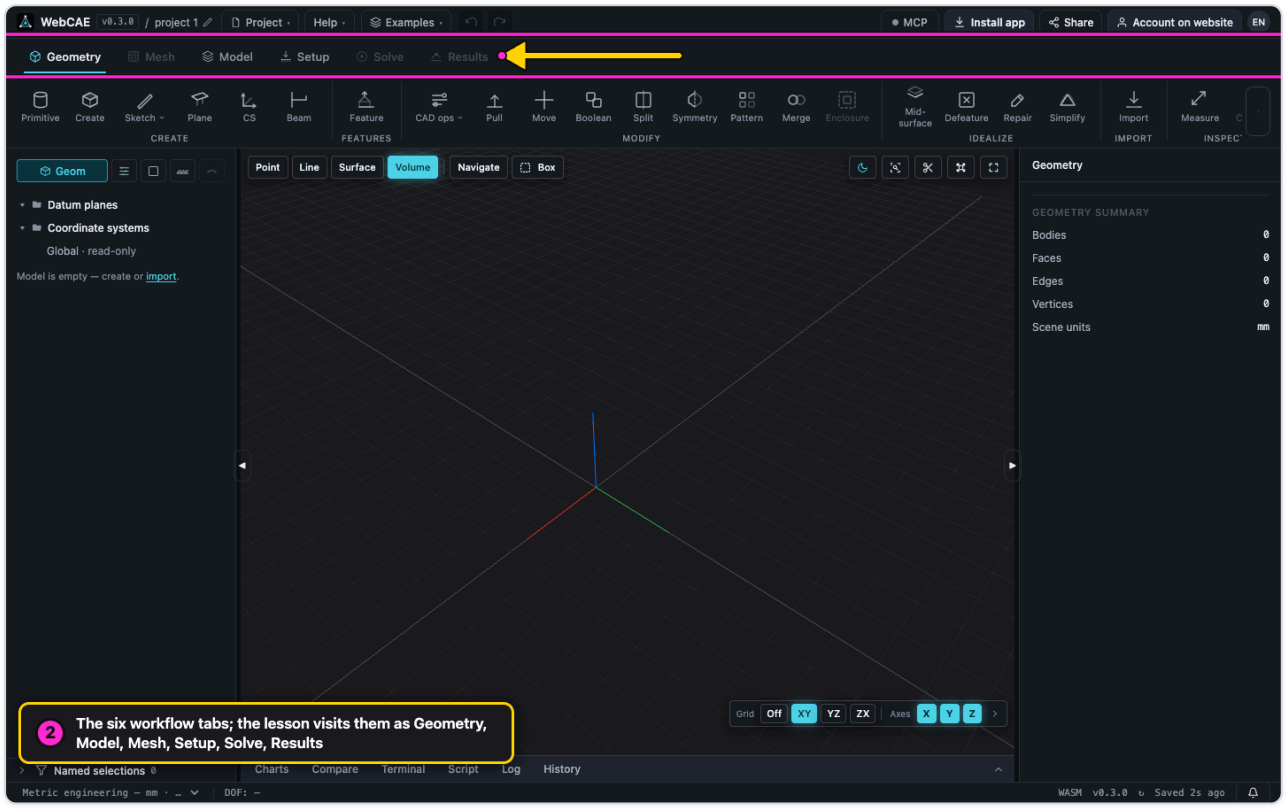
When WebCAE opens, it shows the **Start** screen: **New project**, **Open project...**, **Import STEP / STL...** and a gallery of examples. While it is on screen the workspace behind it is not usable (a ribbon button highlights, but its panel does not open). Click **New project**: the Start screen closes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



STEP 2

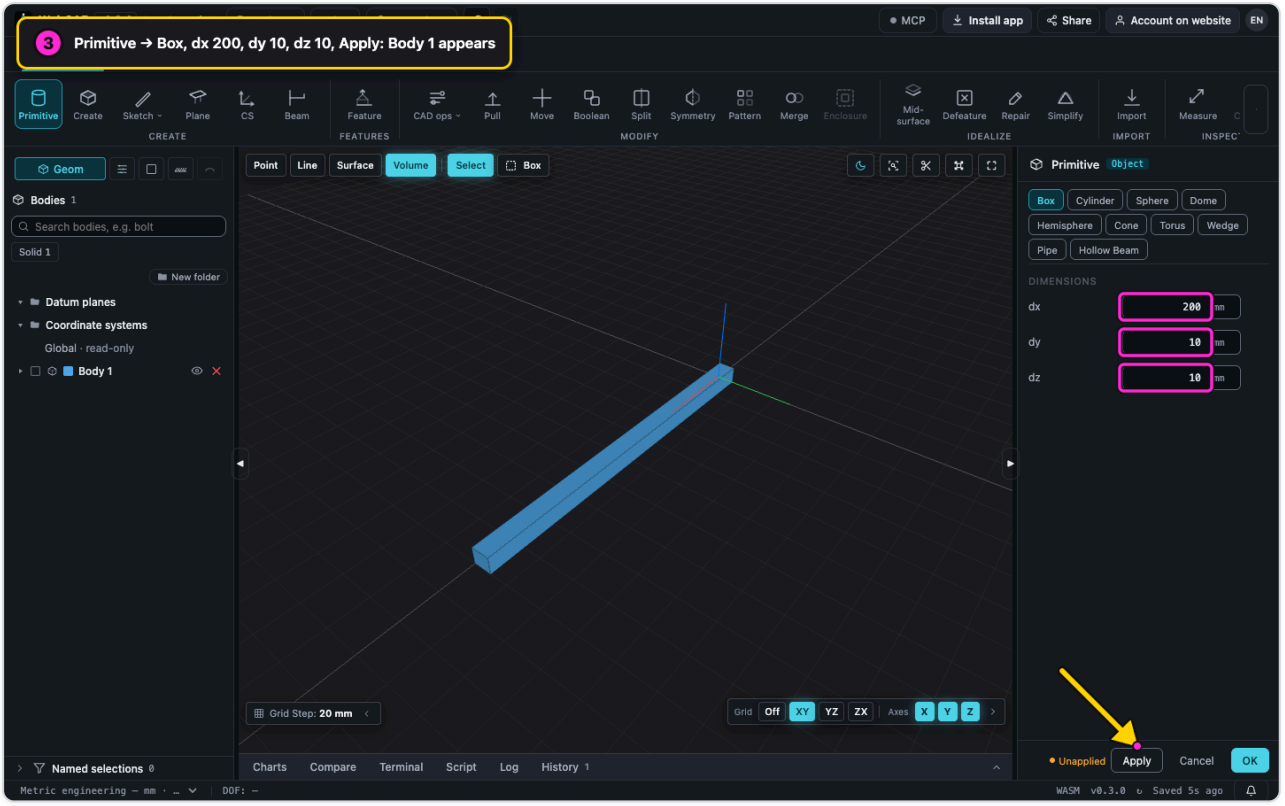
See the six workflow tabs

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.



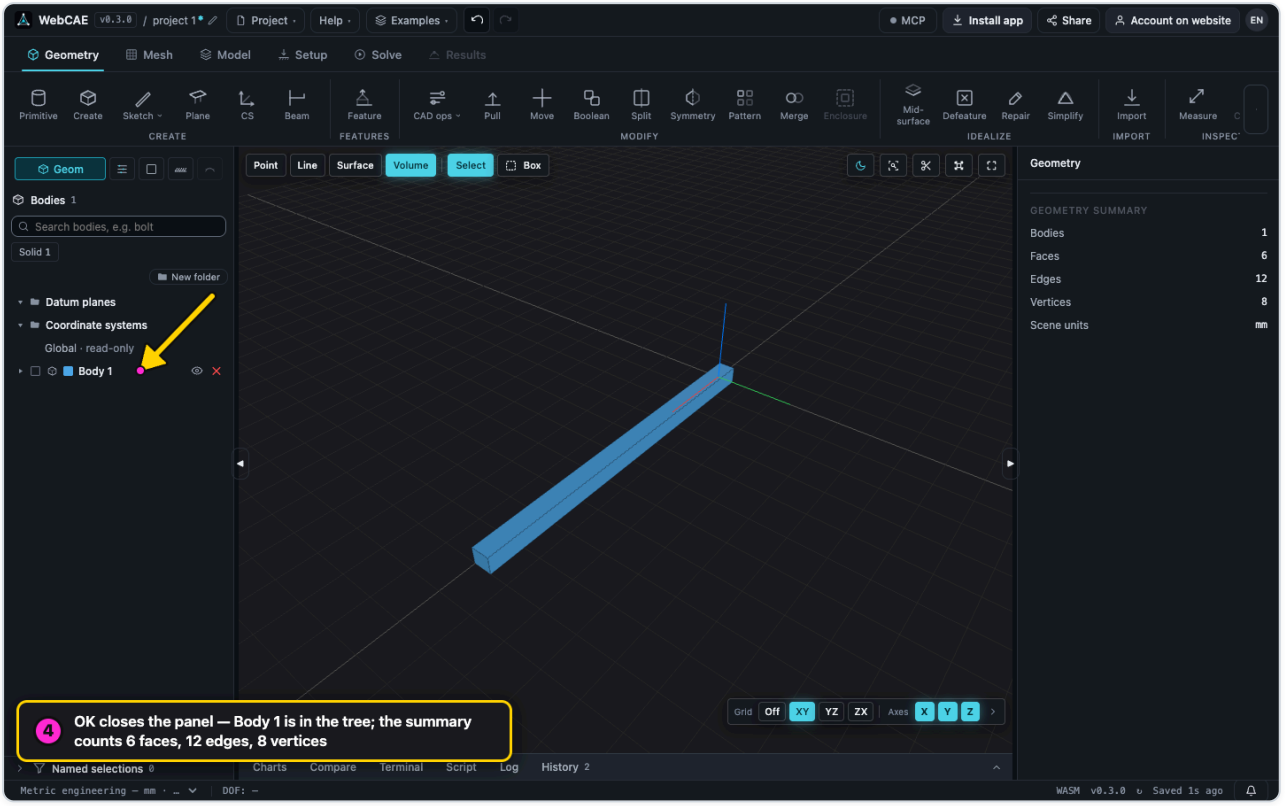
STEP 3 Create the 200 × 10 × 10 mm bar

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.



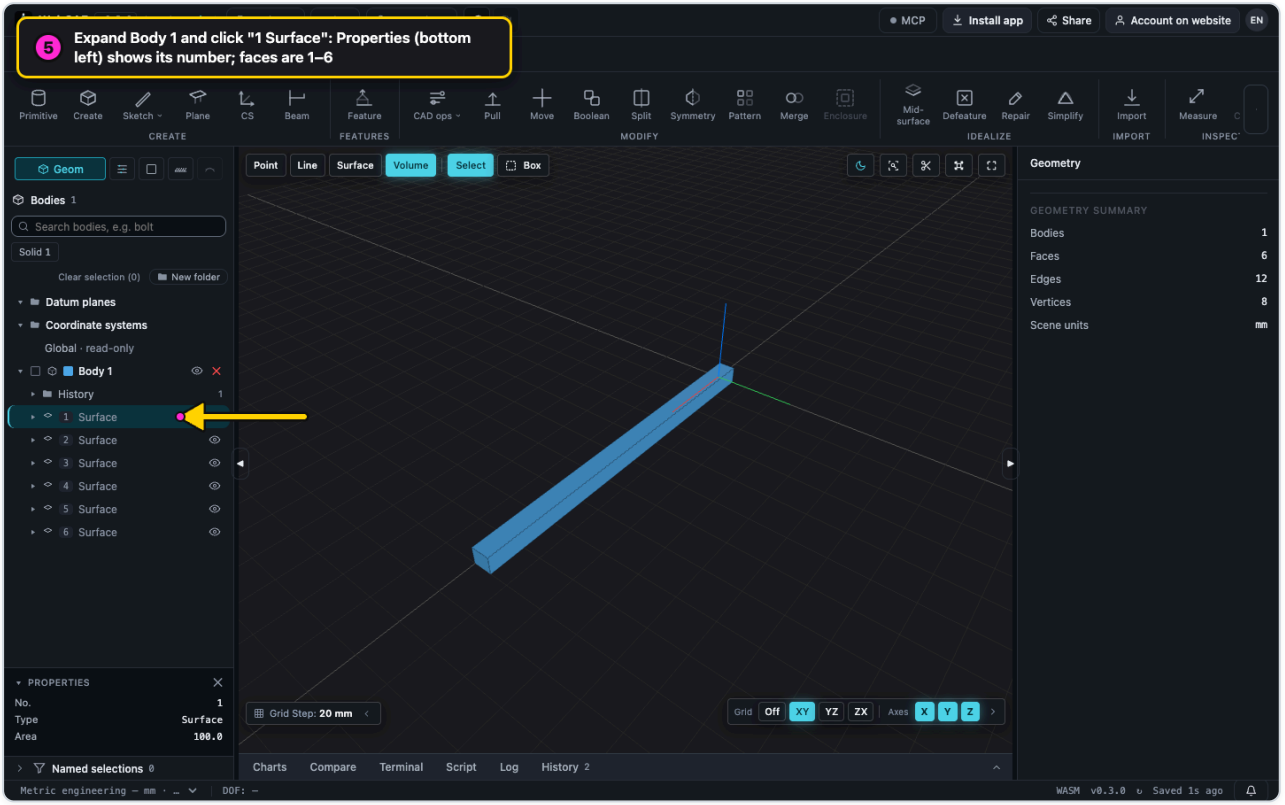
STEP 4 Close the panel

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.



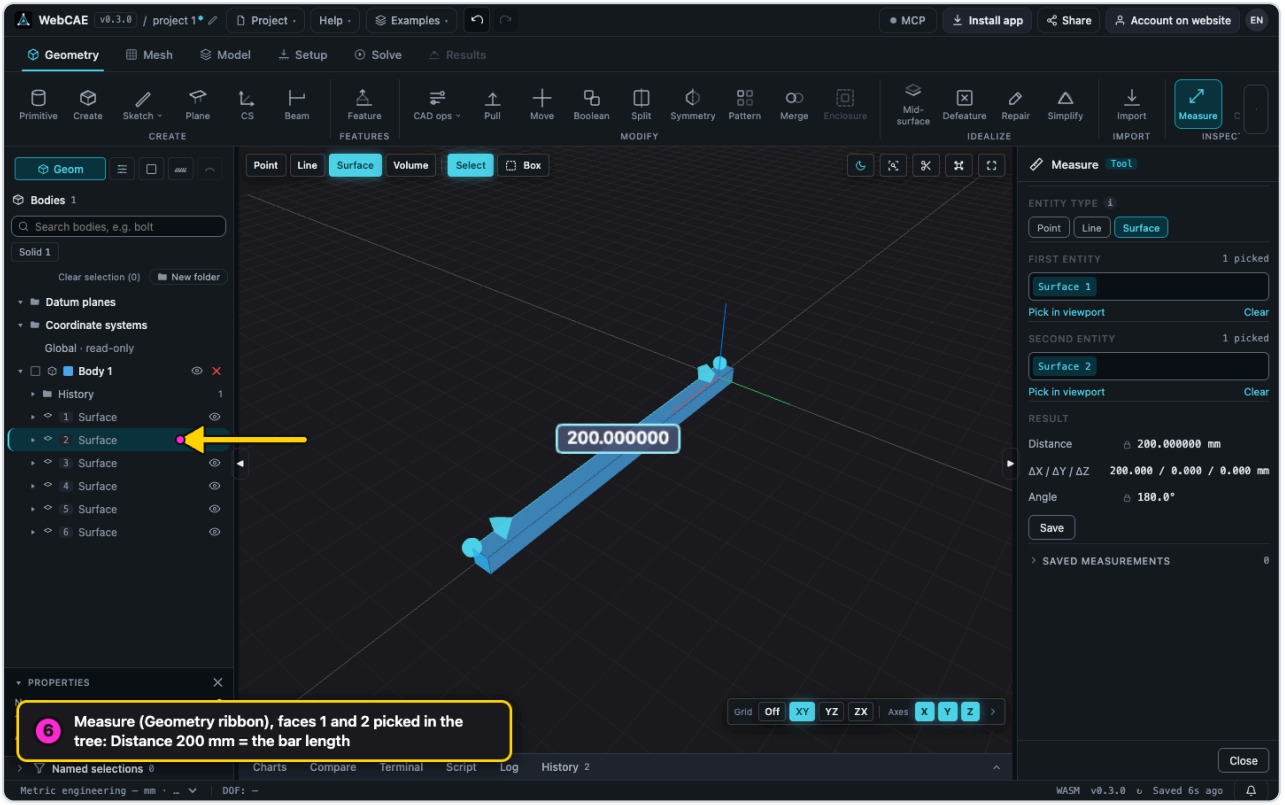
STEP 5 Find a face number

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

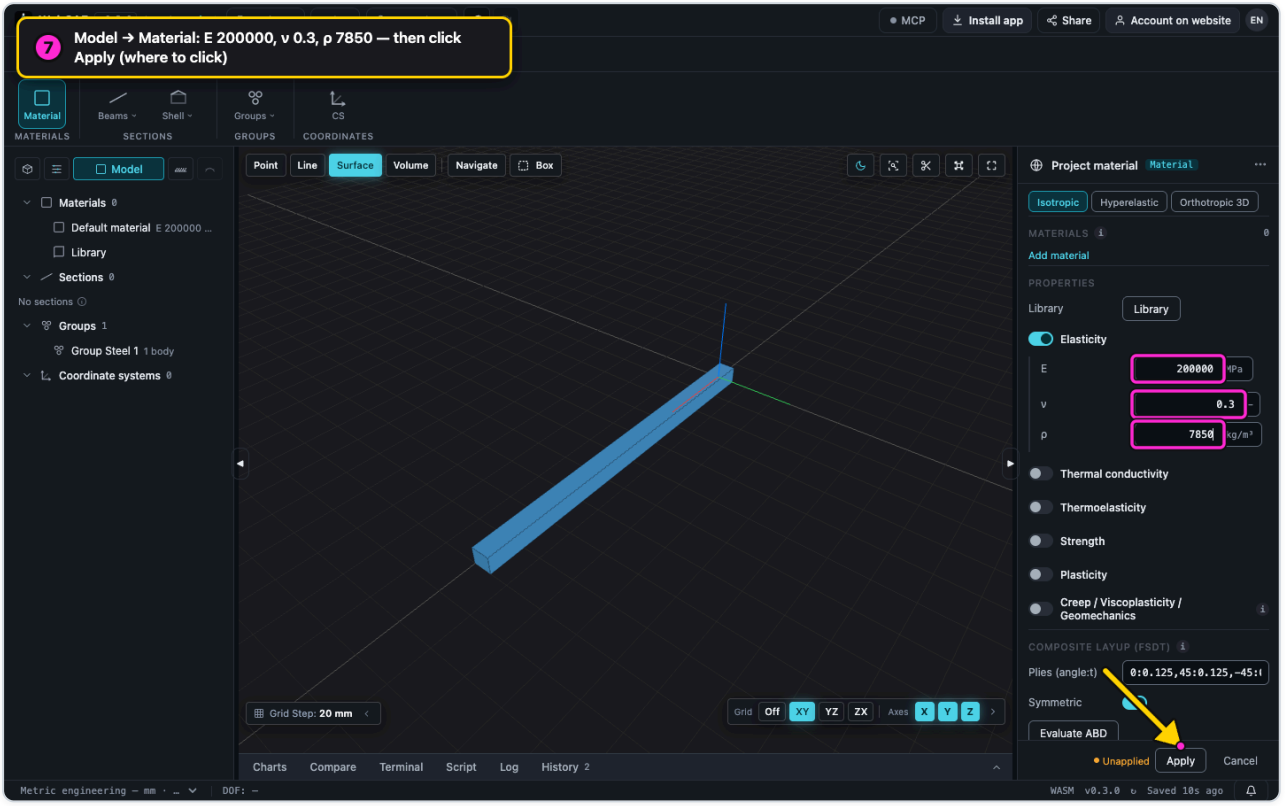
Click **Measure** in the **Geometry** ribbon (Inspect group; the entity type is **Surface**), then click **1 Surface** and **2 Surface** in the tree. The panel's **Distance** reads **200.000000 mm** (ΔX / ΔY / ΔZ = 200 / 0 / 0): the two end faces are 200 mm apart, so face 2 is the free end. Click **Close**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



STEP 7

Assign structural steel properties

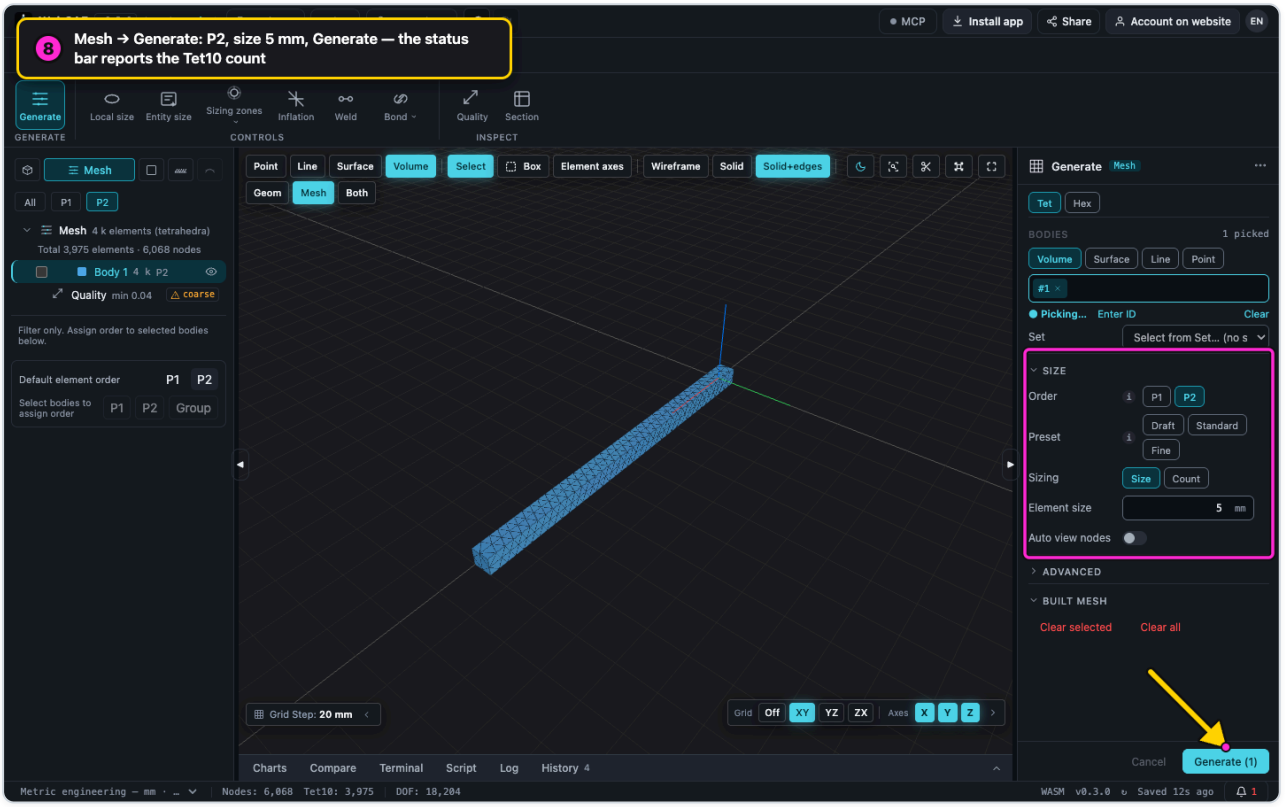
Open the **Model** tab and click **Material**. Enter Young's modulus **E** = `200000` MPa, Poisson's ratio **ν** = `0.3` and density **ρ** = `7850` kg/m³, then click **Apply**.



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.

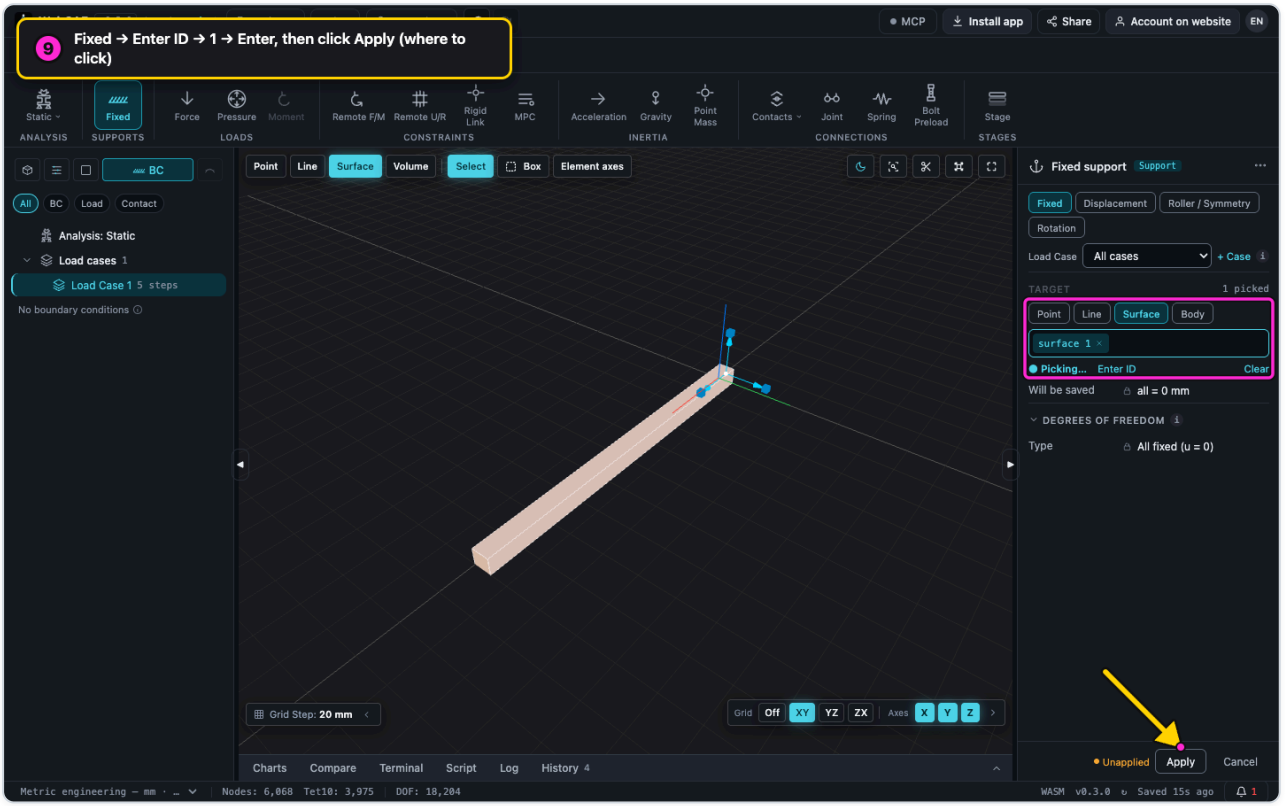
> **WHY** Quadratic 10-node tetrahedra (Tet10) do not lock in bending, so a coarse mesh is enough.



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*.

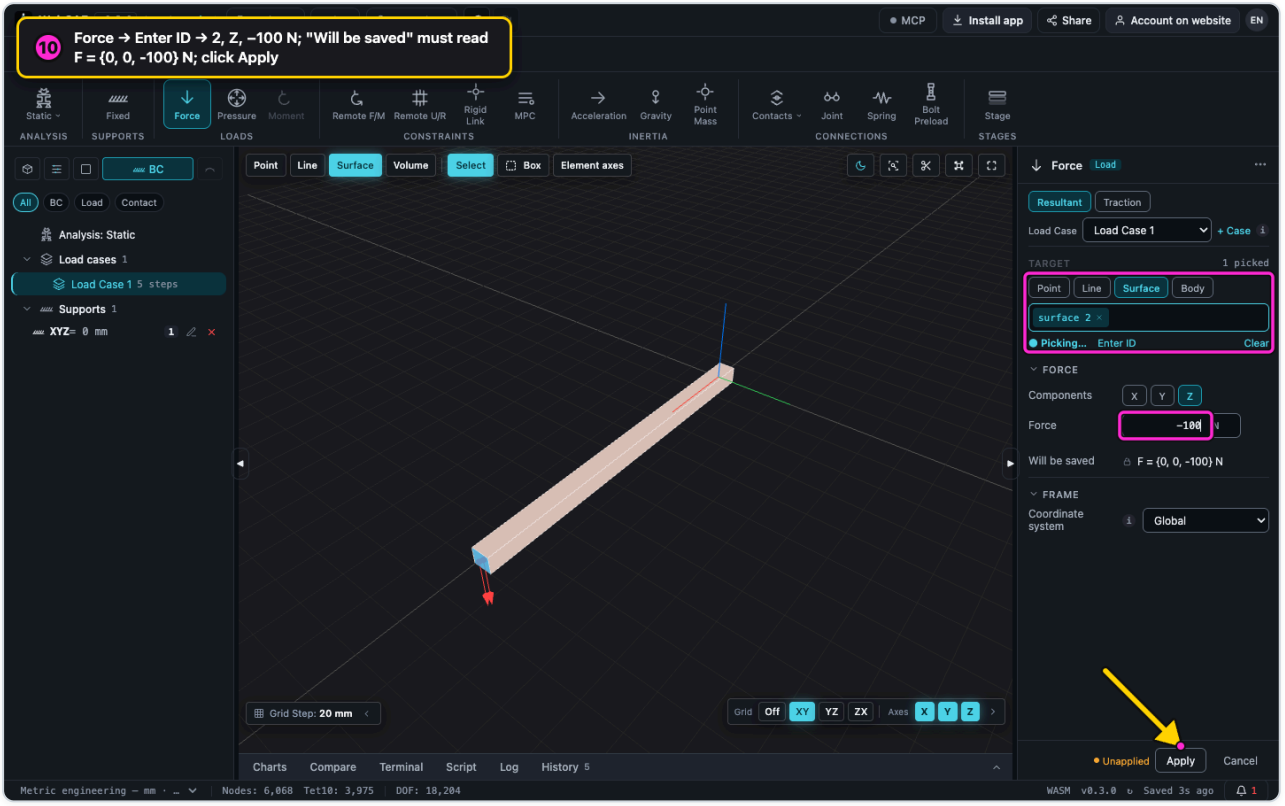
WHY A fixed support removes all three translations of the root face (u = 0).



STEP 10

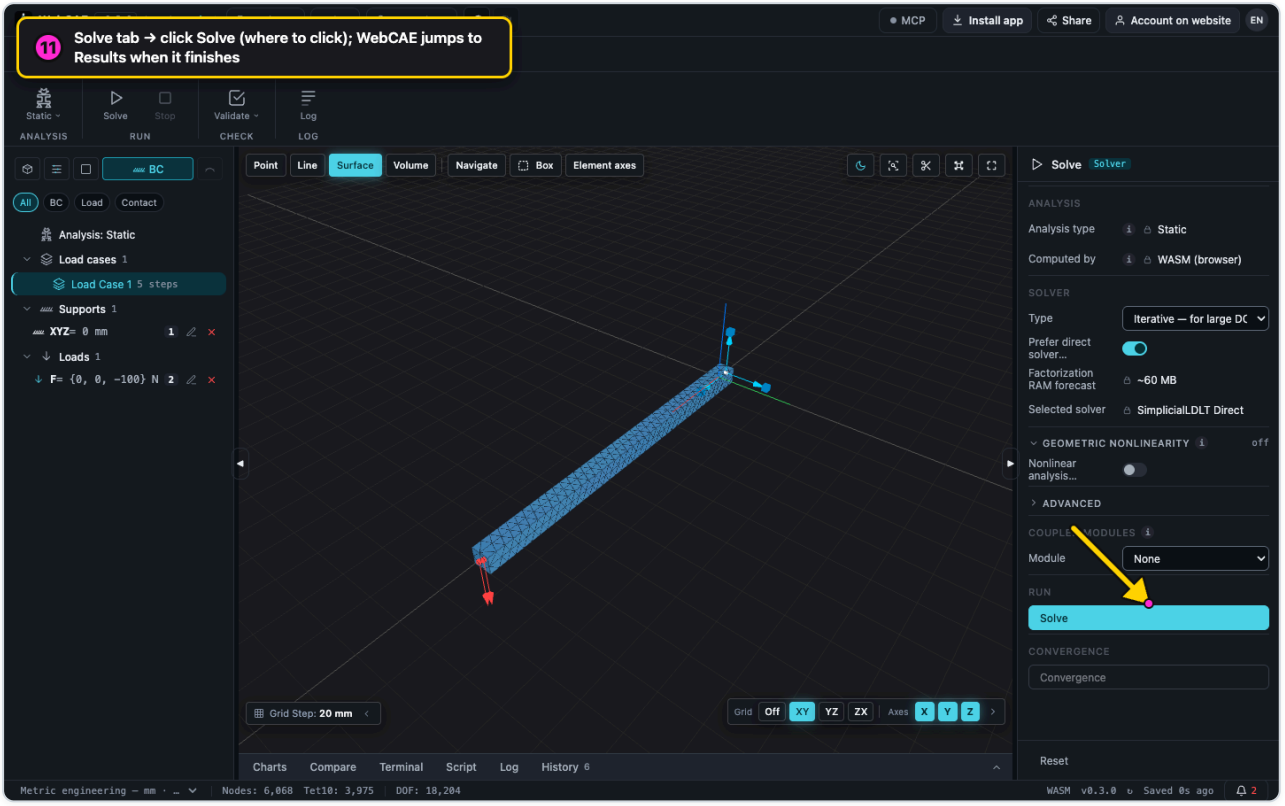
Apply the 100 N tip force

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

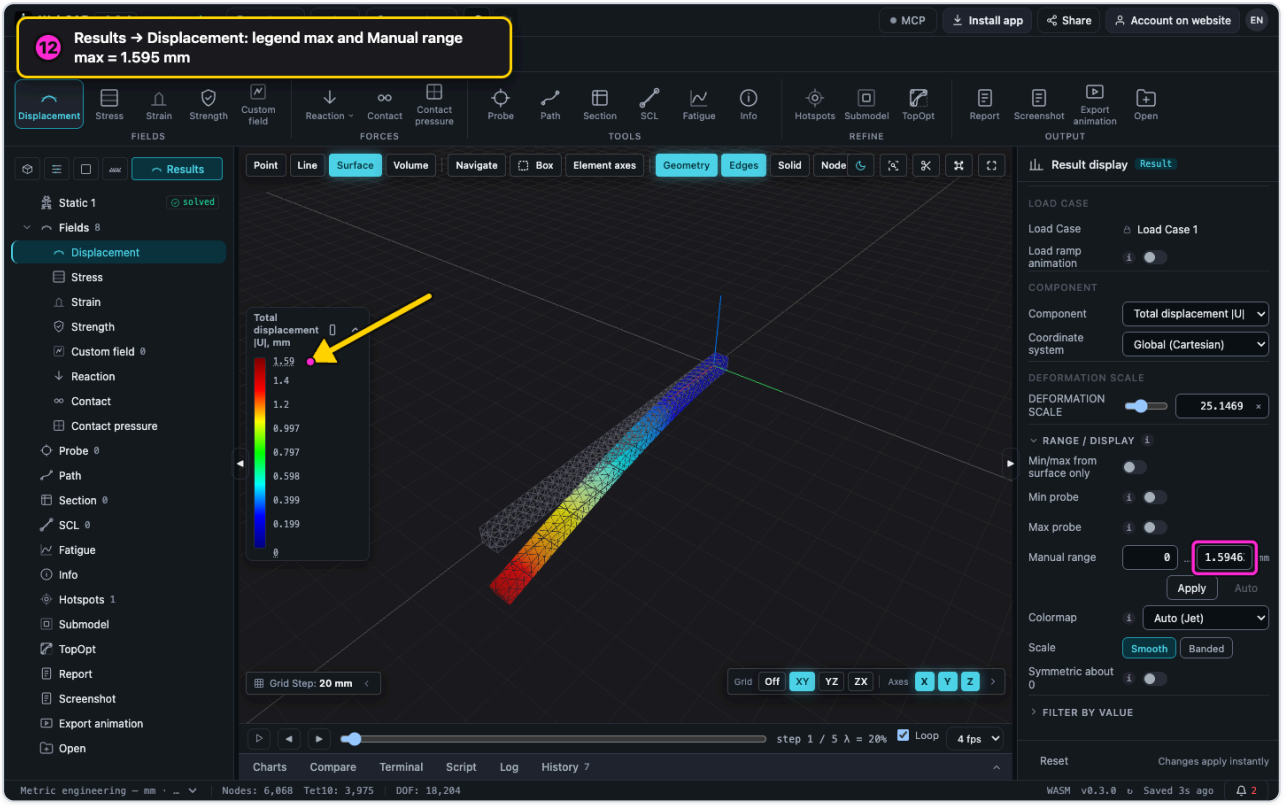
Open the **Solve** tab and click **Solve**. Wait a few seconds: when the solver finishes, WebCAE switches to the **Results** tab by itself.



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.



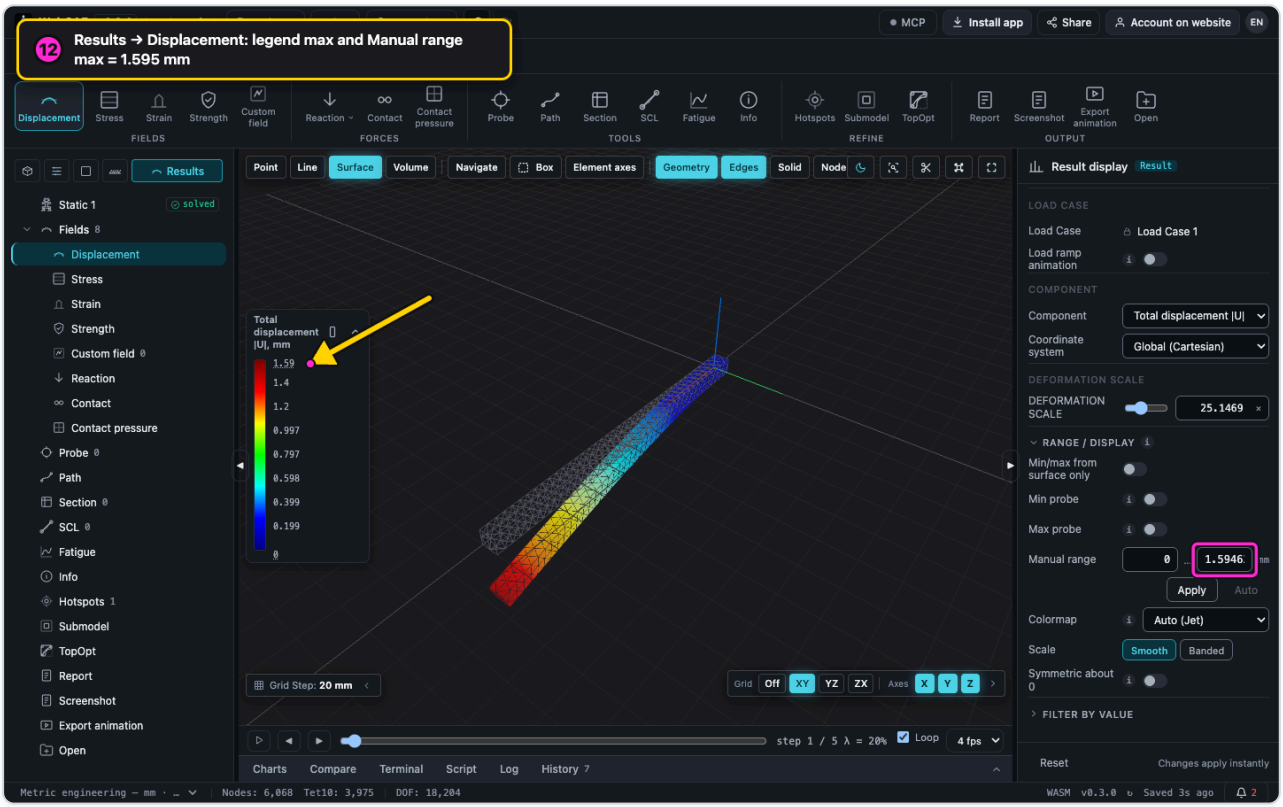
VERIFICATION BENCHMARK

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

$\delta = \frac{PL^3}{3EI} = \frac{100\cdot200^3}{3\cdot200000\cdot833.33} = 1.600\ \text{mm}$ (Timoshenko shear adds $0.003\ \text{mm} \rightarrow 1.603\ \text{mm}$). 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.

WebCAE 101

LESSON 2 · F02

A bracket by operations, and is it strong enough?

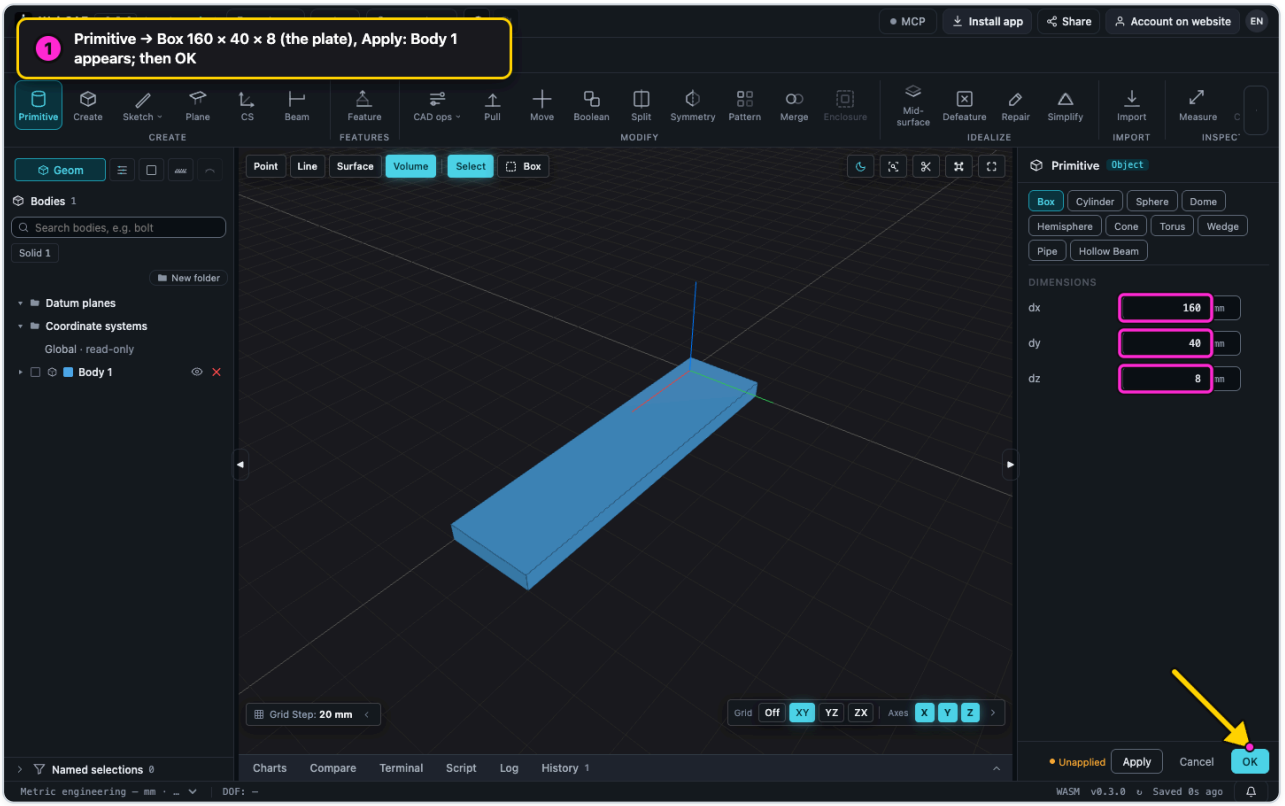
Goal Build a plate with a Boolean Cut, a Hole and Fillets, pull it with 10 kN and check the peak stress at the hole and the safety factor against Peterson's formula.

Outcome Peak von Mises stress 157.8 MPa (hand formula 156.8 MPa) and minimum safety factor 1.584 (hand 1.594) in WebCAE web; 157.4 MPa and 1.588 in WebCAE desktop.

12–15 min | web (WASM) | Run: 2026-11-04-32ffaed2 / F02

STEP 1 Create the plate

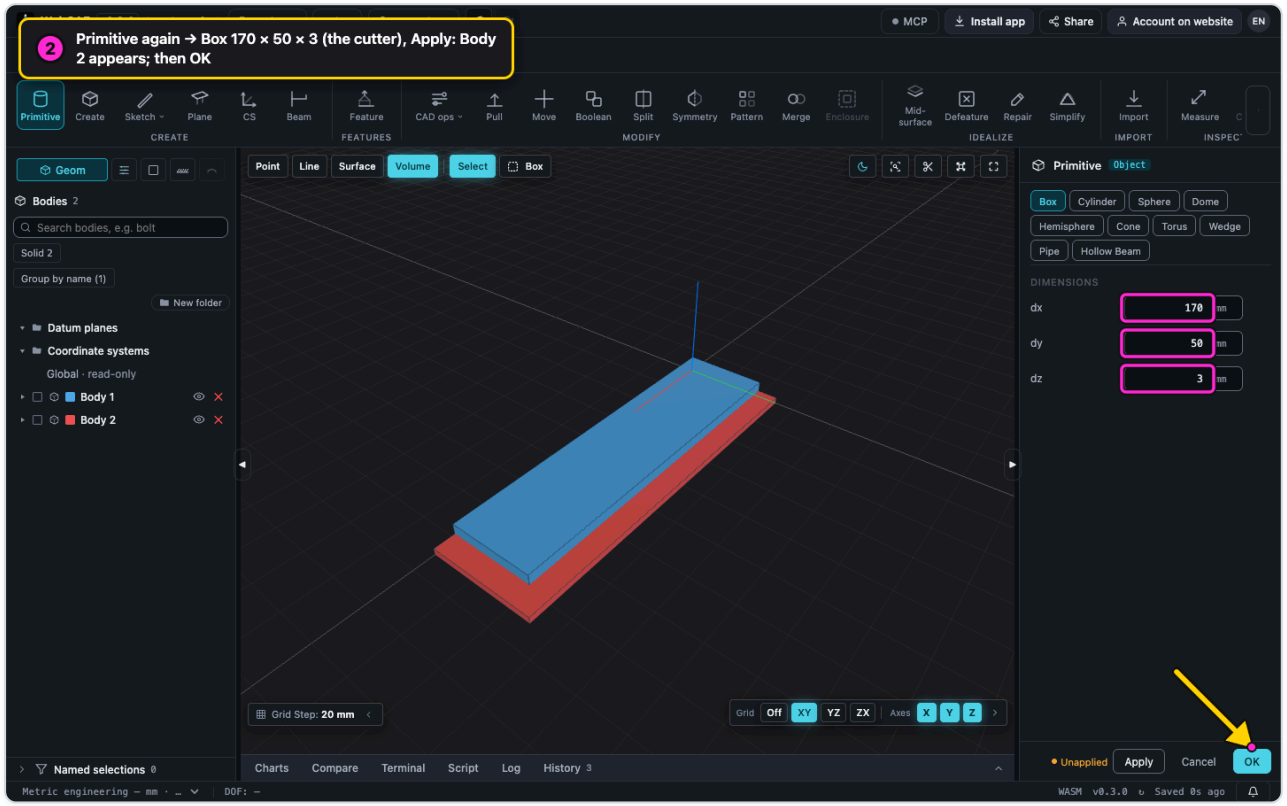
In **Geometry**, click **Primitive**, keep **Box**, type `160` in **dx**, `40` in **dy**, `8` in **dz**, click **Apply** (Body 1 appears), then **OK**.



STEP 2 Create the cutter

Click **Primitive** again: **Box**, **dx** `170`, **dy** `50`, **dz** `3`, **Apply** (Body 2 appears, a thin larger slab under the plate), then **OK**.

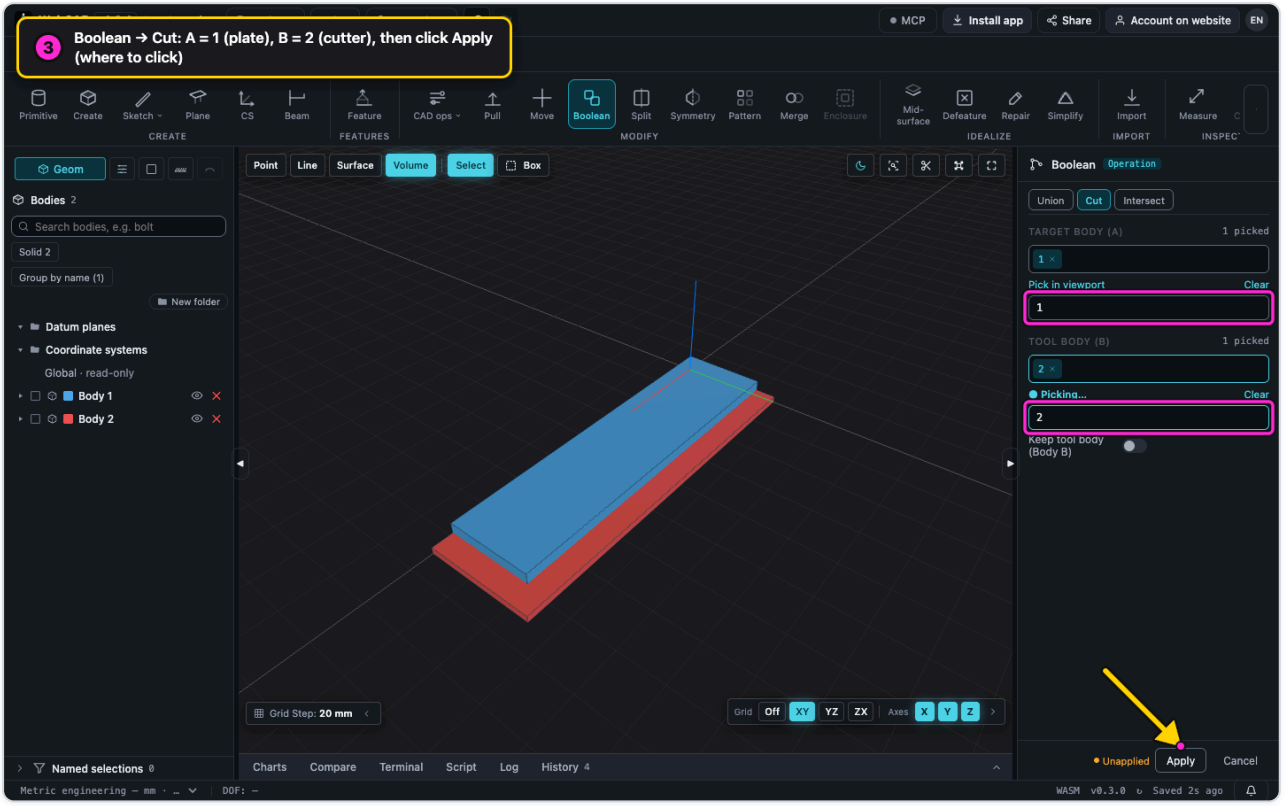
> **WHY** The slab covers the bottom 3 mm of the plate; subtracting it leaves a plate 8 – 3 = 5 mm thick.



STEP 3

Boolean Cut: body 1 minus body 2

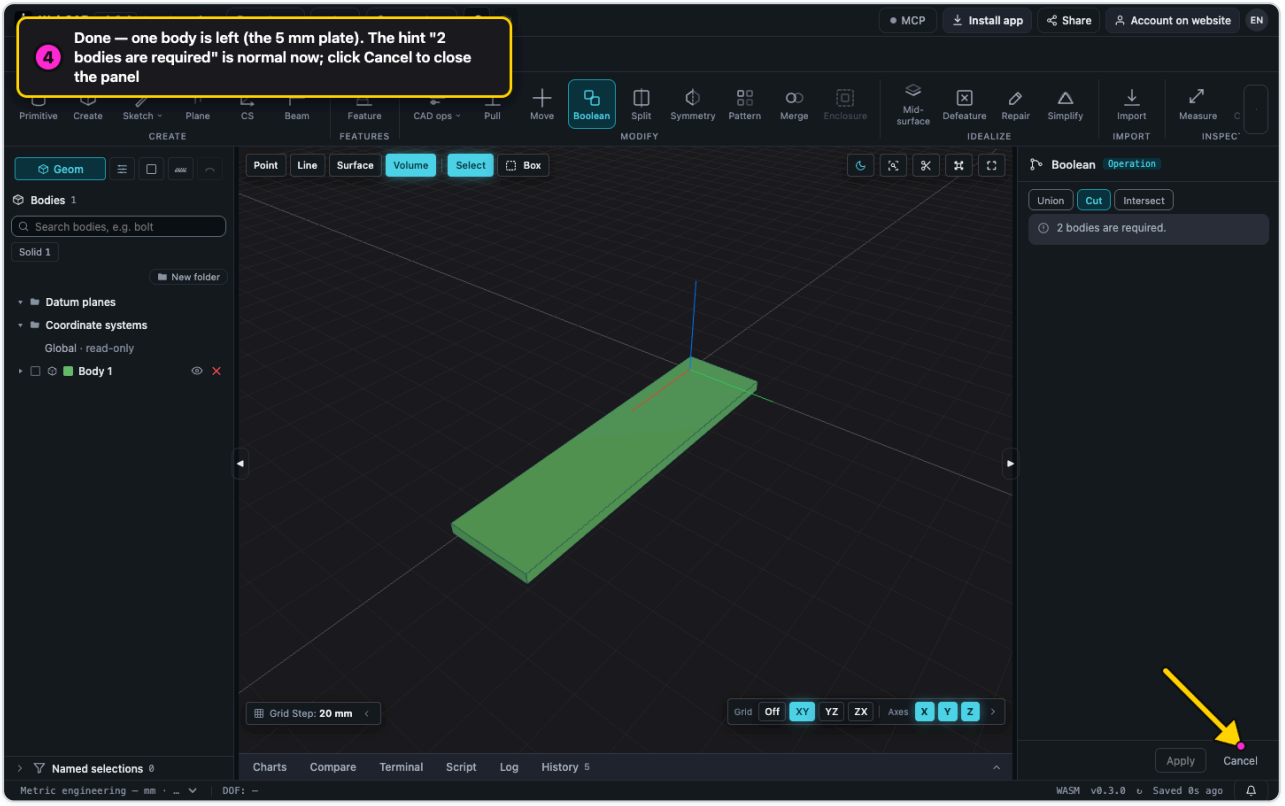
Click **Boolean** in the ribbon, choose **Cut**. Click in **Target body (A)**, type `1`, press **Enter**; click in **Tool body (B)**, type `2`, press **Enter**. Leave **Keep tool body (Body B)** off. Click **Apply**.



STEP 4

Close the Boolean panel

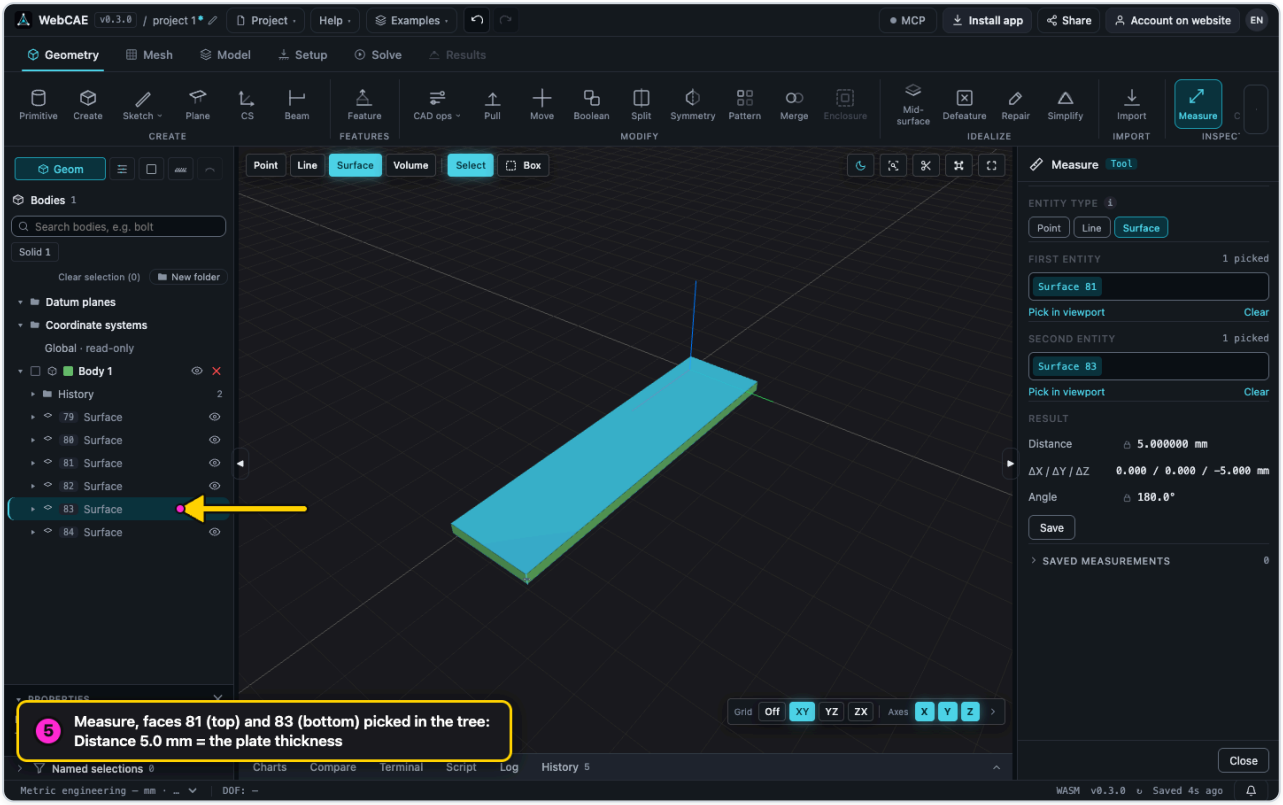
After Apply only one body is left, so the panel shows the hint **2 bodies are required** — that is expected, not an error. The tree shows a single **Body 1**. Click **Cancel** to close the panel.



STEP 5

Check the thickness with Measure

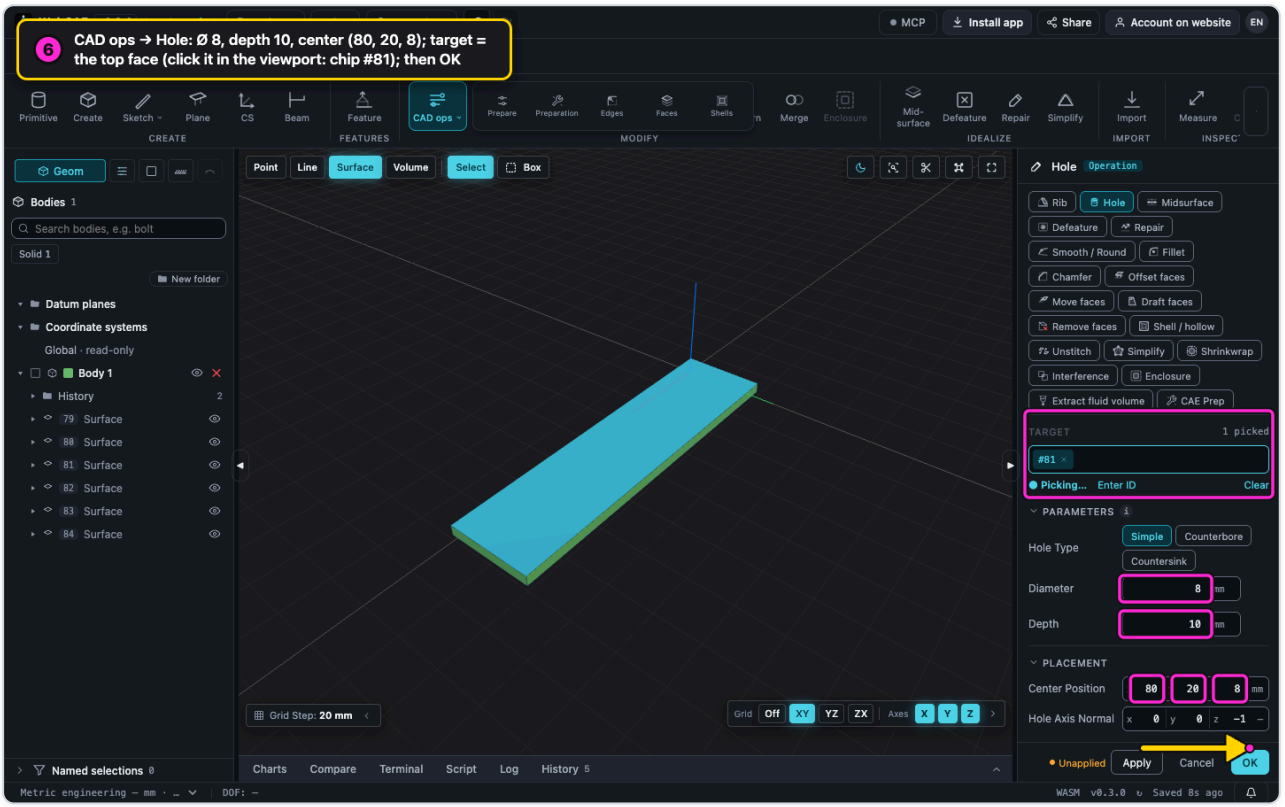
Click **Measure** in the Geometry ribbon (Inspect group), expand **Body 1** in the tree and click faces **81** (top) and **83** (bottom). The **Distance** reads **5.000000 mm**: the plate is 5 mm thick. These are the two big parallel faces of the plate; the top one lights up in the viewport as you pick it. Click **Close** to leave Measure.



STEP 6

Make the Ø 8 mm hole

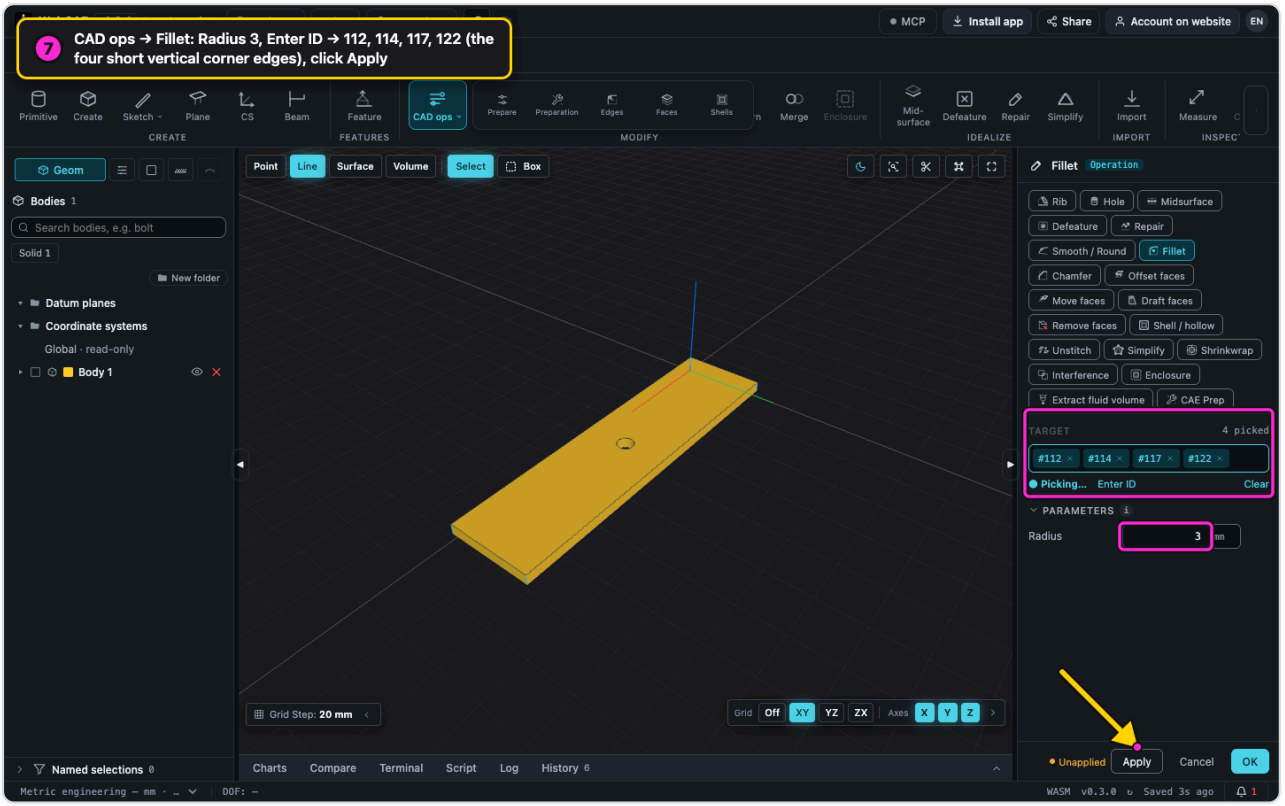
Click **CAD ops** and choose **Hole**. Type **Diameter** `8`, **Depth** `10`, **Center Position** x `80`, y `20`, z `8`; **Hole Axis Normal** stays x `0`, y `0`, z `-1`. The Target needs the face the hole starts in: click the big **top face** in the viewport (it lights up and the chip **#81** appears in Target — you can also type that number with **Enter ID**). Click **OK**.



STEP 7 Pick the four corner edges and fillet them

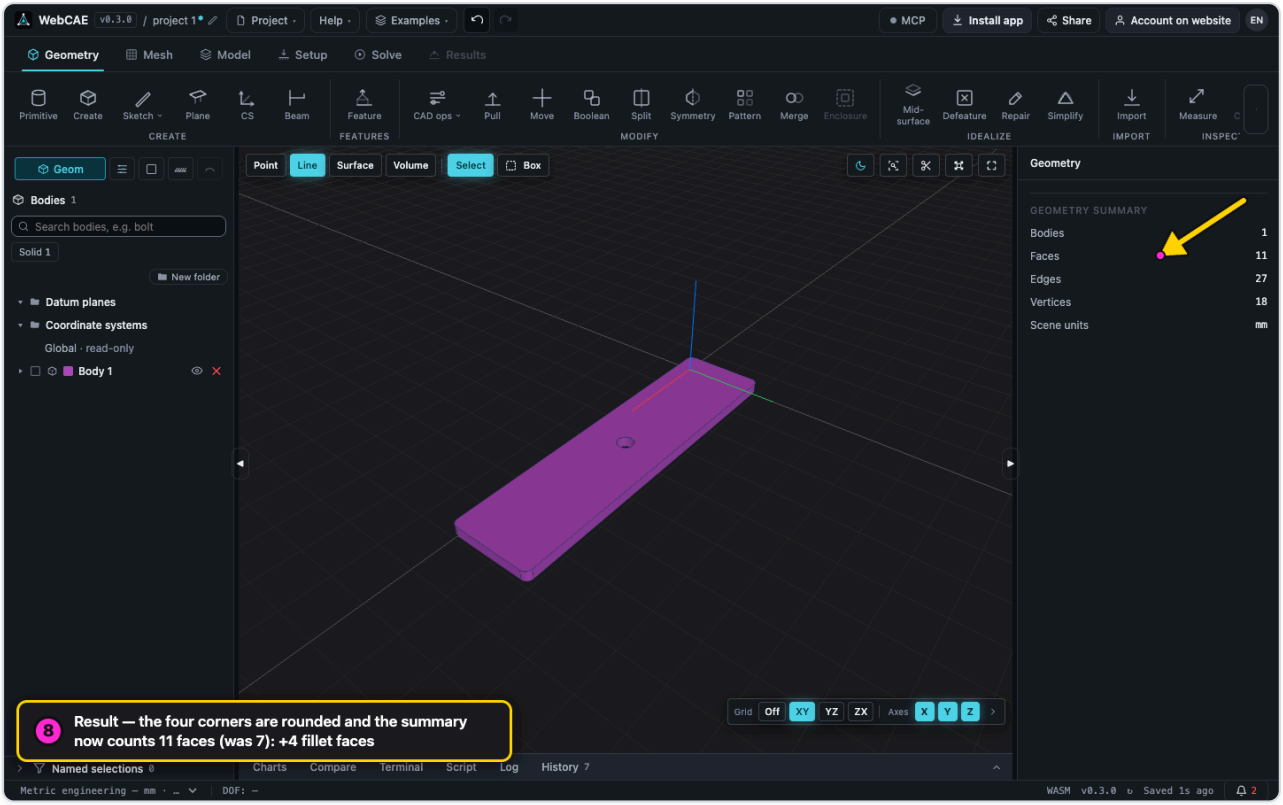
Click **CAD ops**, choose **Fillet**, type **Radius** `3`. Click **Enter ID** under Target and type `112, 114, 117, 122`, press **Enter**: four chips appear. These are the four short (5 mm) vertical edges at the plate's four corners — the edges that stand along Z at x = 0 / 160 and y = 0 / 40. Click **Apply**.

> **WHY** Rounding the corners keeps them from adding stress concentrations of their own.



STEP 8 Check the fillets

Click **Cancel** to close the panel (the fillets are already in the model; the target list is empty now). In the Geometry summary on the right **Faces** went from 7 to **11**: four new rounded faces. (The summary also counts the hole wall as a face.)

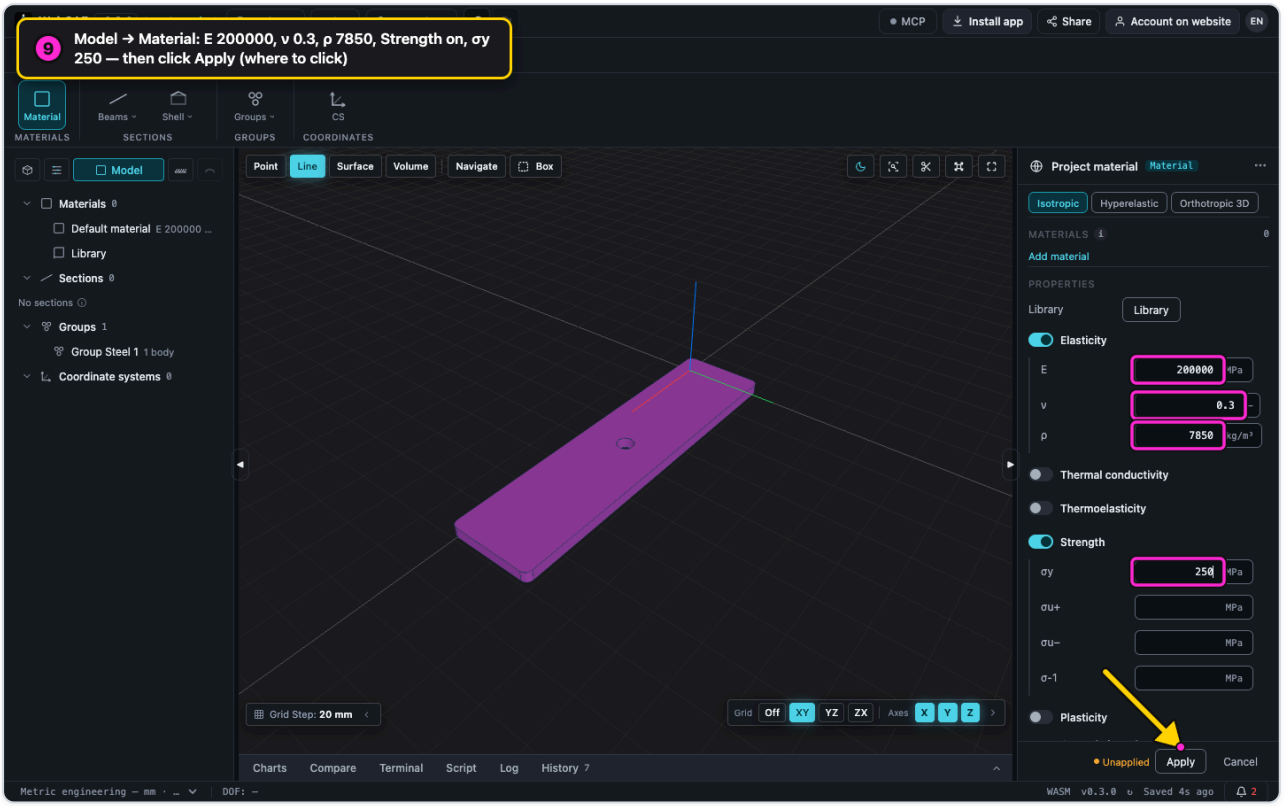


STEP 9

Material with a yield strength

Open **Model**, click **Material**. Type **E** `200000` MPa, **ν** `0.3`, **ρ** `7850` kg/m³. Switch **Strength** on and type **σy** `250` MPa. Click **Apply**.

WHY The safety factor is yield strength divided by the stress, so WebCAE needs σy.

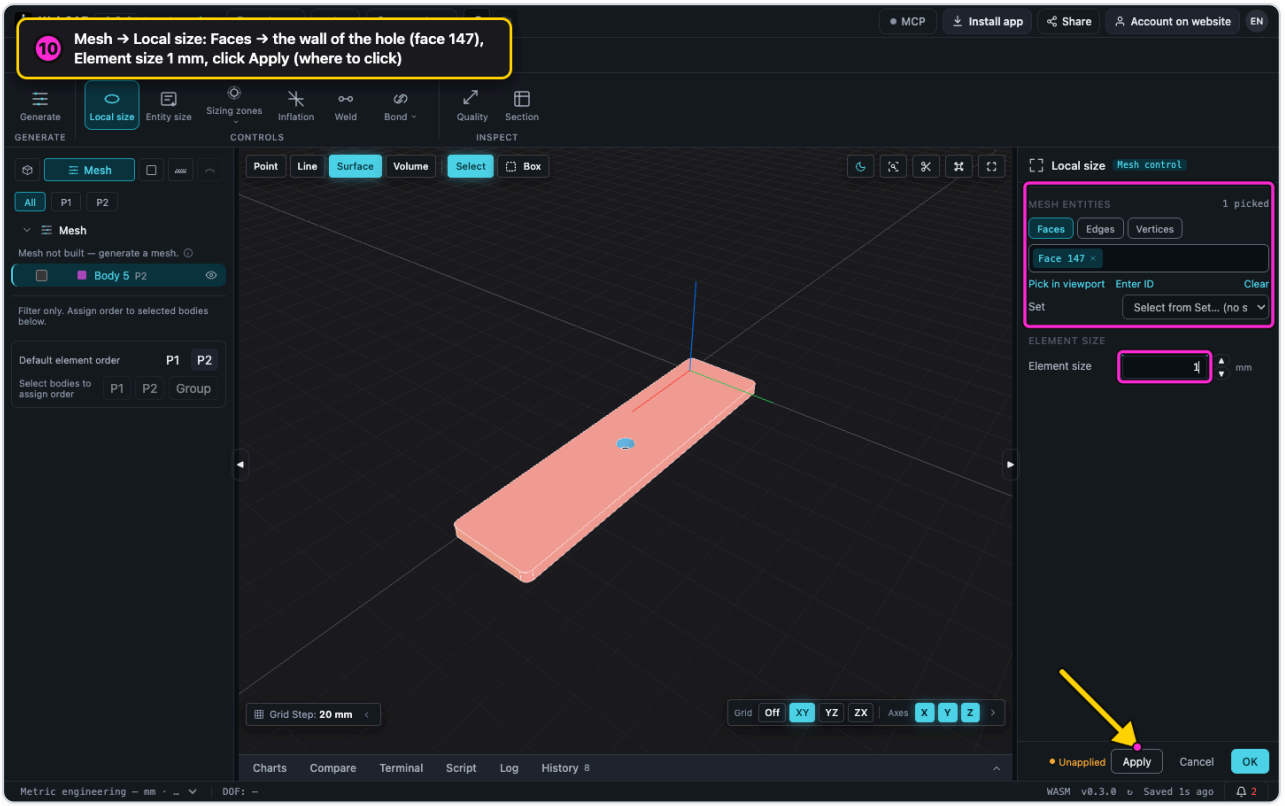


STEP 10

Ask for a fine mesh at the hole

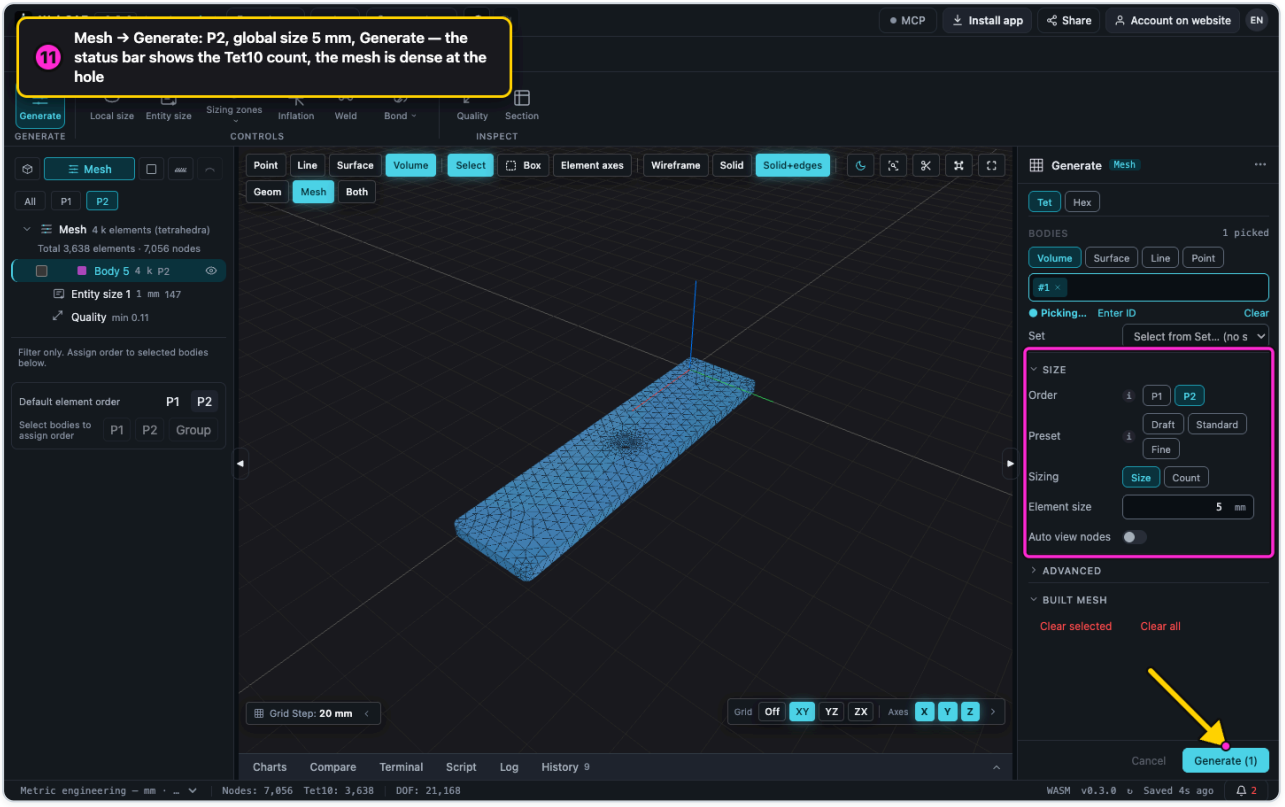
Open **Mesh**, click **Local size**. With **Faces** selected, click inside the hole ring in the viewport: the hole wall lights up and **Face 147** appears in the list (or type 147 with **Enter ID**). Type **Element size** `1` mm and click **Apply**.

> **WHY** The stress changes fast right at the hole; a 1 mm mesh there keeps the element count low elsewhere.



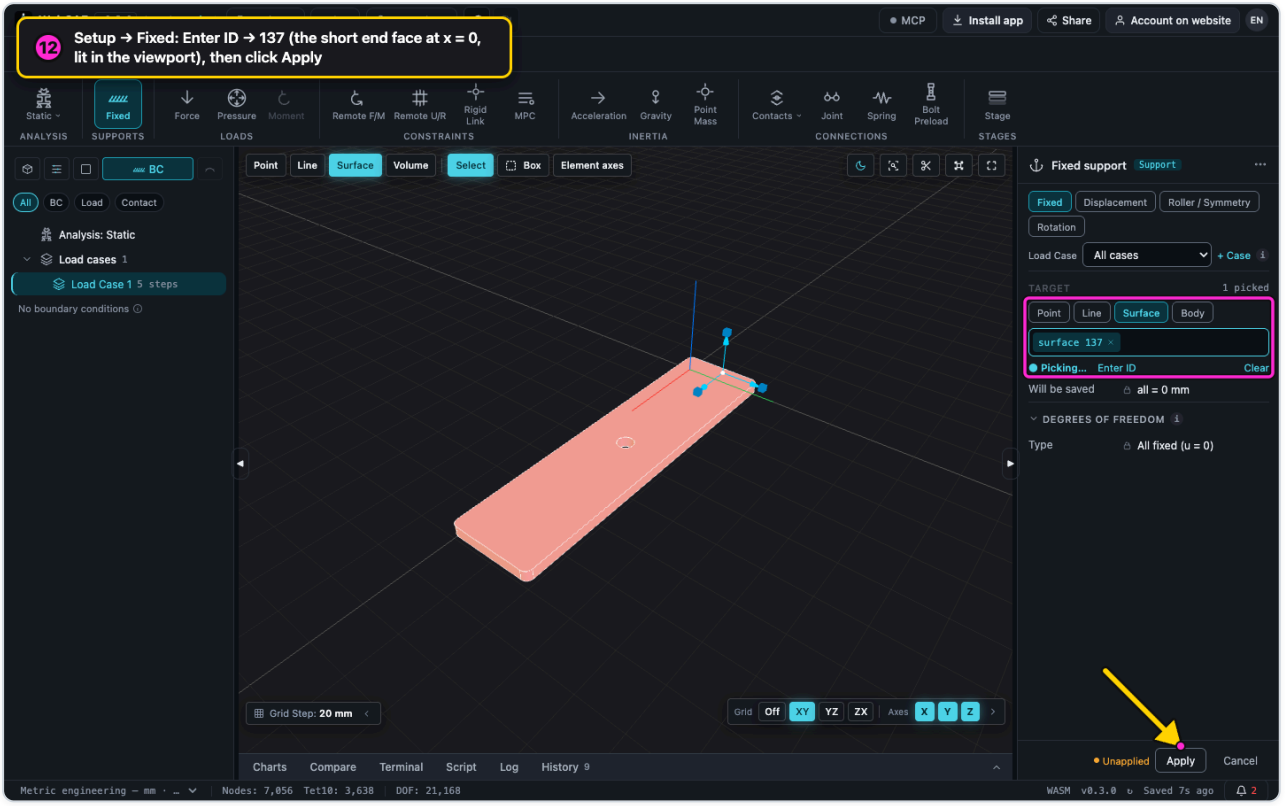
STEP 11 Generate the mesh

Click **Generate** in the Mesh ribbon, pick order **P2**, type **Element size** `5` mm, click **Generate**. The status bar reports **Tet10: 3,638** and the tree shows an *Entity size 1 mm* entry for face 147; the mesh is visibly dense around the hole.



STEP 12 Clamp one end

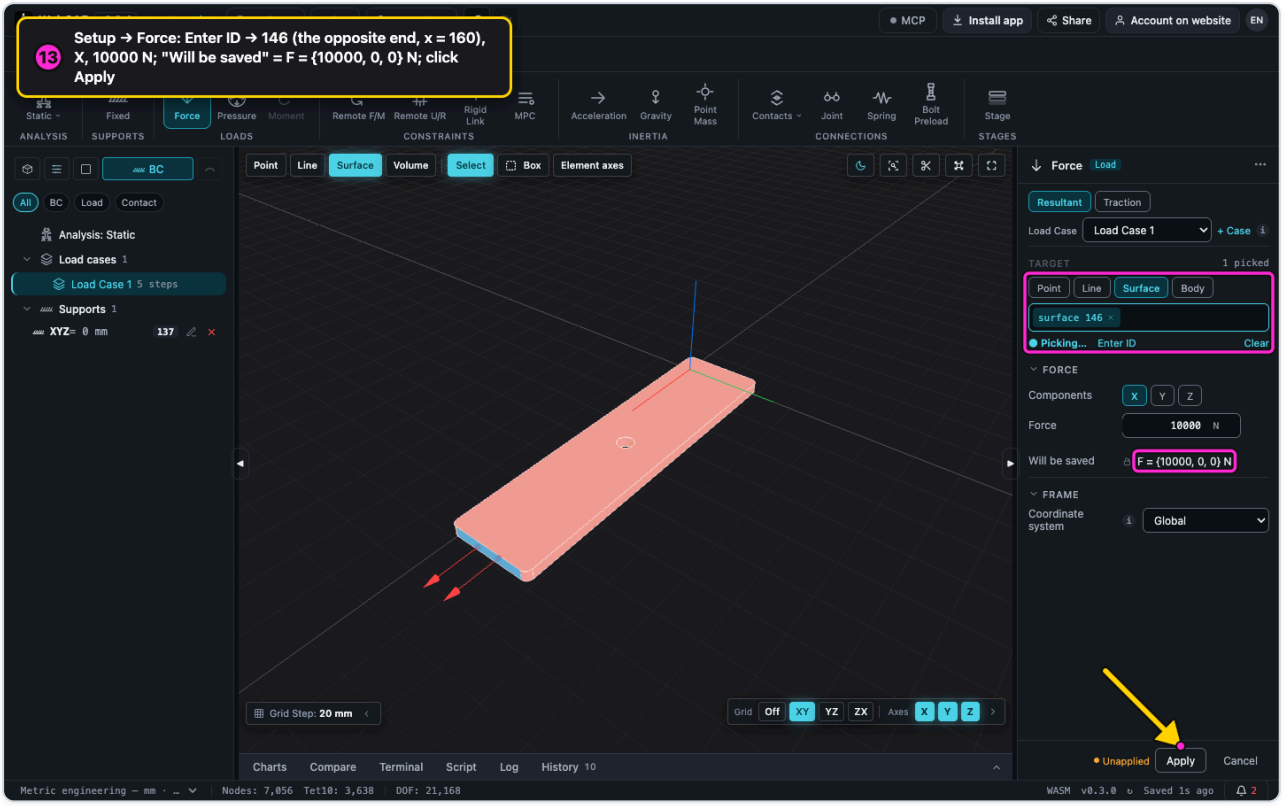
Open **Setup**, click **Fixed**, click **Enter ID**, type `137`, press **Enter**. Face 137 is the short end face at x = 0 (the far end in our view): it lights up and the blue clamp symbols appear on it. Click **Apply**.



STEP 13

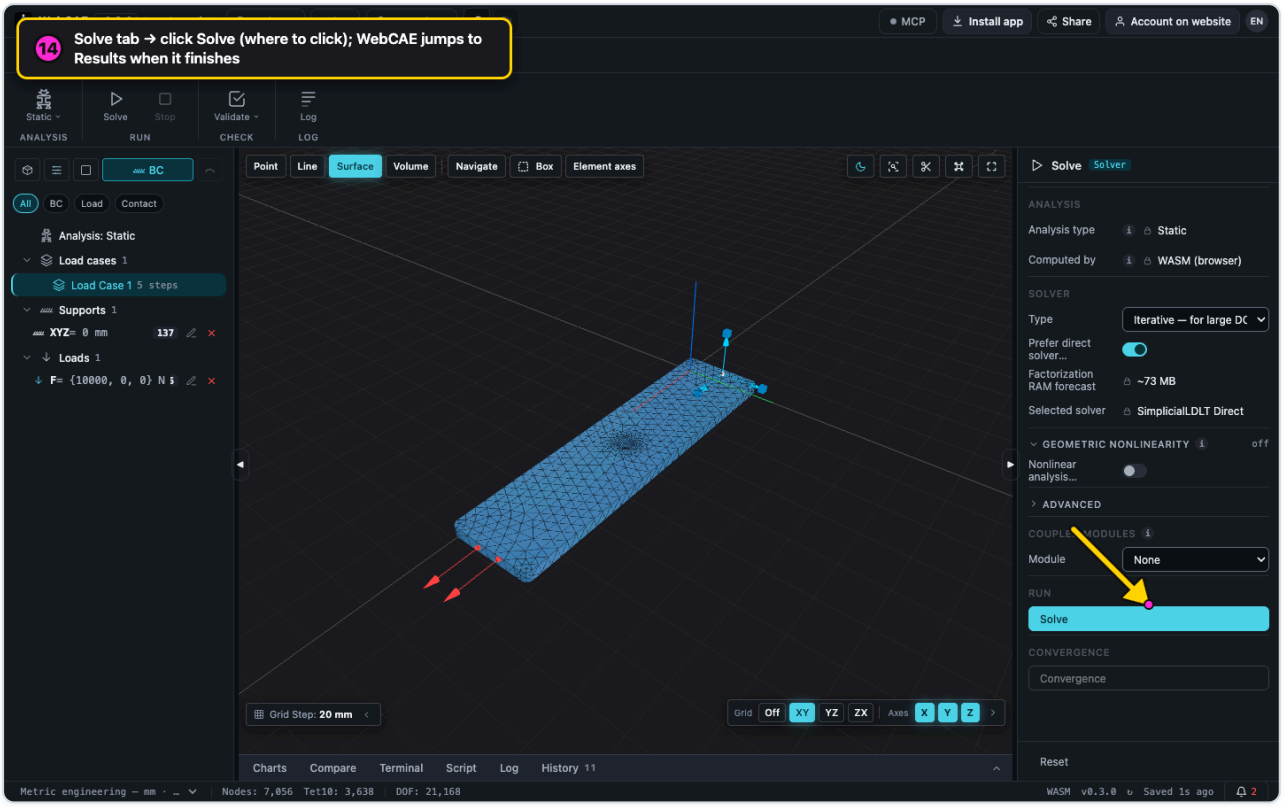
Pull the other end with 10 kN

Click **Force**, **Enter ID**, type `146` (the opposite end face, x = 160, near us), press **Enter**. Choose **X**, type `10000` N. The line **Will be saved** must read `F = {10000, 0, 0} N`. Click **Apply**.



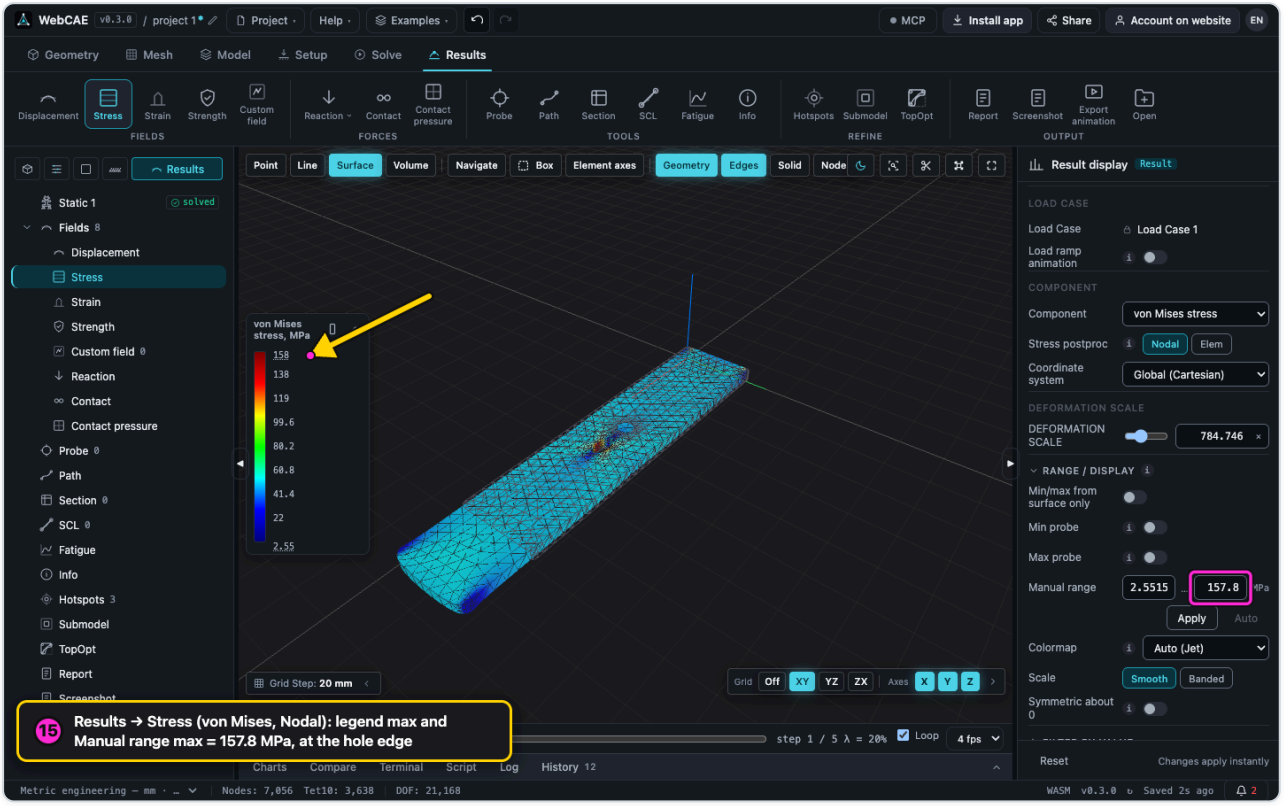
STEP 14 Solve

Open **Solve** and click **Solve**. WebCAE jumps to **Results** when it finishes.



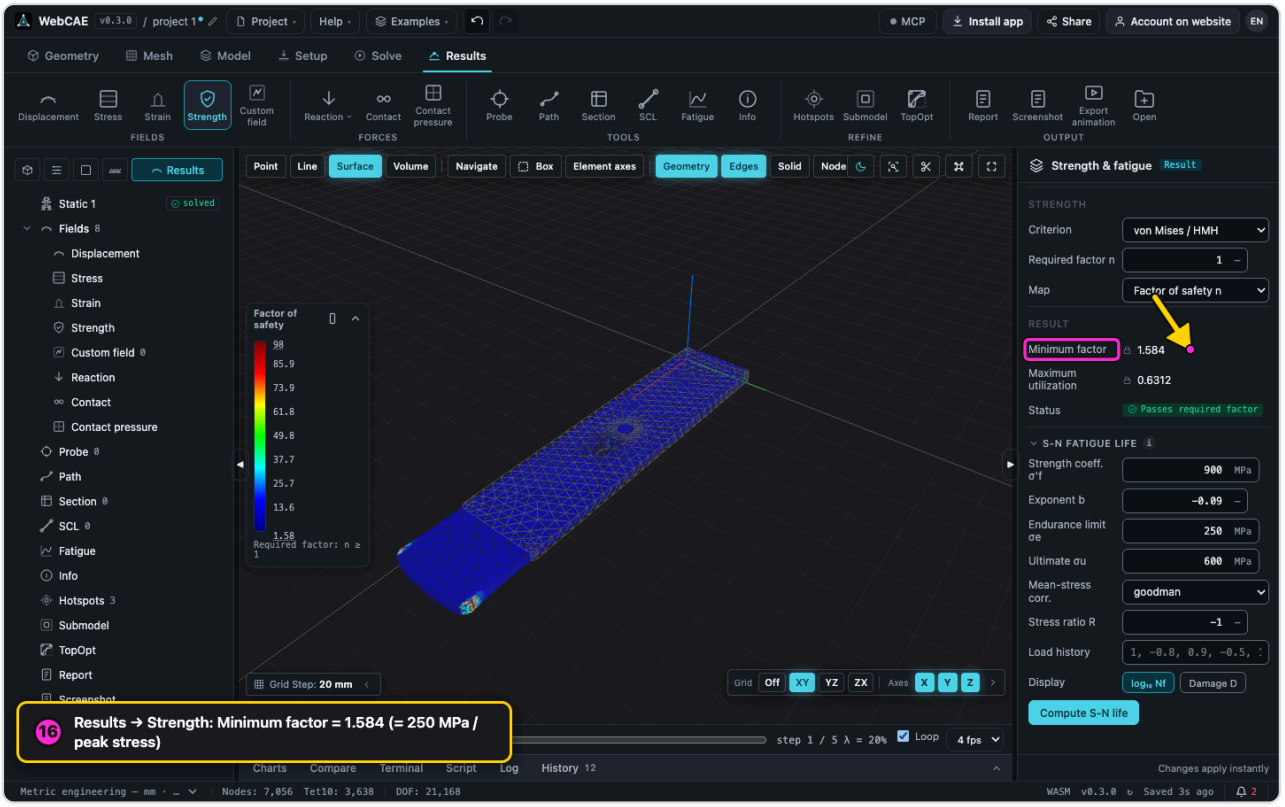
STEP 15 Read the peak stress

In **Results** click **Stress**, component **von Mises stress**, **Nodal**. The legend's top value and the **Manual range** maximum show the peak: **157.8 MPa**, at the hole edge.



STEP 16 Read the safety factor

Click **Strength**. **Minimum factor** reads **1.584**. Check: 250 MPa / 157.8 MPa = 1.584.



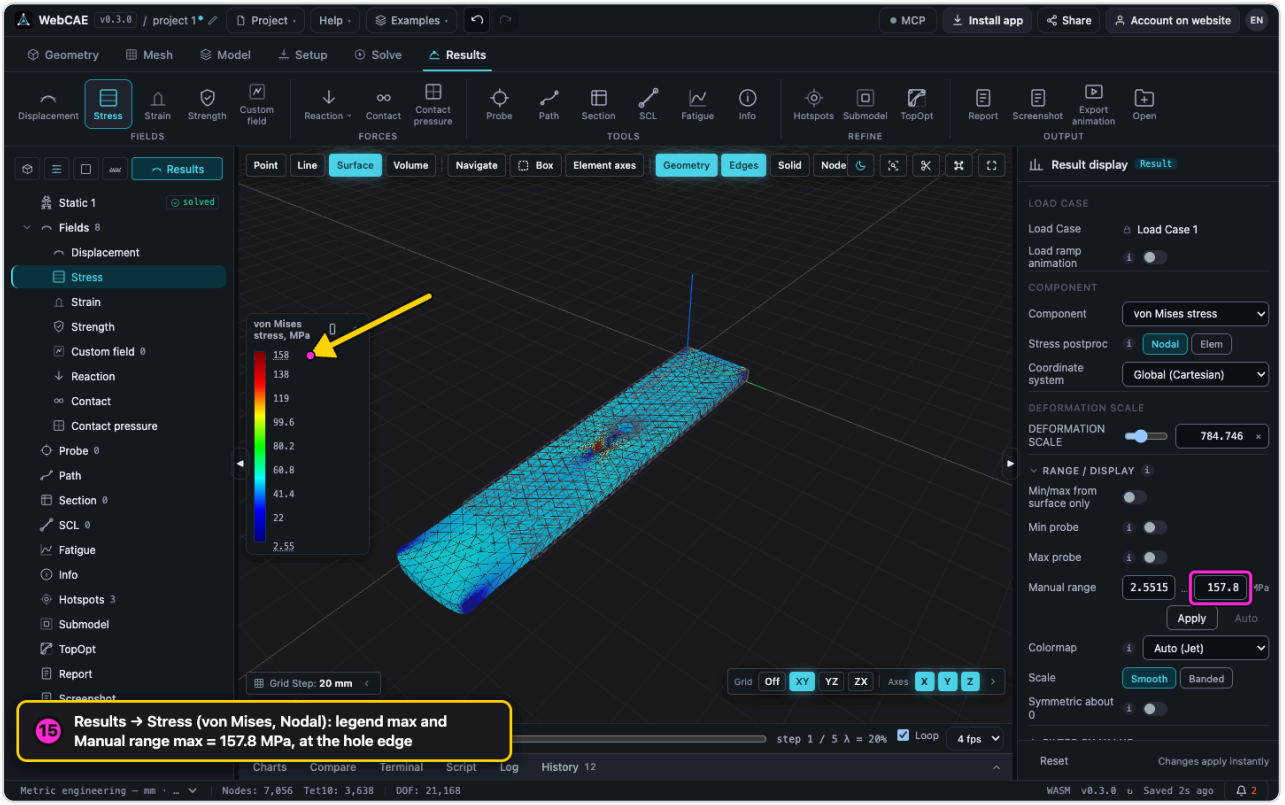
VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Error web / native
Peak von Mises stress $\sigma_{\mathrm{vM,max}}$	$K_{t,\mathrm{net}} \cdot \sigma_{\mathrm{net}} = 2.508 \times 62.5 = 156.80$ MPa	157.8 MPa (Step 15)	157.4 MPa (separate desktop run, not on a frame)	0.64 % / 0.41 %
Minimum safety factor $\mathrm{SF}_{\min}$	$\sigma_y / \sigma_{\max} = 250/156.80 = 1.594$	1.584 (Step 16)	1.588 (same desktop run)	0.65 % / 0.41 %

FORMULA

Net section stress $\sigma_{\mathrm{net}} = \frac{P}{(W-d)t} = \frac{10000}{(40-8)\times 5} = 62.5\ \mathrm{MPa}$. With $d/W = 8/40 = 0.20$, Peterson's concentration factor is $K_{t,\mathrm{net}} = 3.00 - 3.13(0.2) + 3.66(0.2)^2 - 1.53(0.2)^3 = 2.508$, giving $\sigma_{\max} = 156.8\ \mathrm{MPa}$ and $\mathrm{SF} = 250/156.8 = 1.594$. Applicability: this is the Peterson chart for a finite-width plate with a central circular hole under uniform axial tension (Pilkey, Peterson's Stress Concentration Factors, Chart 4.1, net-section basis). Here $d/W = 0.20$ is inside the chart range, and the plate is $L/W = 4$ long with the hole at mid-length, so the clamp and load-introduction effects (which die out within about one width, $W = 40$ mm) do not reach the hole. At the hole edge the stress state is uniaxial, so the von Mises peak equals the principal stress and can be compared with the formula directly.



TROUBLESHOOTING & NOTES

Common mistakes

- **Clicking OK instead of Apply in the Boolean panel:** the Boolean panel only has Apply (it performs the cut) and Cancel; Cancel without Apply discards the operation.
- **Forgetting the force axis:** the Force panel opens with no component chosen. Click **X** and check `Will be saved: F = {10000, 0, 0} N`.
- **Typing old face numbers:** numbers change after every geometry operation. Take them from the tree after the last operation, and check on screen that the right face lights up.
- **Reading the stress from the wrong view:** use **Stress** with **Nodal** (as here). The element-wise view averages the elements and shows a lower peak at the hole.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#3 — Meshing: when is the mesh good enough? (F03)**: how the peak stress depends on the mesh, and when to trust a number.

WebCAE 101

LESSON 3 · F03

Meshing: when is the mesh good enough?

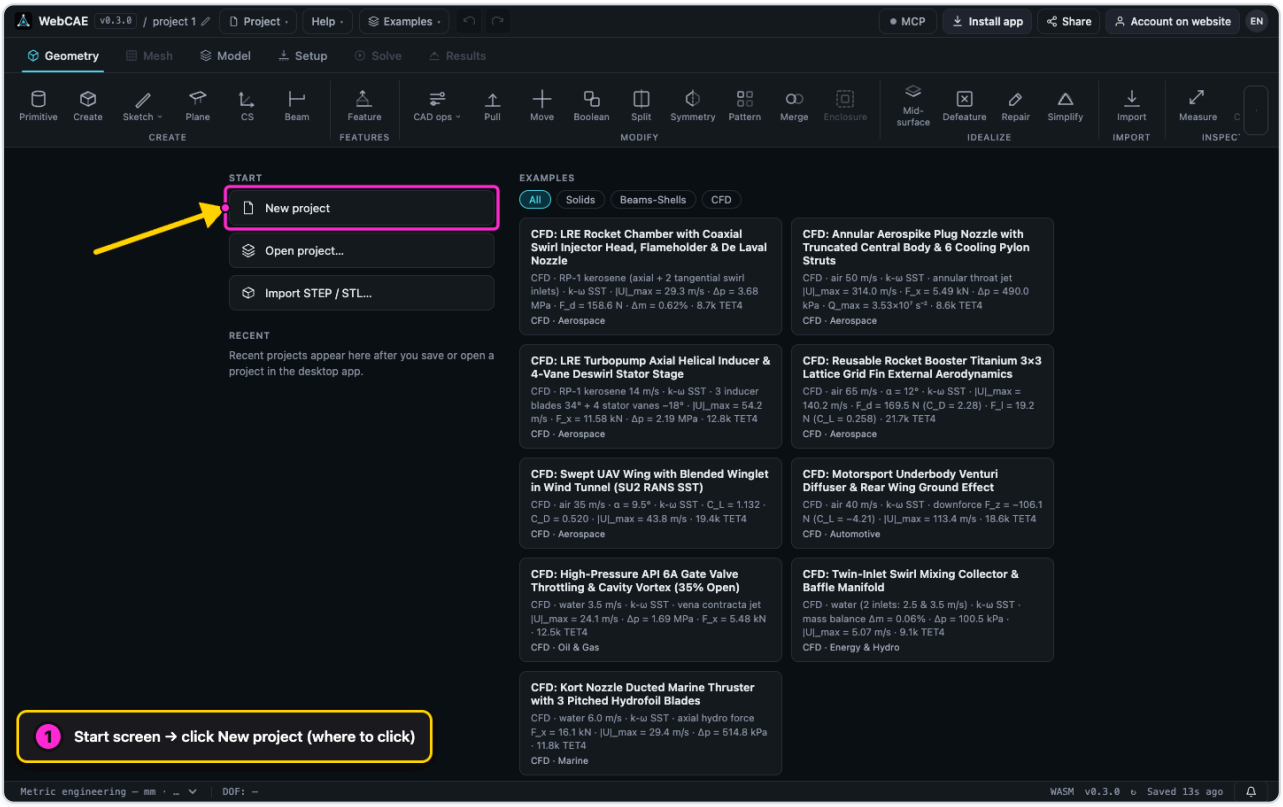
Goal Watch the peak stress at a hole change with the mesh, find out which mesh change matters, and know when to trust the number.

Outcome A convergence table for the same plate-with-hole as lesson 2: uniform meshes (8 mm, 4 mm) under-read the peak by 7–8 %; refining only the hole wall (2 → 1 → 0.5 mm) brings it to 158.7 MPa, within 1.2 % of Peterson's 156.8 MPa, and the last step no longer changes it.

12–14 min | web (WASM) | Run: 2026-11-04-32ffaed2 / F03

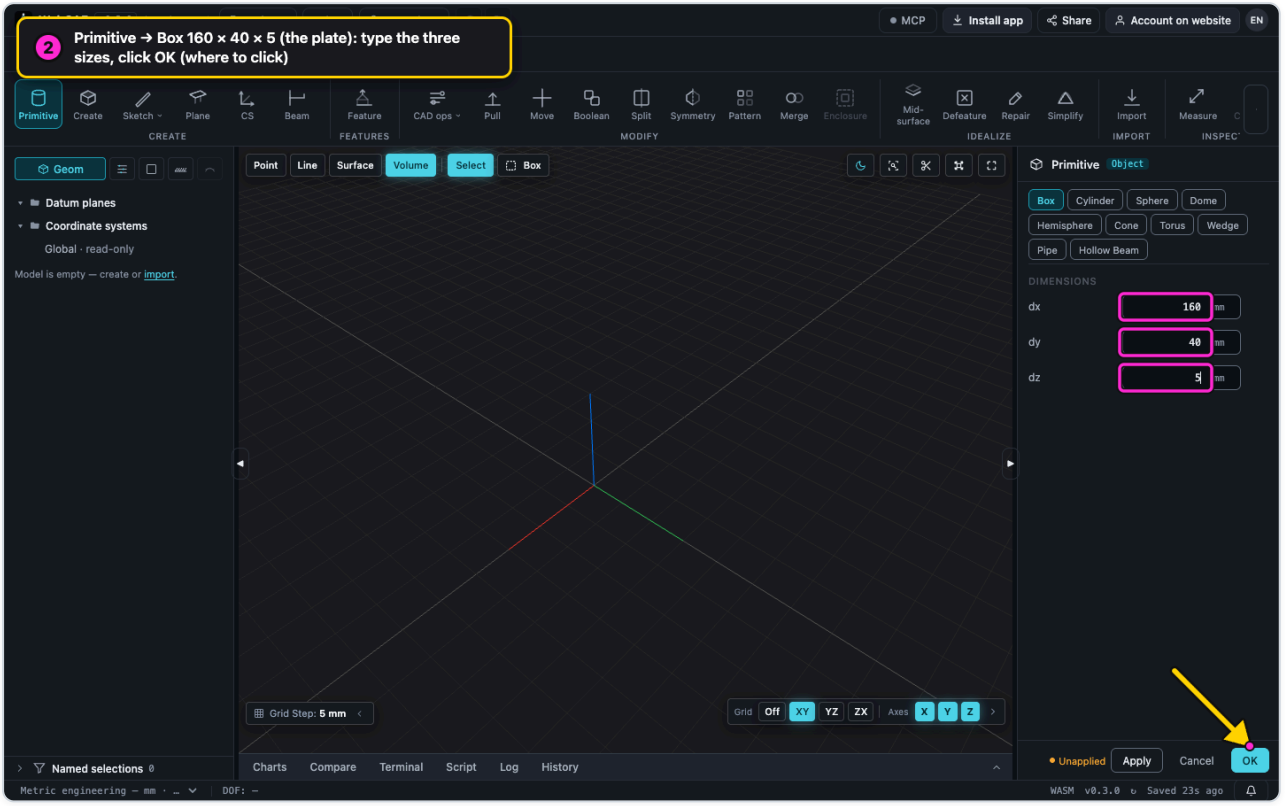
STEP 1 Start a new project

When WebCAE opens, it shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and a gallery of examples). The workspace behind it is not usable until you leave it: click **New project**.



STEP 2 Create the plate

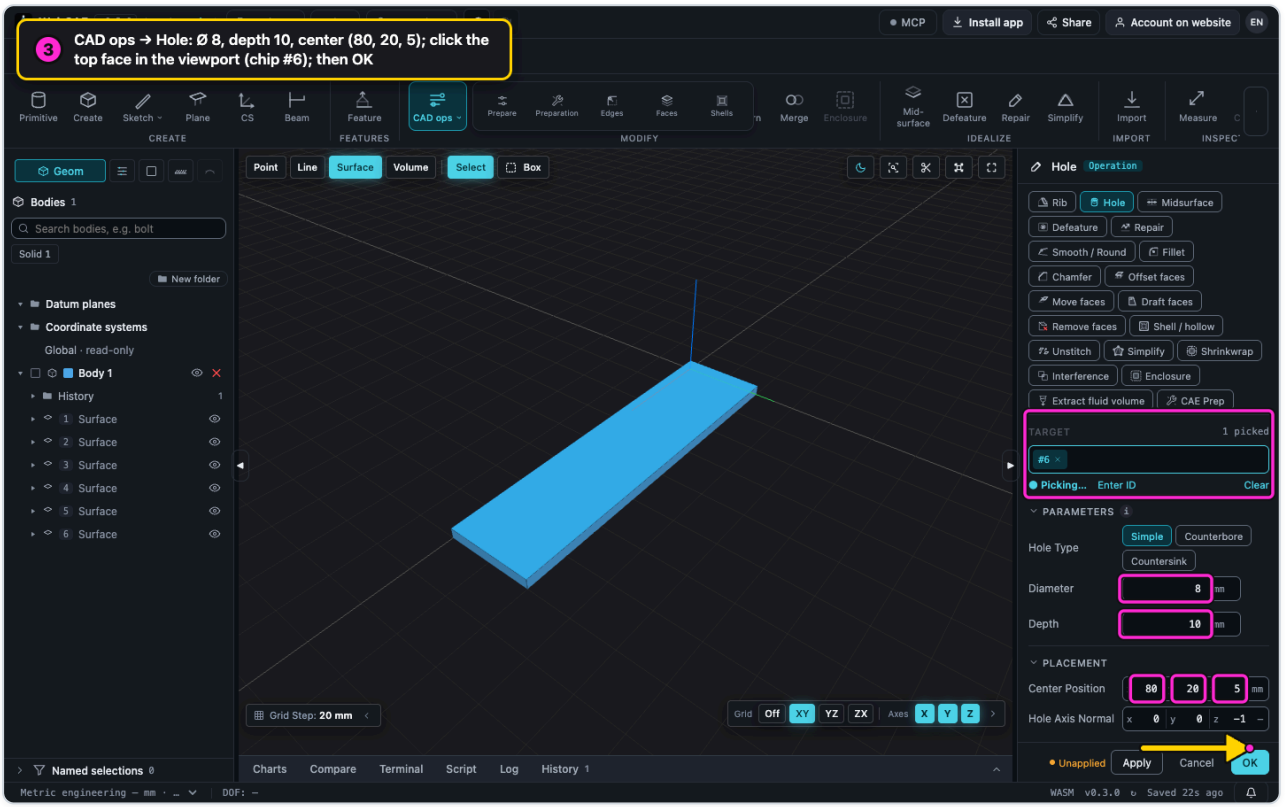
In **Geometry** click **Primitive**, keep **Box**, type `160` in **dx**, `40` in **dy**, `5` in **dz** and click **OK** (OK creates the body and closes the panel).



STEP 3

Make the Ø 8 mm hole

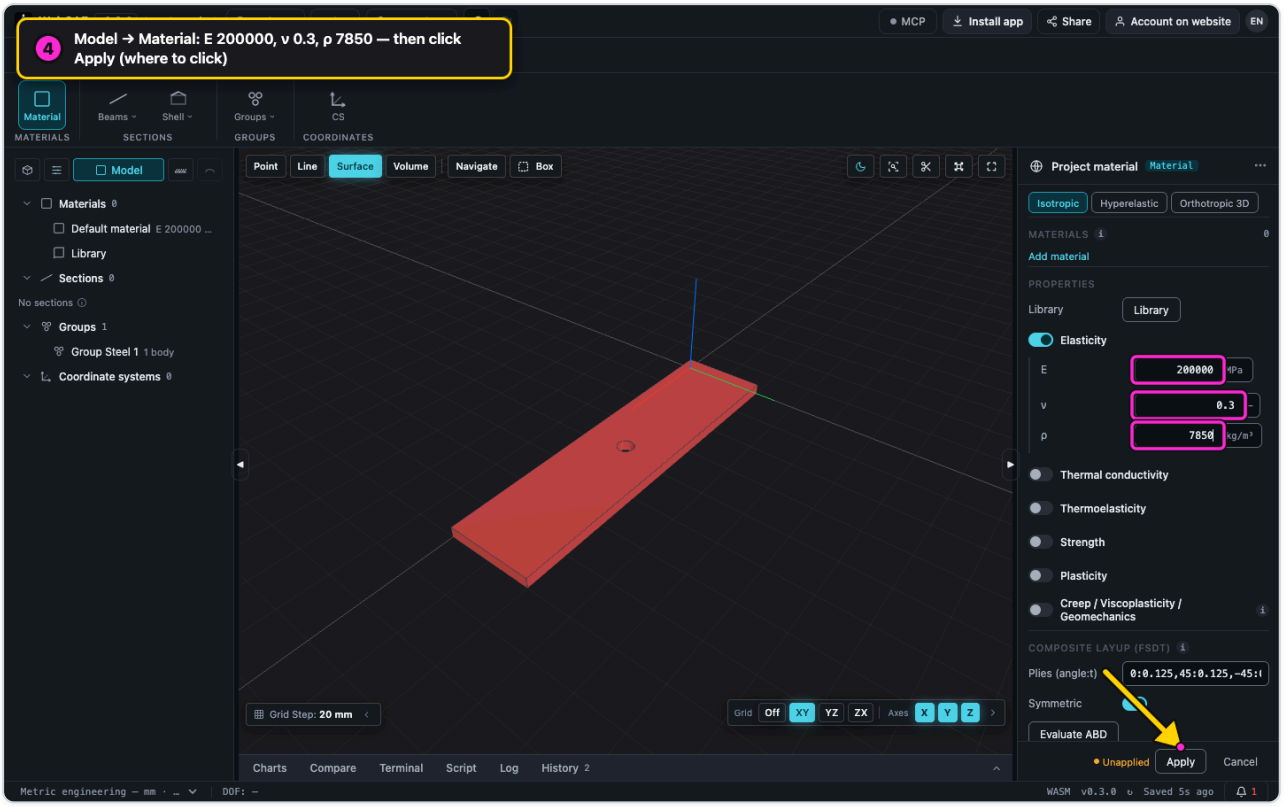
Expand **Body 1** in the tree (click its arrow). Click **CAD ops**, choose **Hole**. Type **Diameter** `8`, **Depth** `10`, **Center Position** x `80`, y `20`, z `5`; **Hole Axis Normal** x `0`, y `0`, z `-1`. Select the **top face**: click the big face on top in the viewport, or (more reliable) click **Enter ID** under **Target**, type `6` and press **Enter**. Either way the face lights up and the chip **#6** appears in Target. Click **OK**.



STEP 4

Material

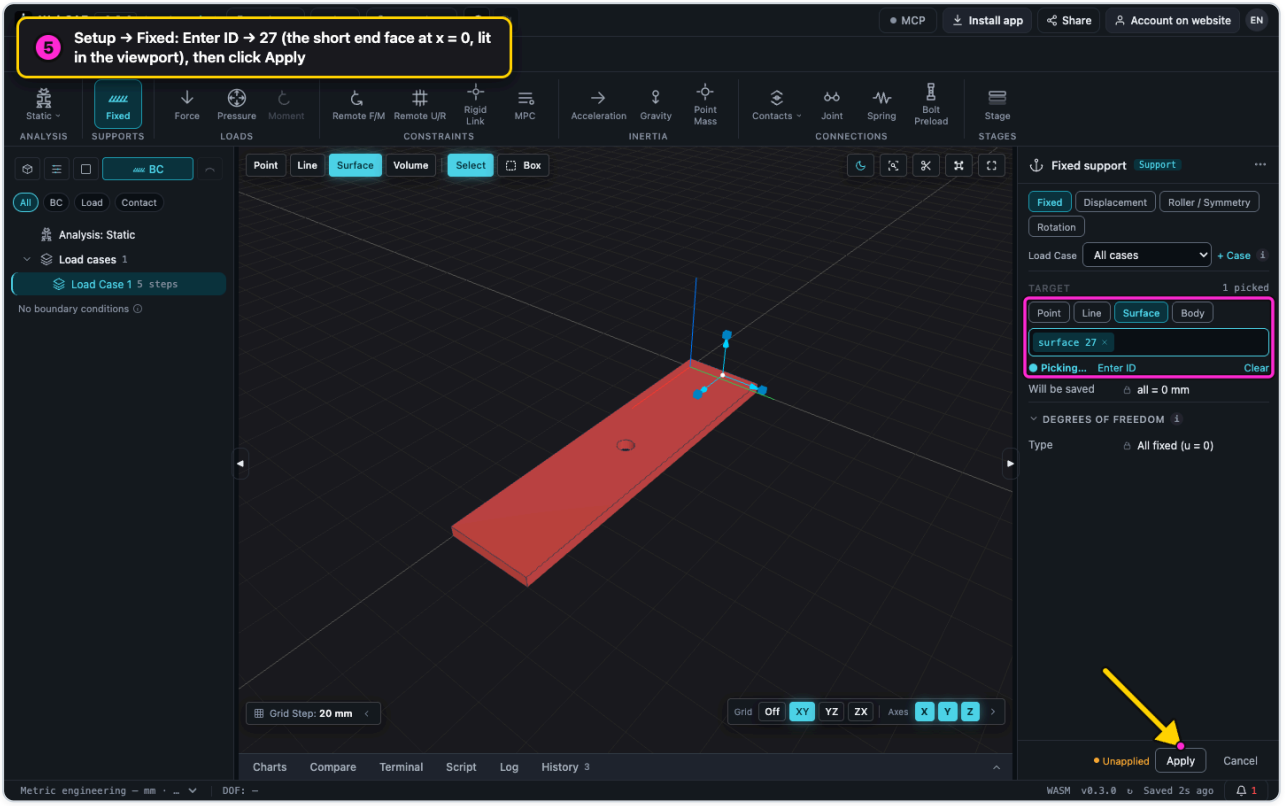
Open **Model**, click **Material**: **E** `200000` MPa, **ν** `0.3`, **ρ** `7850` kg/m³, click **Apply**.



STEP 5

Clamp one end

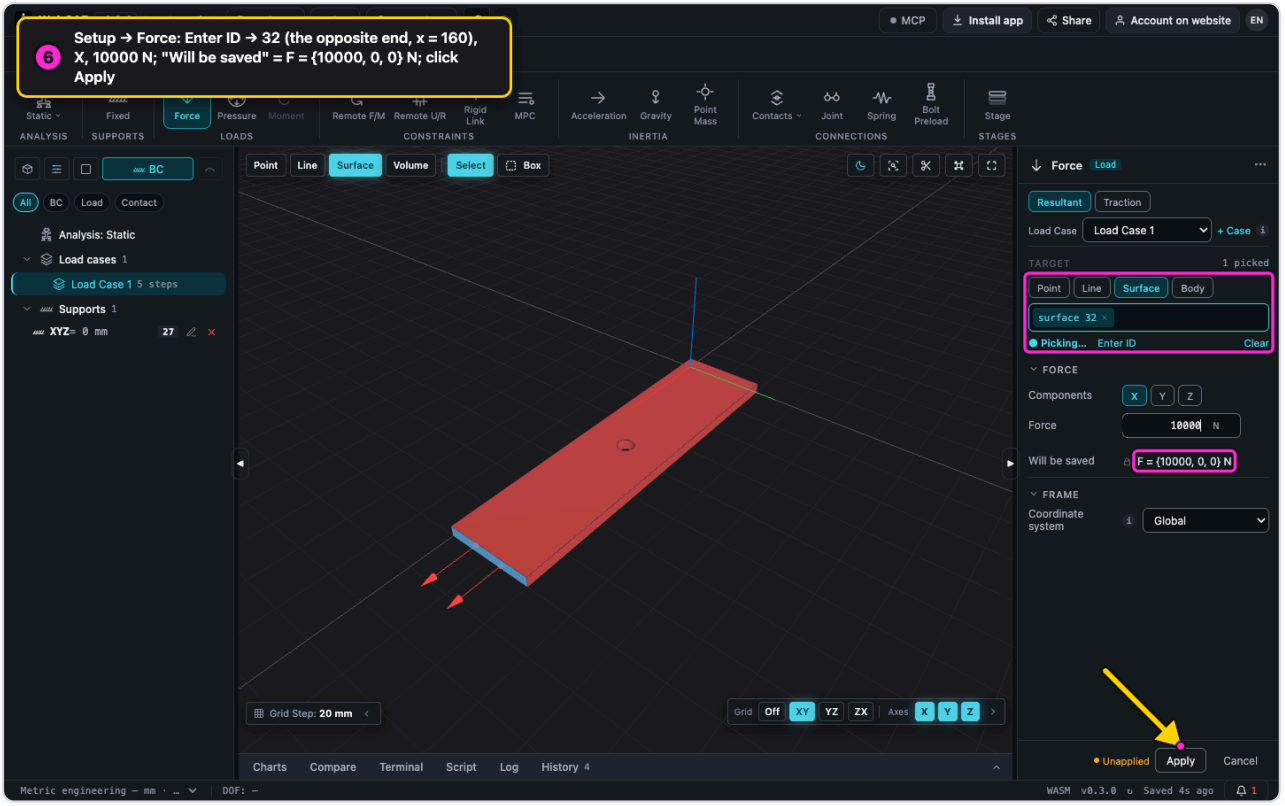
Open **Setup**, click **Fixed**, **Enter ID**, type `27`, press **Enter**: the short end face at x = 0 lights up (the far end in our view). Click **Apply**.



STEP 6

Pull the other end with 10 kN

Click **Force**, **Enter ID**, type `32` (the opposite end, x = 160), press **Enter**. Choose **X**, type `10000` N; **Will be saved** must read `F = {10000, 0, 0} N`. Click **Apply**.

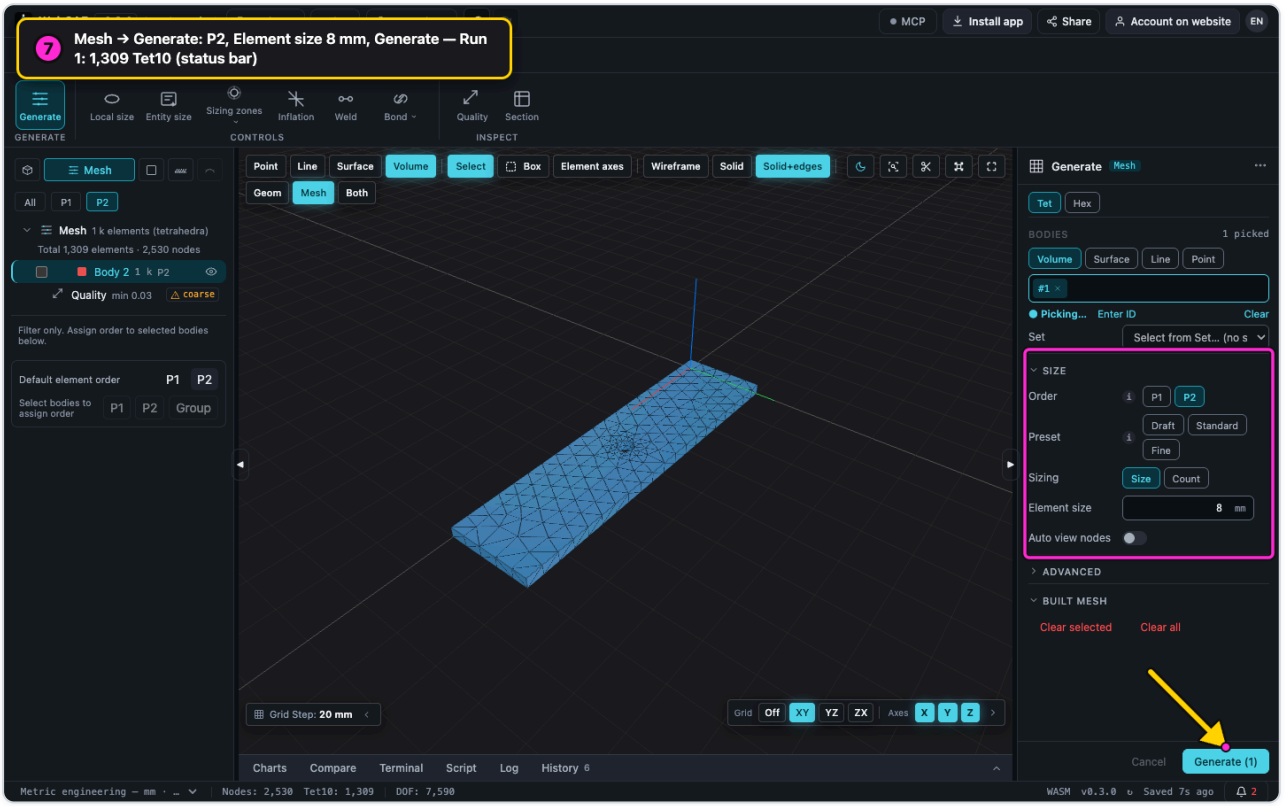


STEP 7

Run 1: a coarse mesh, 8 mm

Open **Mesh**, click the **Generate** tile in the ribbon, choose **P2** (quadratic tetrahedra), type **Element size** 8 mm and click **Generate (1)** at the bottom of the panel (the number in brackets is how many bodies are picked). The status bar reports **Tet10 1,309**.

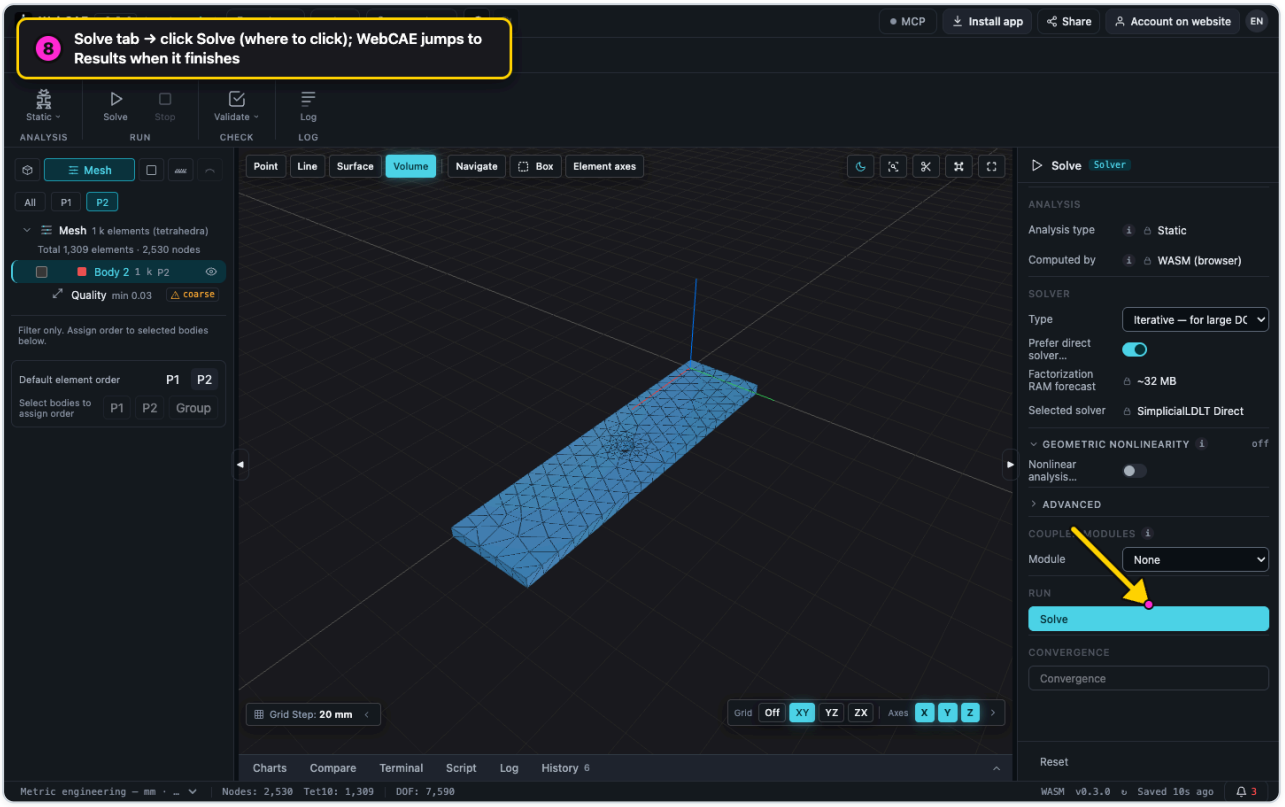
> **WHY** A coarse mesh is the quick first look; the hole edge gets only a few elements.



STEP 8

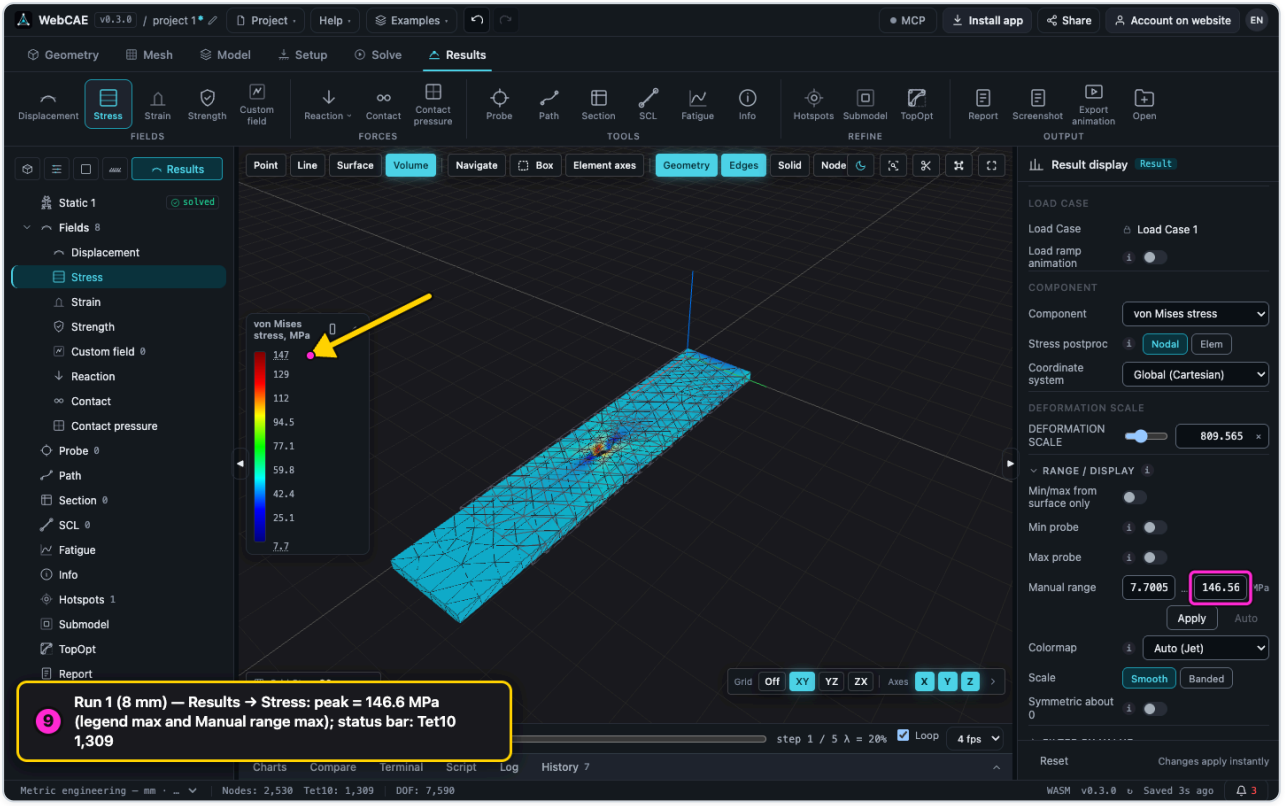
Solve

Open **Solve** and click **Solve**. WebCAE jumps to **Results** when it finishes.



STEP 9 Read the peak of Run 1

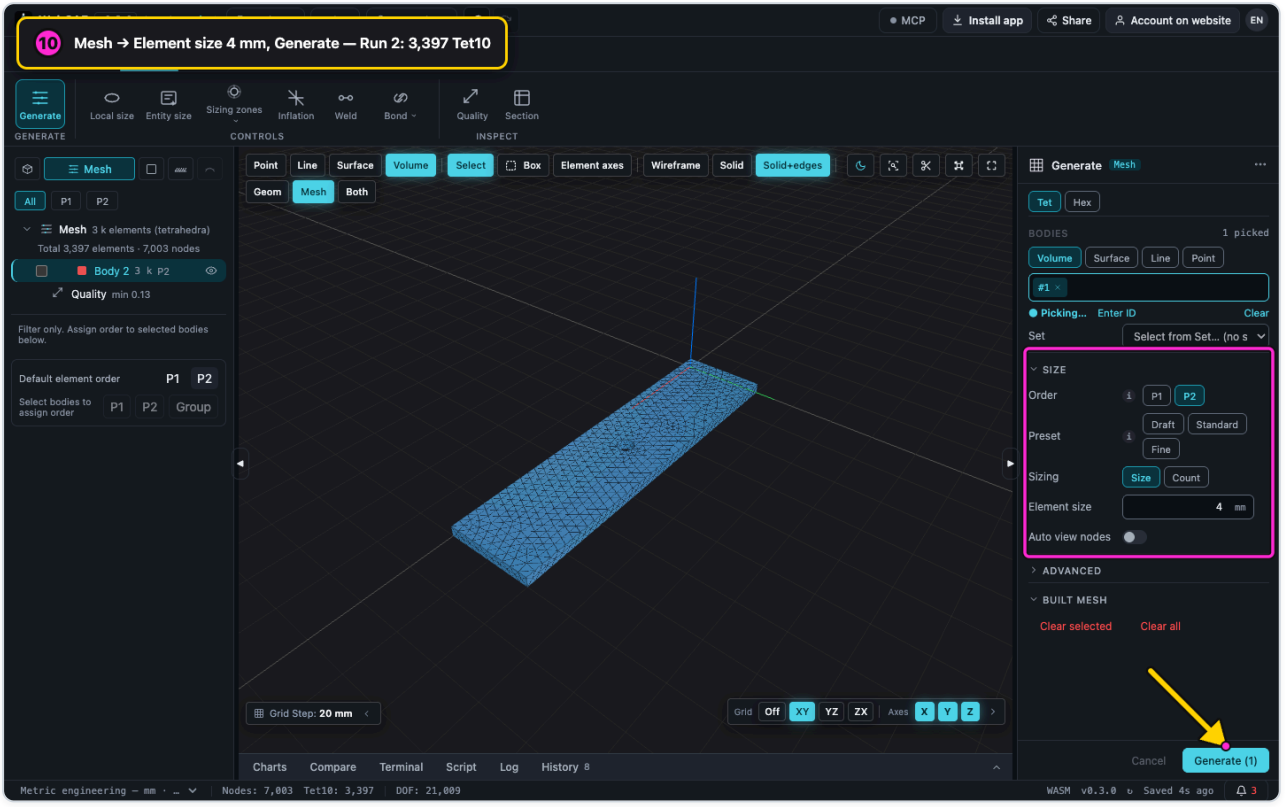
In **Results** click **Stress** (von Mises, Nodal). The legend maximum and the **Manual range** maximum show **146.6 MPa**.



STEP 10

Run 2: halve the element size everywhere, 4 mm

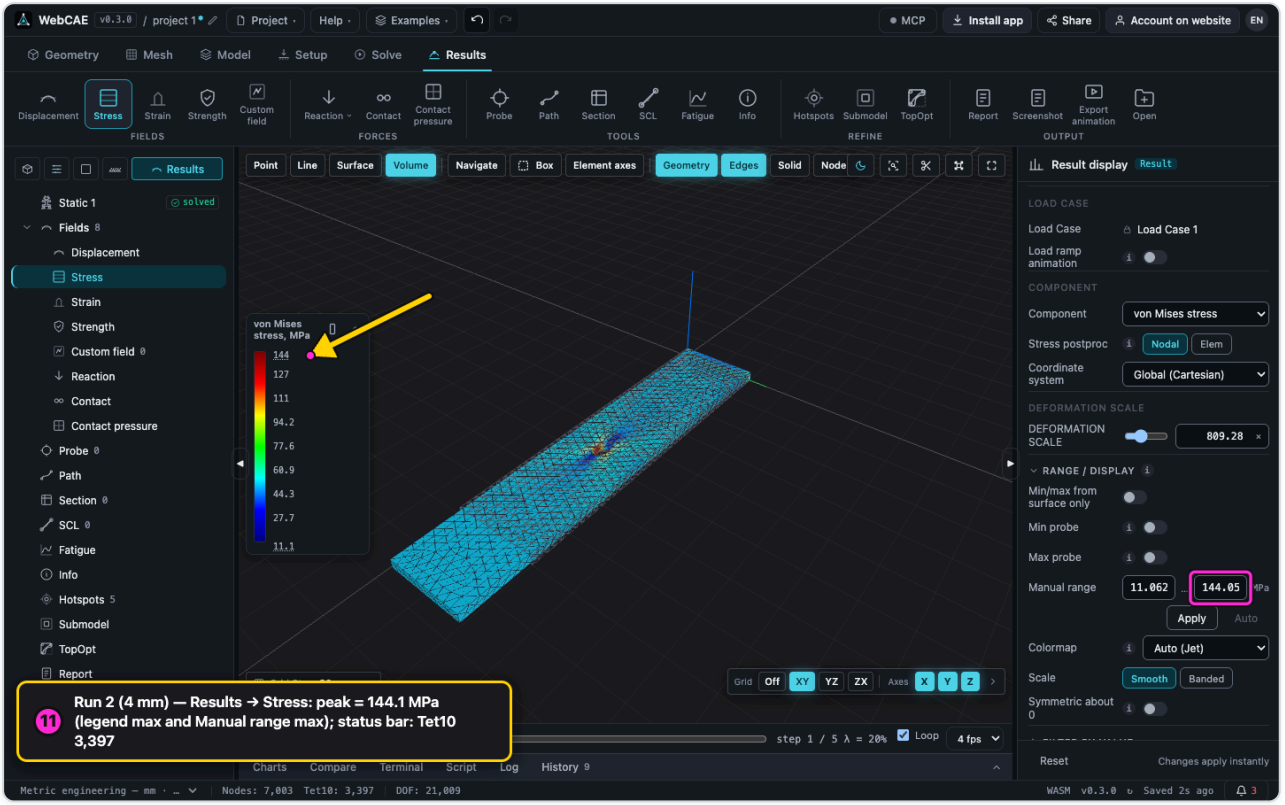
Open **Mesh → Generate**, change **Element size** from its default (16 mm for this plate) to `4` mm and click **Generate (1)**: **Tet10 3,397**, 2.6 times more than Run 1. Solve again (Solve → Solve) and open **Results → Stress**.



STEP 11

Read the peak of Run 2

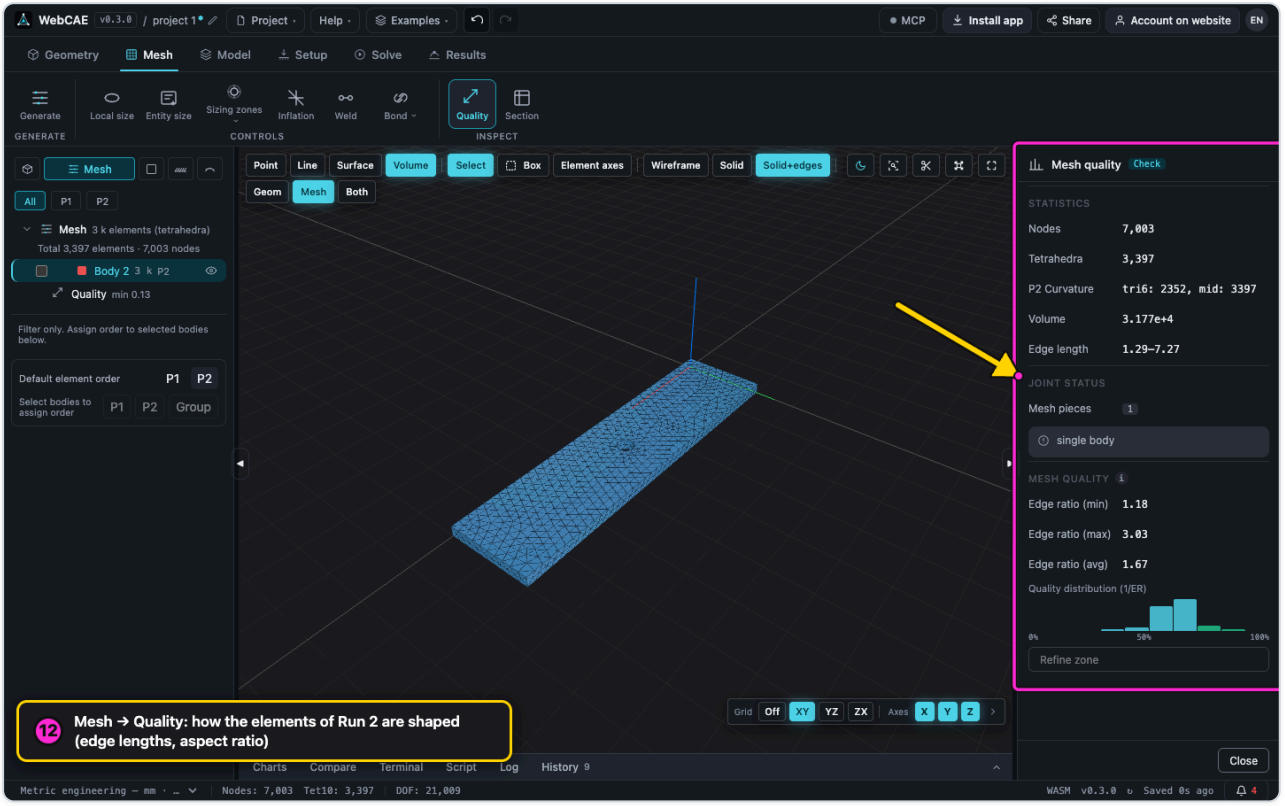
The peak is **144.1 MPa**. Compare with Run 1 (146.6 MPa): the mesh got 2.6 times bigger and the peak did not come closer to the formula (156.8 MPa) — it is still 8 % low. The hole edge is still resolved by too few elements, and a uniform mesh pays for refinement everywhere.



STEP 12

Look at the element quality

Open **Mesh** and click **Quality**. The panel summarises the shape of the elements of the current mesh. Use it to check that a mesh with small elements at the hole does not get badly shaped ones.

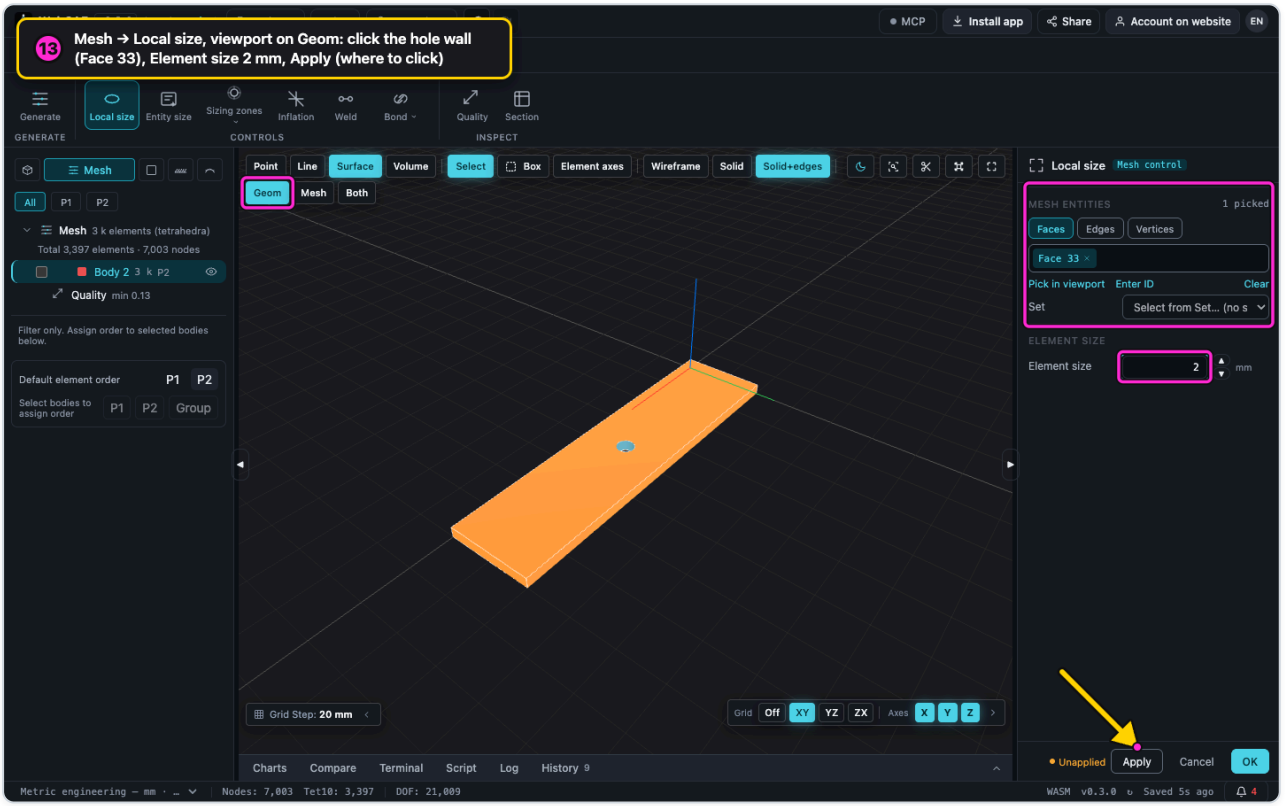


STEP 13

Refine only the hole wall (local size 2 mm)

Open **Mesh** and click **Local size**. Local size picks faces in the viewport, and after a mesh is built the viewport shows the mesh — so first click **Geom** (second row of buttons at the top of the viewport) to show the geometry again. Keep **Faces** selected and click inside the ring of the hole: the wall of the hole lights up and **Face 33** appears in the list (or, more reliably, click **Enter ID** and type `33`). Type **Element size** `2` mm and click **Apply**.

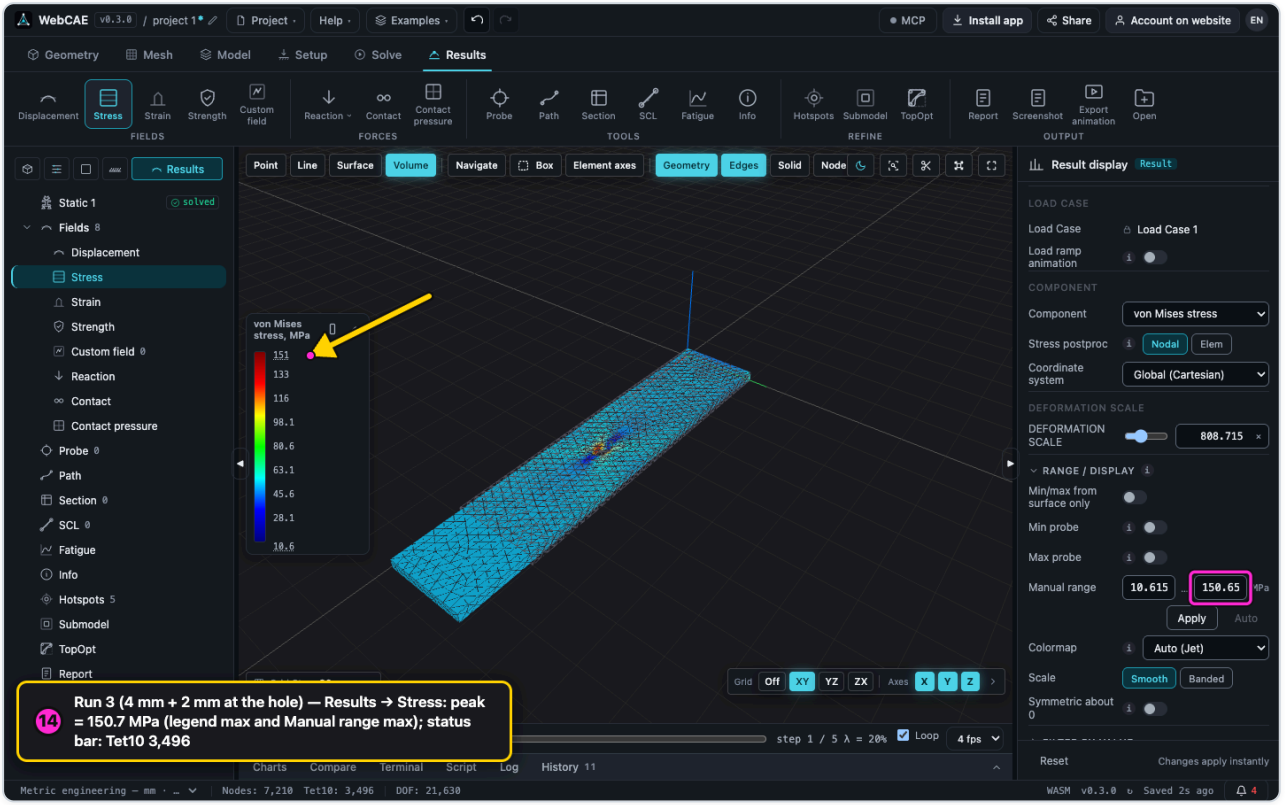
> **WHY** The stress is steep right at the hole edge; small elements only there cost few elements, while the rest of the plate stays at 4 mm.



STEP 14

Run 3: 2 mm at the hole

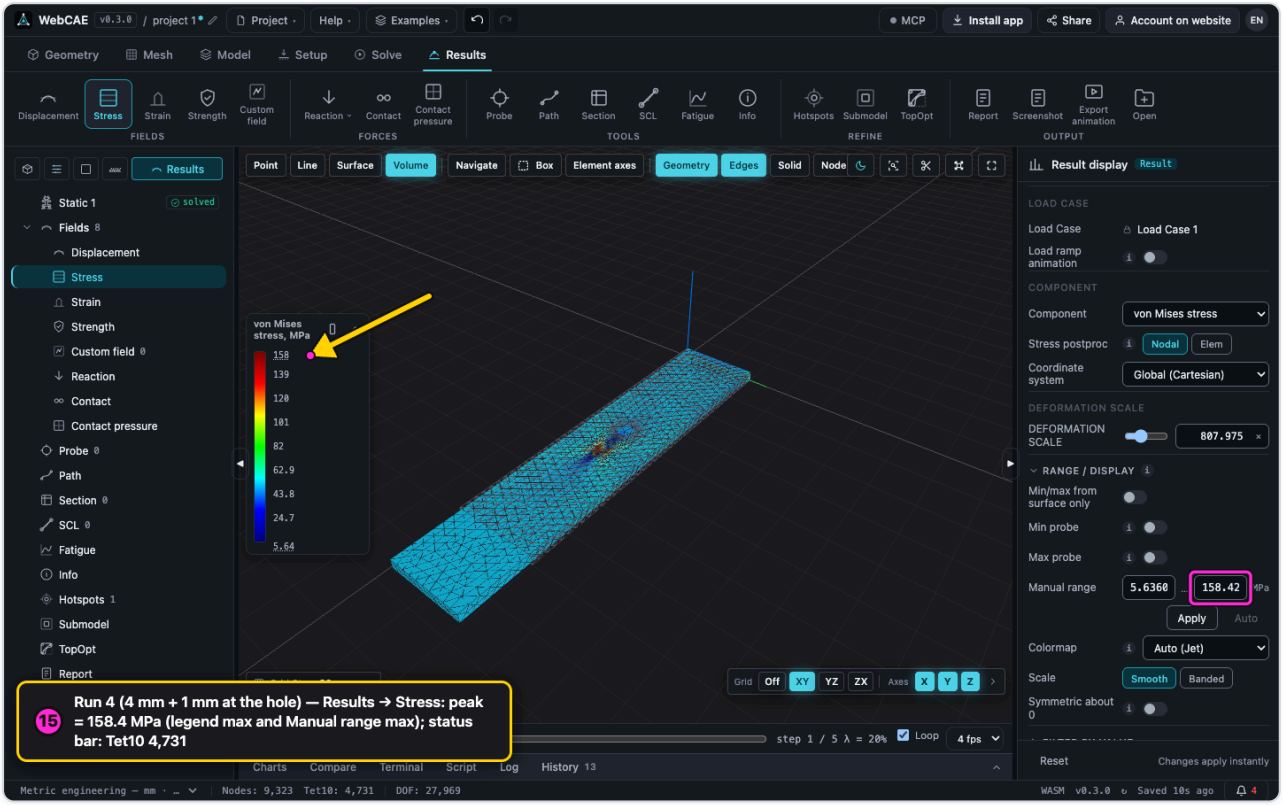
Open **Mesh → Generate**. The form keeps **P2**, but **Element size** is back at its default (16 mm for this plate) every time you reopen it, so type `4` again, click **Generate (1)**, then **Solve**, open **Results → Stress**. The peak (legend and **Manual range**) is **150.7 MPa**, the status bar shows **Tet10 3,496**. Up from 144.1 MPa, but still 3.9 % below the formula: 2 mm is too coarse at the edge.



STEP 15

Run 4: 1 mm at the hole

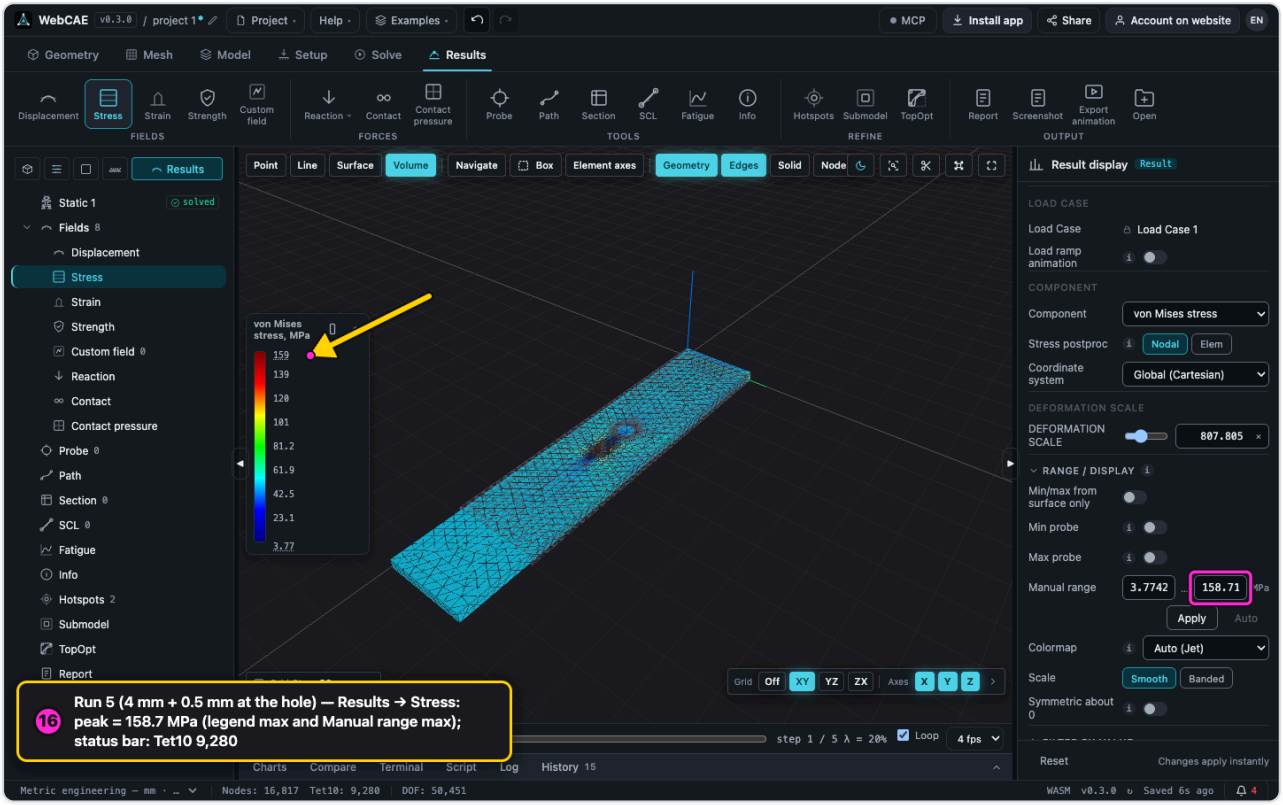
Open **Mesh → Local size**, pick the hole wall again (click it, or **Enter ID** → `33`), change **Element size** to `1` mm and click **Apply** (it replaces the previous 2 mm entry; the tree still shows one *Entity size 1* line — change it with the **Local size** tile, not from that tree line). Then open **Mesh → Generate**, check that **P2** is still selected, type `4` again in **Element size** (it is back at 16 mm) and click **Generate (1)**, then **Solve**, **Results → Stress**. The peak (legend and **Manual range**) is **158.4 MPa**, the status bar shows **Tet10 4,731**. A jump of 7.8 MPa for 1.4 times the elements; now 1.0 % from the formula.



STEP 16

Run 5: 0.5 mm at the hole

Open **Mesh → Local size**, pick the hole wall again (click it, or **Enter ID** → `33`), change **Element size** to `0.5` mm and click **Apply** (it replaces the previous 1 mm entry; the tree still shows one *Entity size 1* line — change it with the **Local size** tile, not from that tree line). Then open **Mesh → Generate**, check that **P2** is still selected, type `4` again in **Element size** (it is back at 16 mm) and click **Generate (1)**, then **Solve**, **Results → Stress**. The peak (legend and **Manual range**) is **158.7 MPa**, the status bar shows **Tet10 9,280**. It moved by only 0.2 % while the element count doubled (4,731 → 9,280): the peak has stopped depending on the mesh.



VERIFICATION BENCHMARK

Check against the formula

Run	Global size	Local size (hole wall)	Tet10, web	Peak σ vM, web (browser)	Error web	WebCAE desktop (native)
Run 1	8 mm	none	1309	146.6 MPa	6.5 %	140.2 MPa (985 tets), 10.6 %
Run 2	4 mm	none	3397	144.1 MPa	8.1 %	138.1 MPa (3400 tets), 11.9 %
Run 3	4 mm	2 mm (hole wall)	3496	150.7 MPa	3.9 %	—
Run 4	4 mm	1 mm (hole wall)	4731	158.4 MPa	1.0 %	156.4 MPa (4606 tets), 0.3 %
Run 5	4 mm	0.5 mm (hole wall)	9280	158.7 MPa	1.2 %	—

The desktop column comes from a separate scripted run of the same model with the native mesher and solver (not on a frame): its meshes differ from the browser's (different element counts), so only the trend is comparable — uniform meshes under-read by about 11–12 %, the 1 mm local refinement lands at 156.4 MPa (–0.3 %).

FORMULA

Net cross-section stress:

$$\sigma_{\text{net}} = \frac{F}{(w-d)\cdot t} = \frac{10\,000\ \text{N}}{(40-8)\ \text{mm}\times 5\ \text{mm}} = 62.5\ \text{MPa}$$

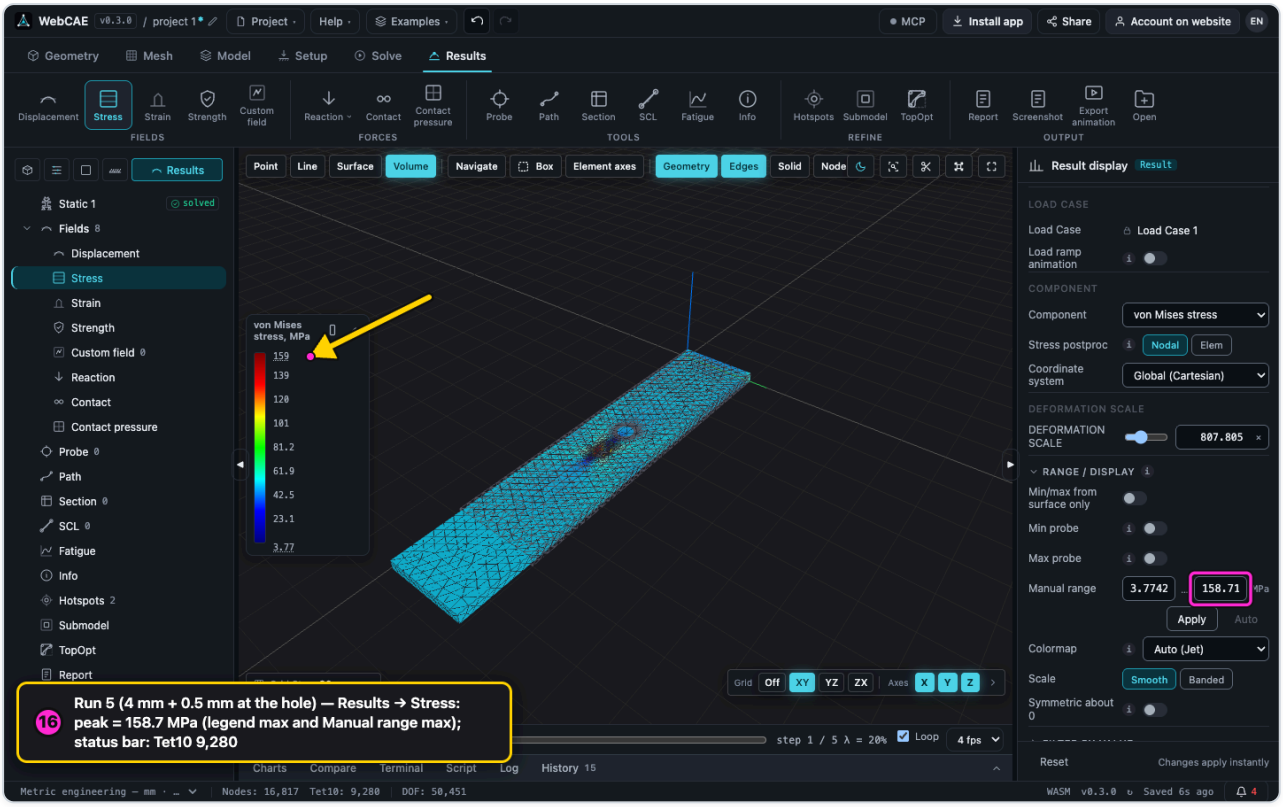
Stress concentration factor for $d/w = 8/40 = 0.20$:

$$K_{t,\text{net}} = 3.00 - 3.13(d/w) + 3.66(d/w)^2 - 1.53(d/w)^3 = 2.508$$

Peak theoretical stress:

$$\sigma_{\max} = K_{t,\text{net}}\cdot\sigma_{\text{net}} = 2.508\times 62.5\ \text{MPa} = 156.8\ \text{MPa}$$

Applicability: Peterson's chart for a finite-width plate with a central circular hole under uniform axial tension (Pilkey, Peterson's Stress Concentration Factors, Chart 4.1, net-section basis); $d/w = 0.20$ is inside its range and the hole sits 80 mm from each end (two plate widths), so clamp effects do not reach it. K_t is itself a curve fit, so do not read more into a 1.2 % difference than "the same number". What the lesson shows is the **trend**: uniform refinement barely moved the peak (146.6 → 144.1 MPa), local refinement at the hole took it to 158.4 and 158.7 MPa, and the last doubling of the elements changed it by 0.2 %.



TROUBLESHOOTING & NOTES

Common mistakes

- **Judging the mesh by the number of elements:** Run 2 has 2.6 times the elements of Run 1 and about the same peak; Run 4 has 1.4 times the elements of Run 2 and a very different one. What counts is the element size where the stress changes fast.
- **Stopping at one run:** a single number tells nothing about convergence. Refine until two successive runs agree (here Runs 4 and 5).
- **Picking the hole wall in the mesh view:** the viewport shows the mesh after Generate; click **Geom** first, or use **Enter ID** and type the face number.
- **Reading the stress from a different view:** use **Stress** with **Nodal** in every run, otherwise the runs are not comparable.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#4 — Natural frequencies in WebCAE (F04)**: modal analysis of a cantilever.

WebCAE 101

LESSON 4 · F04

Natural frequencies in WebCAE

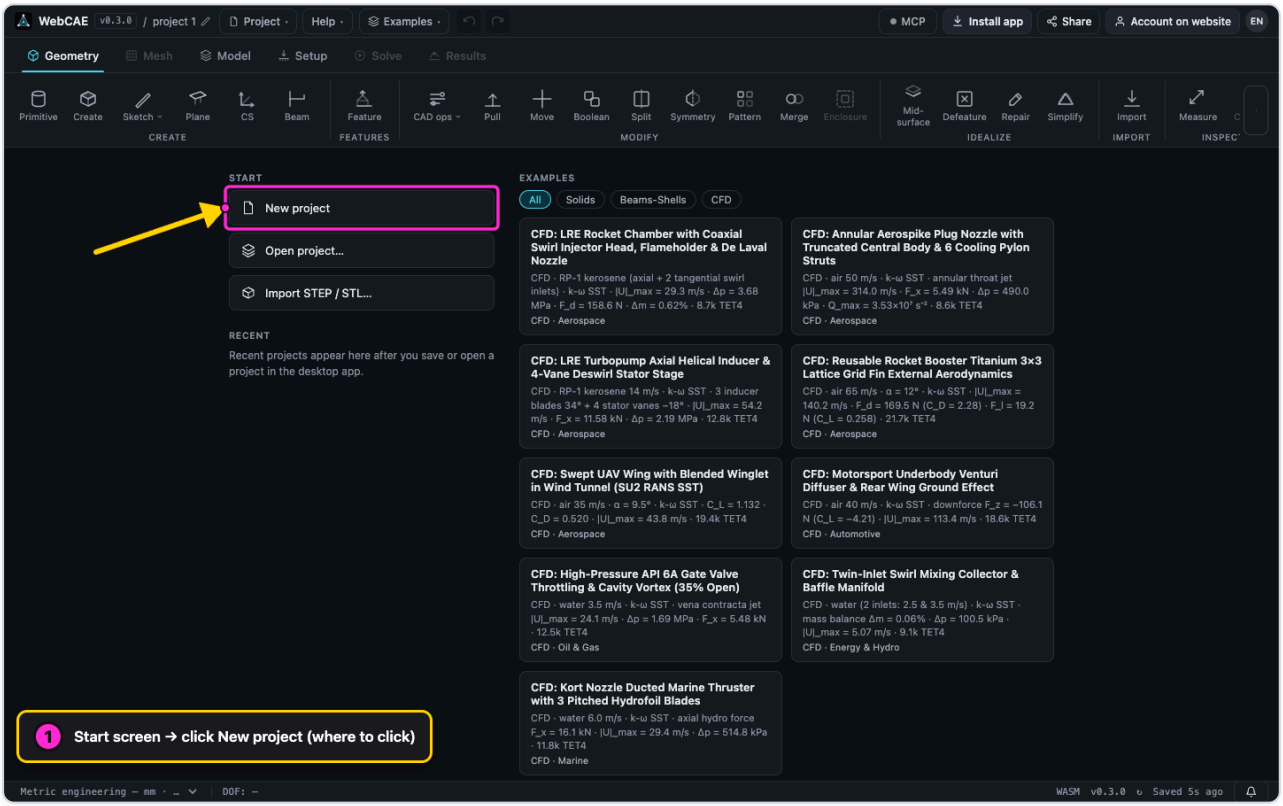
Goal Find the first bending frequencies of a steel cantilever bar and compare them with the textbook Euler–Bernoulli formula.

Outcome The first natural frequency of the bar: 203.85 Hz by formula, 204.23 Hz in WebCAE web (0.18 %) and 204.13 Hz in WebCAE desktop. Modes 1 and 2 form a degenerate pair because the section is square (ratio 1.0005).

7–8 min | web (WASM) | Run: 2026-11-04-32ffaed2 / F04

STEP 1 Start a new project

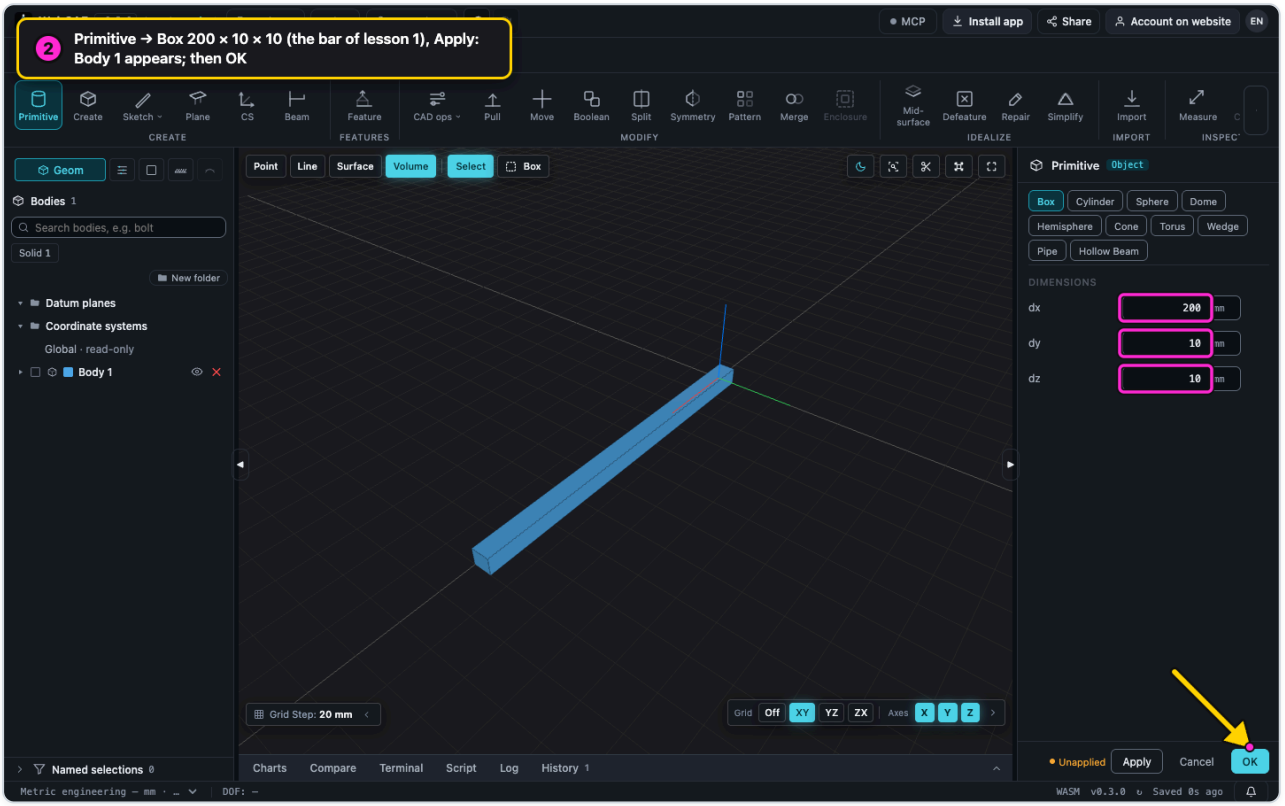
When WebCAE opens, it shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and a gallery of examples). The workspace behind it is not usable until you leave it: click **New project**.



STEP 2 Create the 200 × 10 × 10 mm steel bar

In **Geometry** click **Primitive**, keep **Box**, type `200` in **dx**, `10` in **dy**, `10` in **dz**, click **Apply** (Body 1 appears), then **OK**.

> **WHY** The same bar as lesson 1 — the only new thing in this lesson is the analysis type.

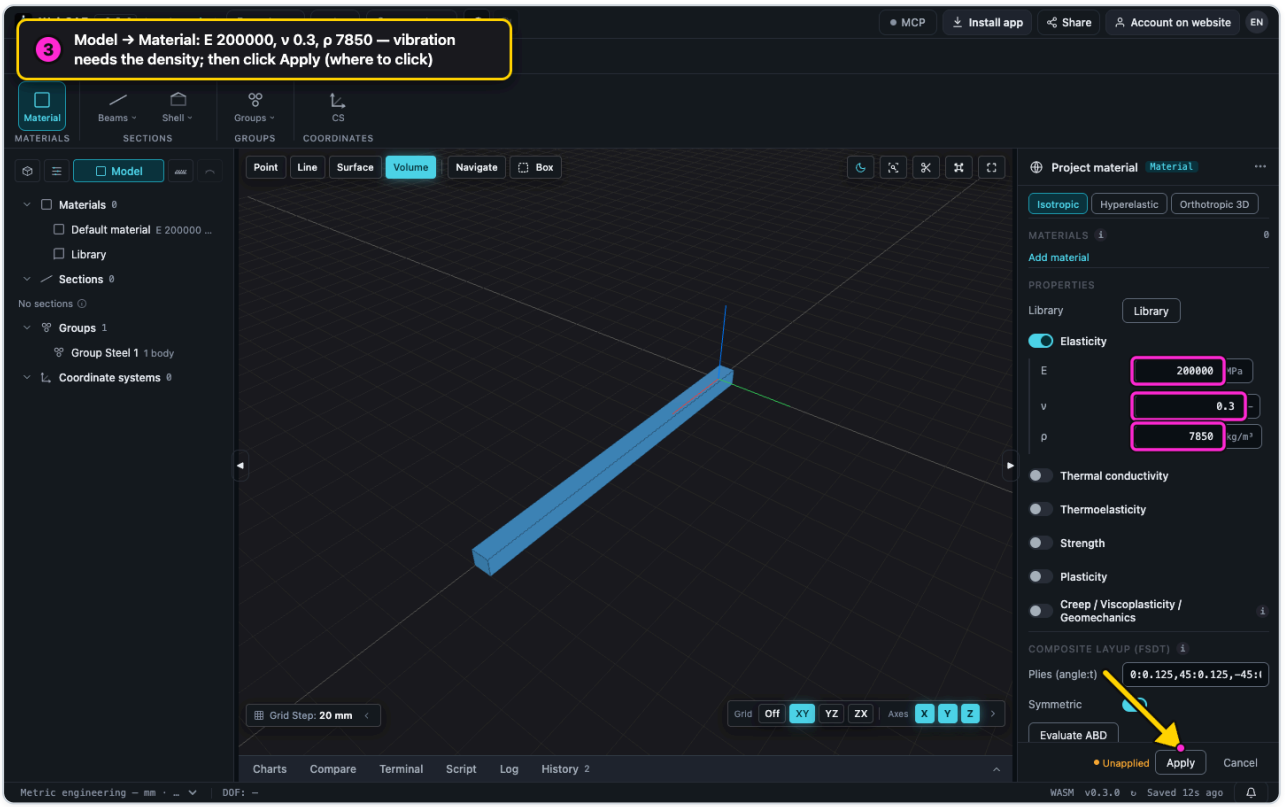


STEP 3

Assign the material, with density

Open **Model** and click **Material**. Type **E** `200000` MPa, **ν** `0.3`, **ρ** `7850` kg/m³ and click **Apply**.

WHY Vibration frequencies depend on the mass matrix; without density modal analysis cannot compute them.



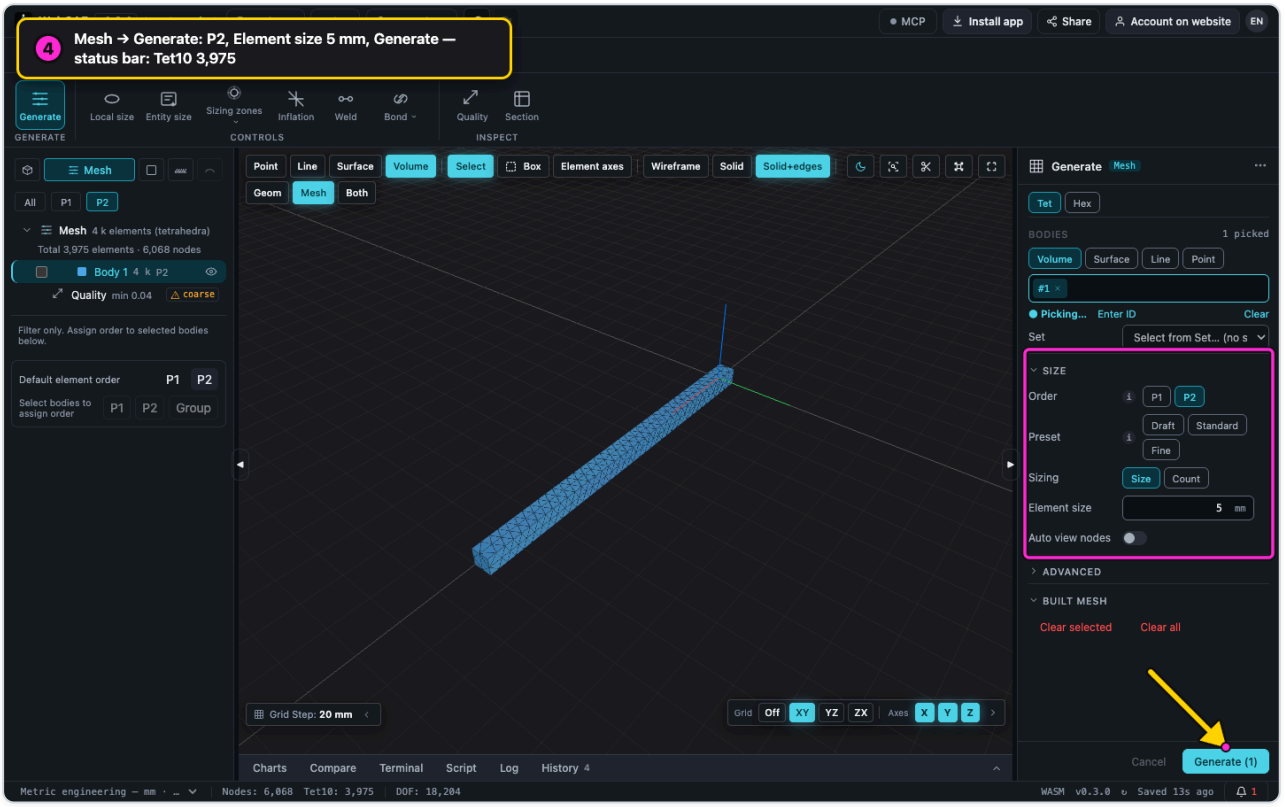
STEP 4

Mesh with quadratic tetrahedra

Open **Mesh**, click the **Generate** tile in the ribbon, choose **P2**, type **Element size** `5` mm and click **Generate (1)** at the bottom of the panel (the number in brackets is how many bodies are picked). The status bar shows **Tet10: 3,975**.

The mesh tree may flag the mesh as *coarse* (quality min ≈ 0.04): that is expected for a 5 mm mesh on a 10 mm section and is accurate enough here.

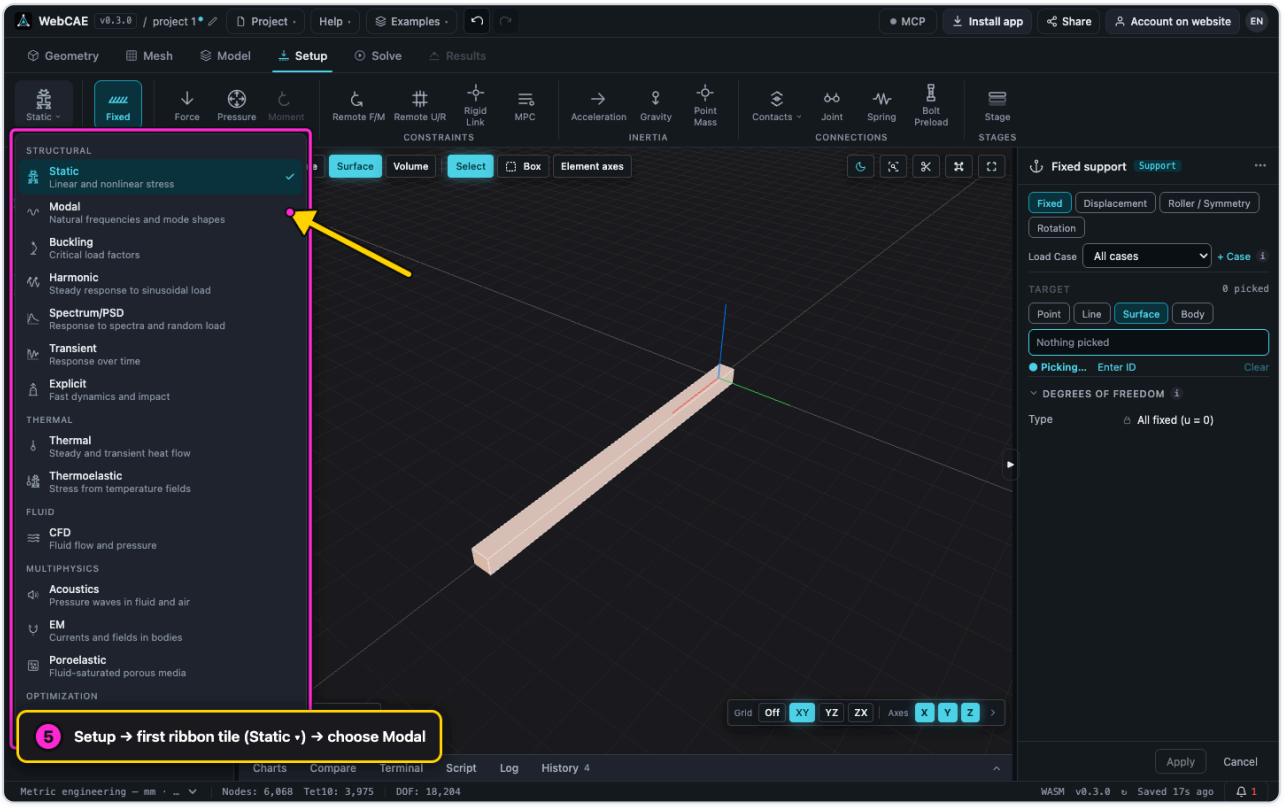
> **WHY** Quadratic tetrahedra do not lock in bending, so a coarse mesh gives accurate frequencies.



STEP 5 Choose the Modal analysis

Open **Setup**. The first tile of the ribbon (group **ANALYSIS**) shows the current analysis type: **Static** ▾ until you change it. Click it and choose **Modal** (*Natural frequencies and mode shapes*) from the menu. The tree on the left now says **Analysis: Modal** and the load tiles (Force, Pressure, …) grey out: free vibration needs supports only.

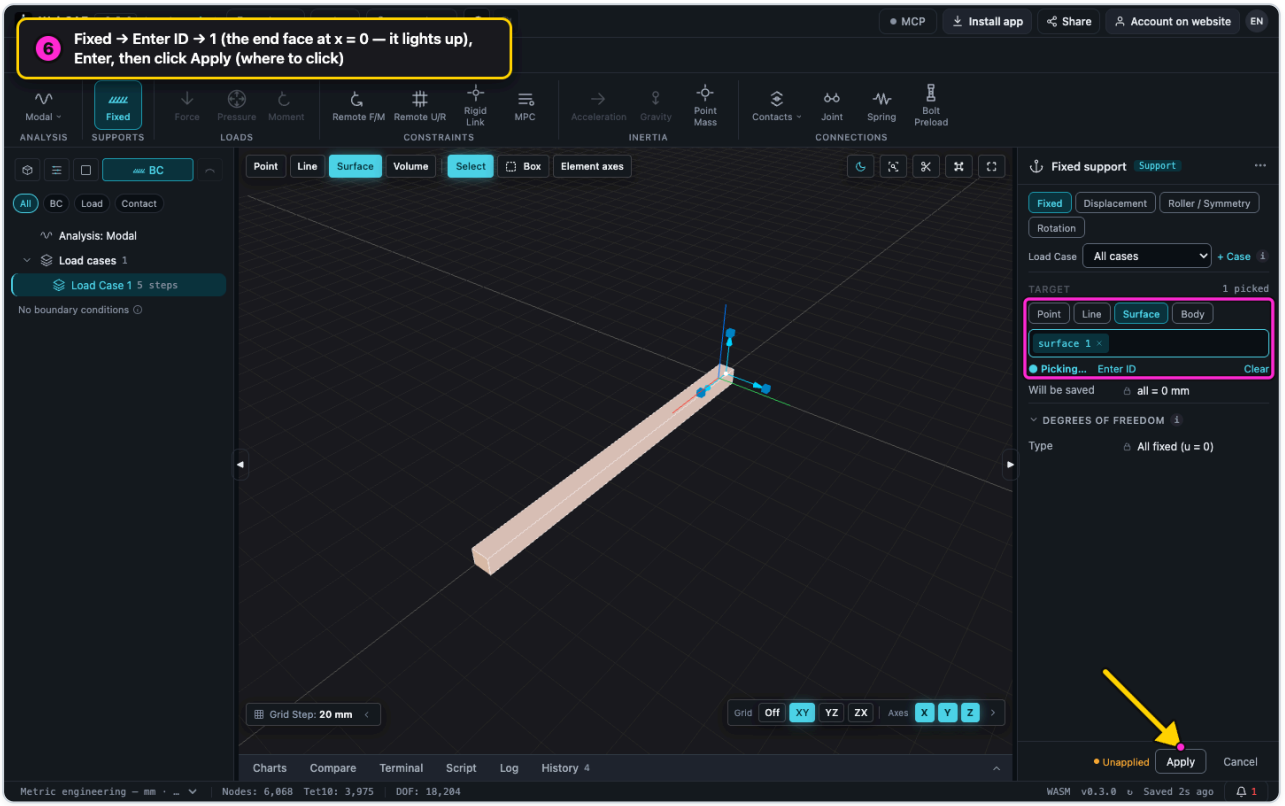
WHY Modal analysis solves $(K - \omega^2 M)\phi = 0$; static loads play no role.



STEP 6 Clamp the root face

Click **Fixed**, click **Enter ID**, type `1`, press **Enter**: the end face at x = 0 (the far end in our view) lights up. Click **Apply**.

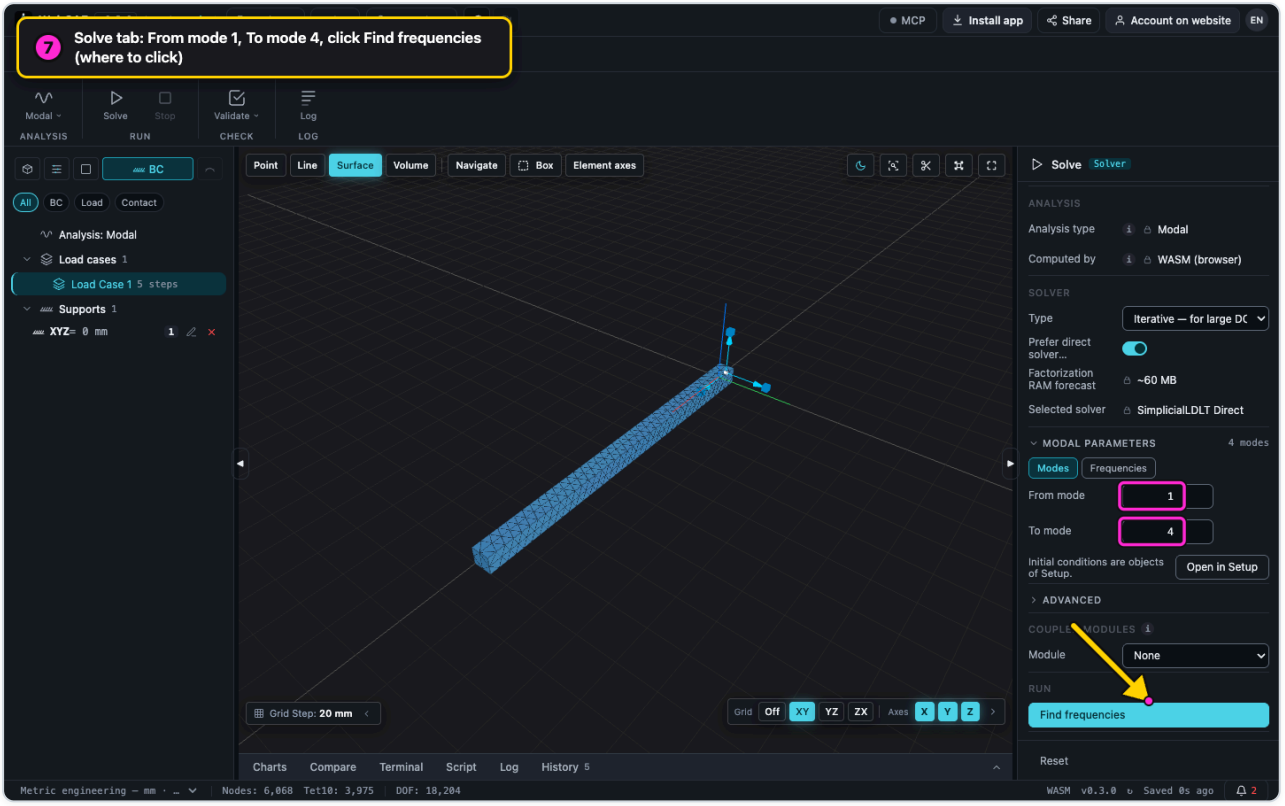
> **WHY** A fixed support removes all translations of the root face, giving the clamped–free bar.



STEP 7

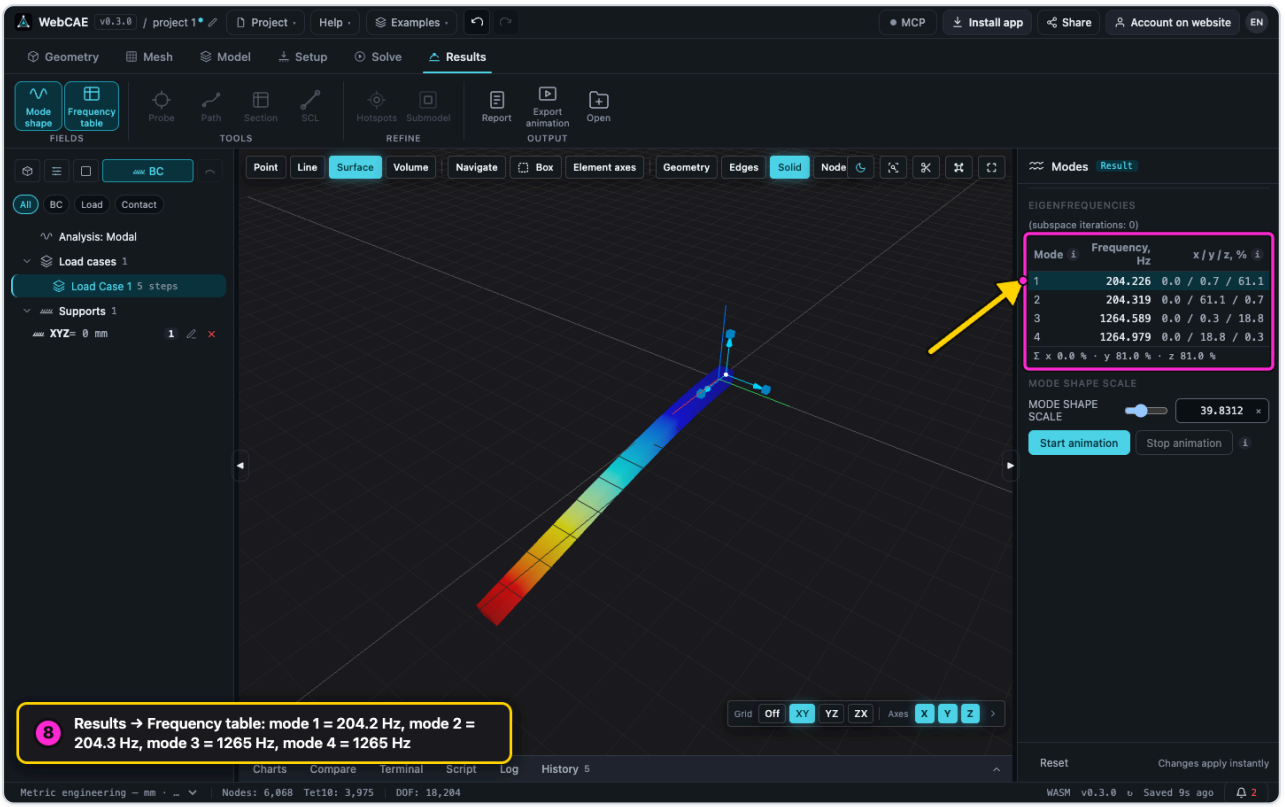
Ask for four modes

Open **Solve**. In the modal parameters keep **From mode** `1` and change **To mode** from its default `6` to `4`, then click **Find frequencies**.



STEP 8 Read the frequencies

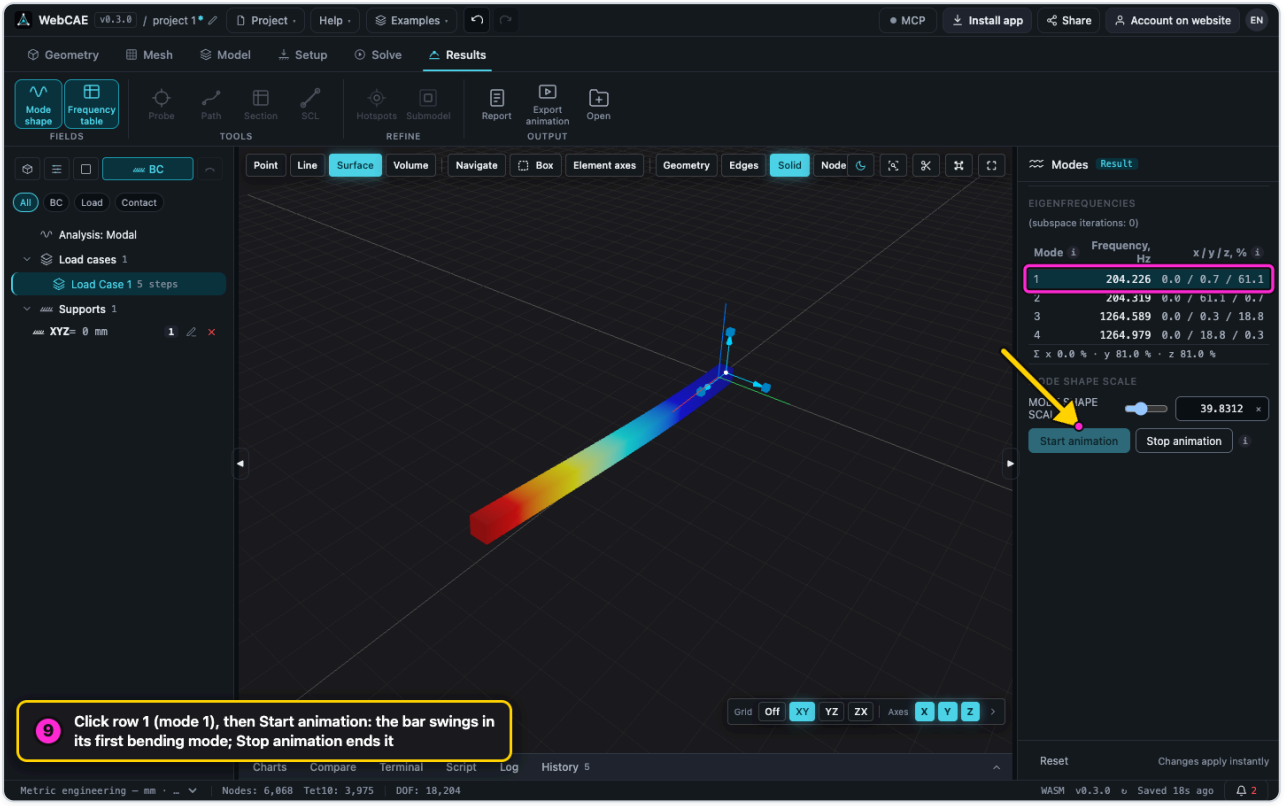
WebCAE jumps to **Results** and shows the **Frequency table**: mode 1 = **204.23 Hz**, mode 2 = **204.32 Hz**, mode 3 = 1264.6 Hz, mode 4 = 1265.0 Hz. The first two are almost equal: a square bar bends equally easily in y and in z (the columns on the right show which direction each mode moves in: 61 % in y for mode 2, 61 % in z for mode 1).



STEP 9

See the mode shape

Click row **1** of the table, then **Start animation** (below the table, next to *Mode shape scale*). The bar swings in its first bending mode; **Stop animation** ends it.



VERIFICATION BENCHMARK

Check against the formula

Mode	Quantity	Theory	WebCAE web (browser)	WebCAE desktop (native)	Difference web / native
Mode 1	Bending f_1	$\frac{1.8751^2}{2\pi L^2}\sqrt{\frac{EI}{\rho A}} = 203.85$ Hz	204.23 Hz (Step 8)	204.13 Hz	0.18 % / 0.14 %
Mode 2	Orthogonal bending f_2	203.85 Hz (same section)	204.32 Hz (Step 8)	204.15 Hz	0.23 % / 0.15 %
Pair ratio	f_2/f_1	1.0000	1.0005	1.0001	0.05 % / 0.01 %
Mode 3	2nd bending f_3	$6.267 \times 203.85 = 1278$ Hz	1264.6 Hz (Step 8)	1263.9 Hz	1.01 % / 1.06 %

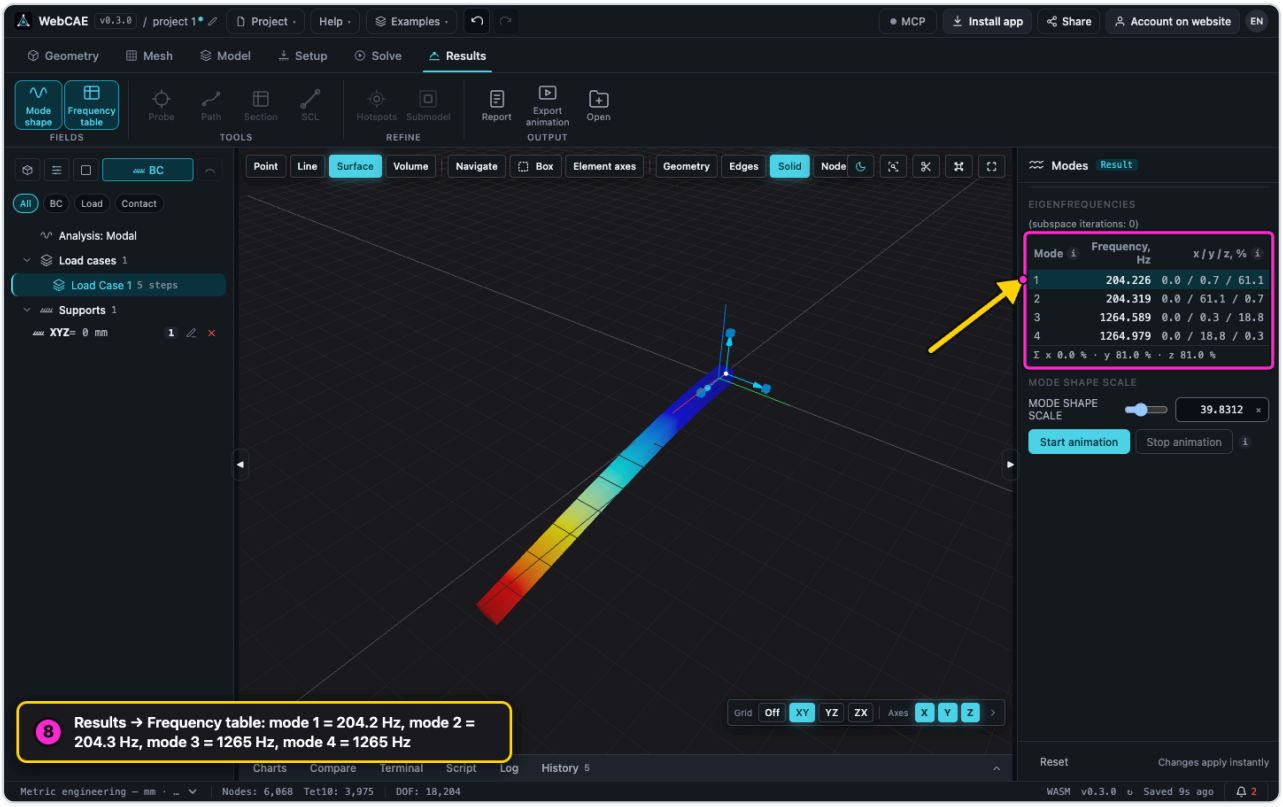
The desktop column comes from a separate desktop run of the same model (native mesher and solver); it is not on a frame.

ANALYTICAL FORMULA

For a uniform cantilever of length $L = 0.2$ m, cross section 10×10 mm ($A = 10^{-4}$ m^2, $I = \frac{10 \times 10^3}{12} \times 10^{-12} = 8.3333 \times 10^{-10}$ m^4), steel $E = 2 \times 10^{11}$ Pa, $\rho = 7850$ kg/m^3:

$$f_1 = \frac{(\beta_1 L)^2}{2\pi L^2}\sqrt{\frac{EI}{\rho A}} = \frac{1.8751^2}{2\pi \times 0.2^2}\sqrt{\frac{2 \times 10^{11} \times 8.3333 \times 10^{-10}}{7850 \times 10^{-4}}} = 203.85 \text{ Hz}$$

The second bending mode has $\beta_2 L = 4.6941$, so $f_3/f_1 = (4.6941/1.8751)^2 = 6.267$. Because the cross-section is square ($I_y = I_z$), the two bending directions are degenerate ($f_1 \approx f_2$). Slenderness is $L/h = 20$, so Euler–Bernoulli is adequate: shear deformation and rotary inertia (Timoshenko) lower f_1 by only about 0.15% to ≈ 203.5 Hz, and more for the second bending mode, which is why mode 3 reads about 1 % below the pure beam value.



TROUBLESHOOTING & NOTES

Common mistakes

- **Omitting material density:** static analysis only needs E and ν, modal analysis needs ρ too. Check $\rho = 7850$ kg/m³ before solving.
- **Looking for the Analysis menu in the wrong place:** it is the first tile of the **Setup** ribbon, not a button in Solve.
- **Expecting loads in modal analysis:** modal analysis computes unforced frequencies; the load tiles are inactive on purpose when Modal is selected.
- **Using the mode animation as a result:** the animation scale is exaggerated on purpose; the numbers are in the table.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#5 — Column buckling in ten steps (F05)**: the Euler critical load of a column under axial compression.

WebCAE 101

LESSON 5 · F05

Column buckling in ten steps

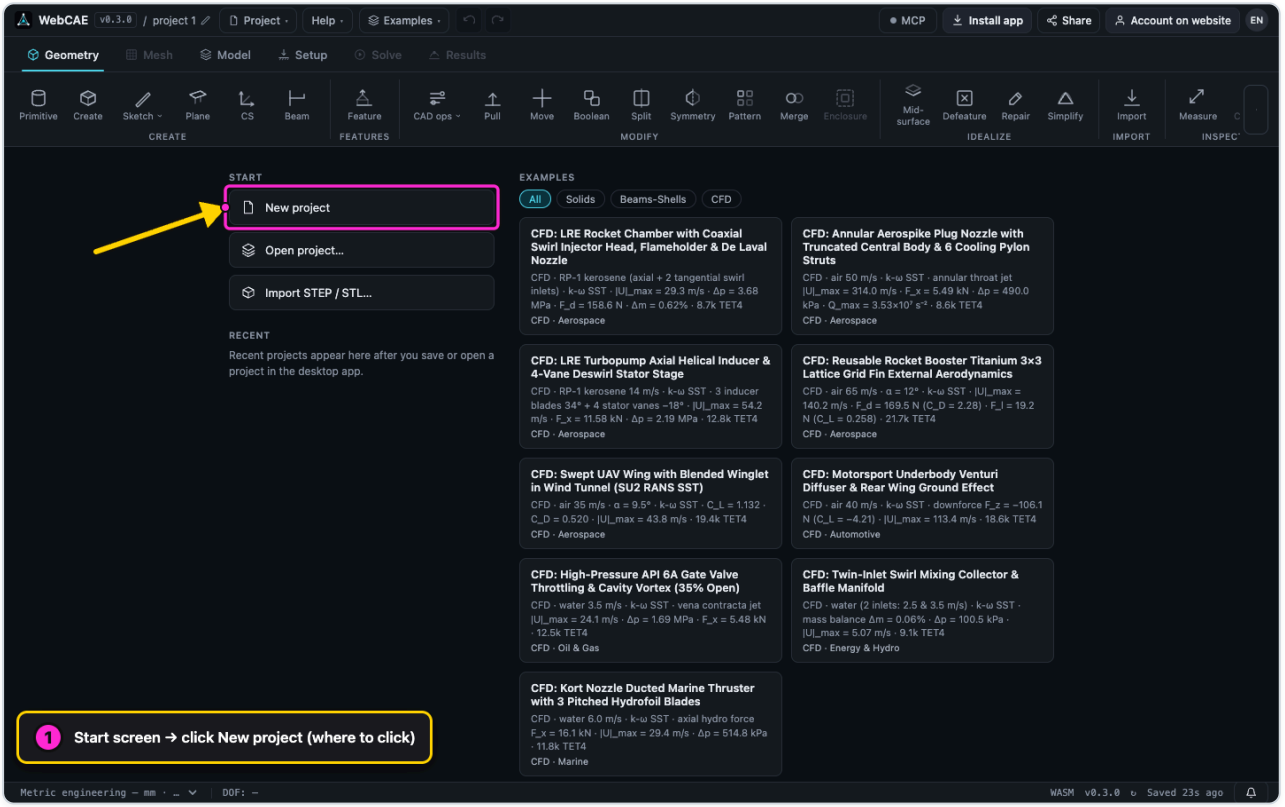
Goal Compute the Euler buckling load of a slender steel column and check it against the classical formula.

Outcome The critical load multiplier λ_1: 4.569 by hand, 4.576 in WebCAE web and 4.576 in WebCAE desktop ($P_{cr} = 4575.8\ \mathrm{N}$ web).

5–6 min | web (WASM) | Run: 2026-11-04-32ffaed2 / F05

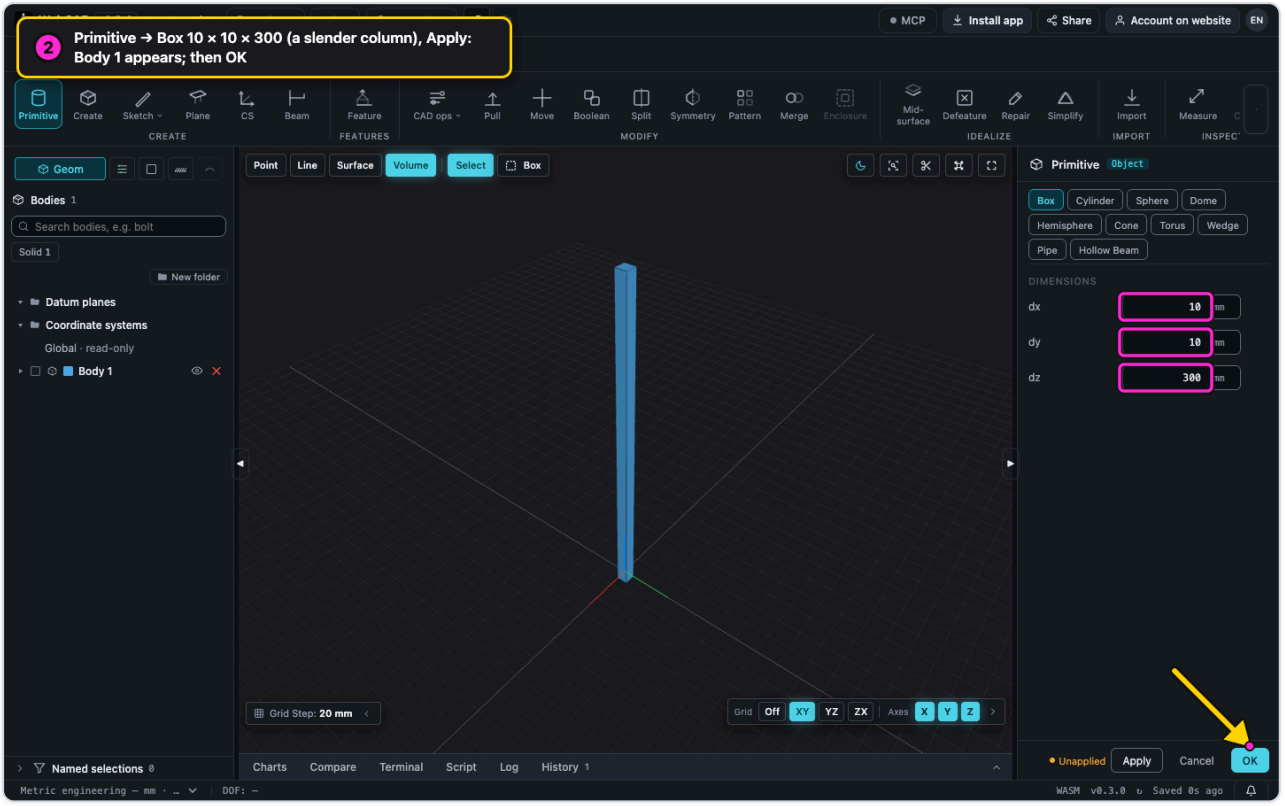
STEP 1 Start a new project

When WebCAE opens, it shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and a gallery of examples). The workspace behind it is not usable until you leave it: click **New project**.



STEP 2 Slender column geometry

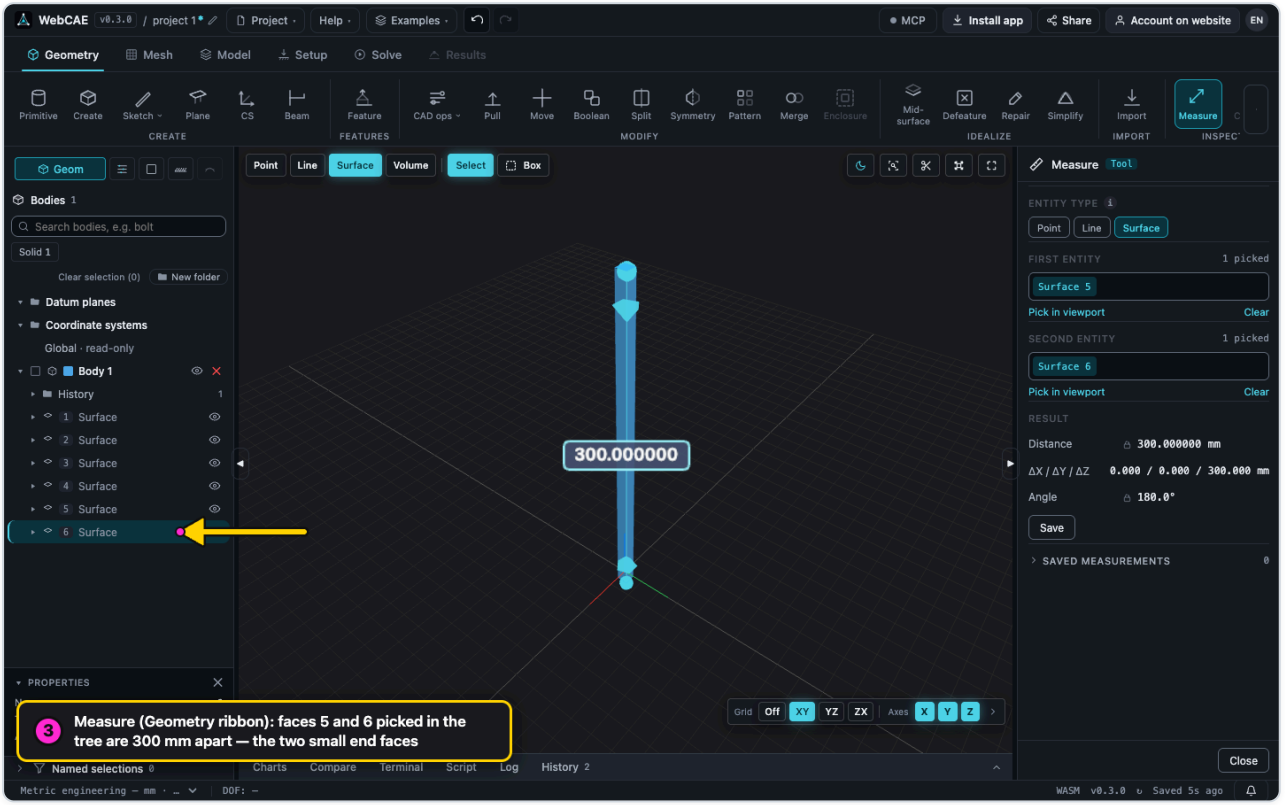
In **Geometry** click **Primitive**, keep **Box**, keep **dx** at `10` (the form starts at 10 × 20 × 30, so **dx** is already right), type **dy** `10` and **dz** `300` (mm), click **Apply** (Body 1 appears), then **OK**.



STEP 3

Find the two small end faces

Expand **Body 1** in the tree, click **Measure** in the Geometry ribbon (Inspect group, entity type **Surface**) and click **5 Surface** and **6 Surface**. The **Distance** is **300 mm**: these are the two small square ends, one at z = 0 and one at z = 300. Face 5 is the bottom one, 6 the top (ΔZ in the Measure panel is the signed difference). Click **Close**.

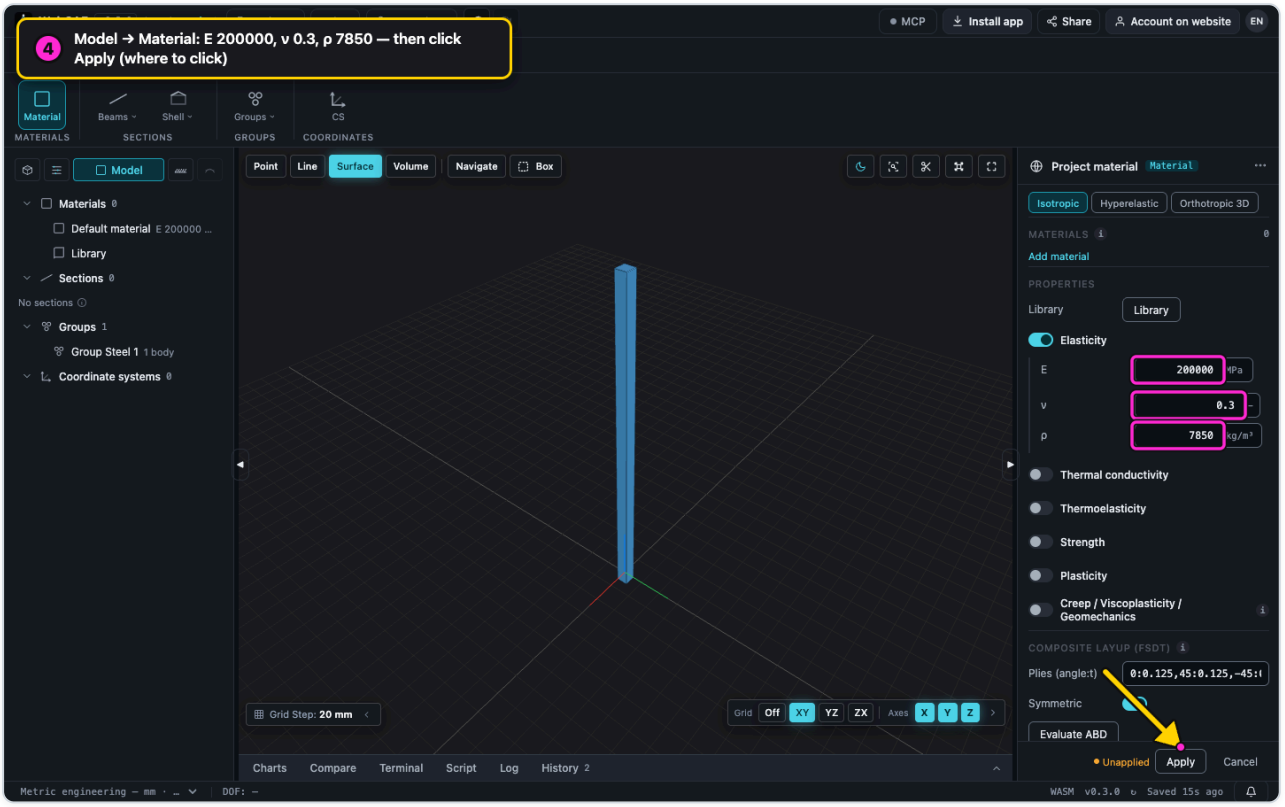


STEP 4

Assign structural steel properties

Open **Model**, click **Material**, set **E** `200000` MPa, **ν** `0.3`, **ρ** `7850` kg/m³ and click **Apply**.

WHY Young's modulus E scales the bending stiffness EI in Euler buckling.



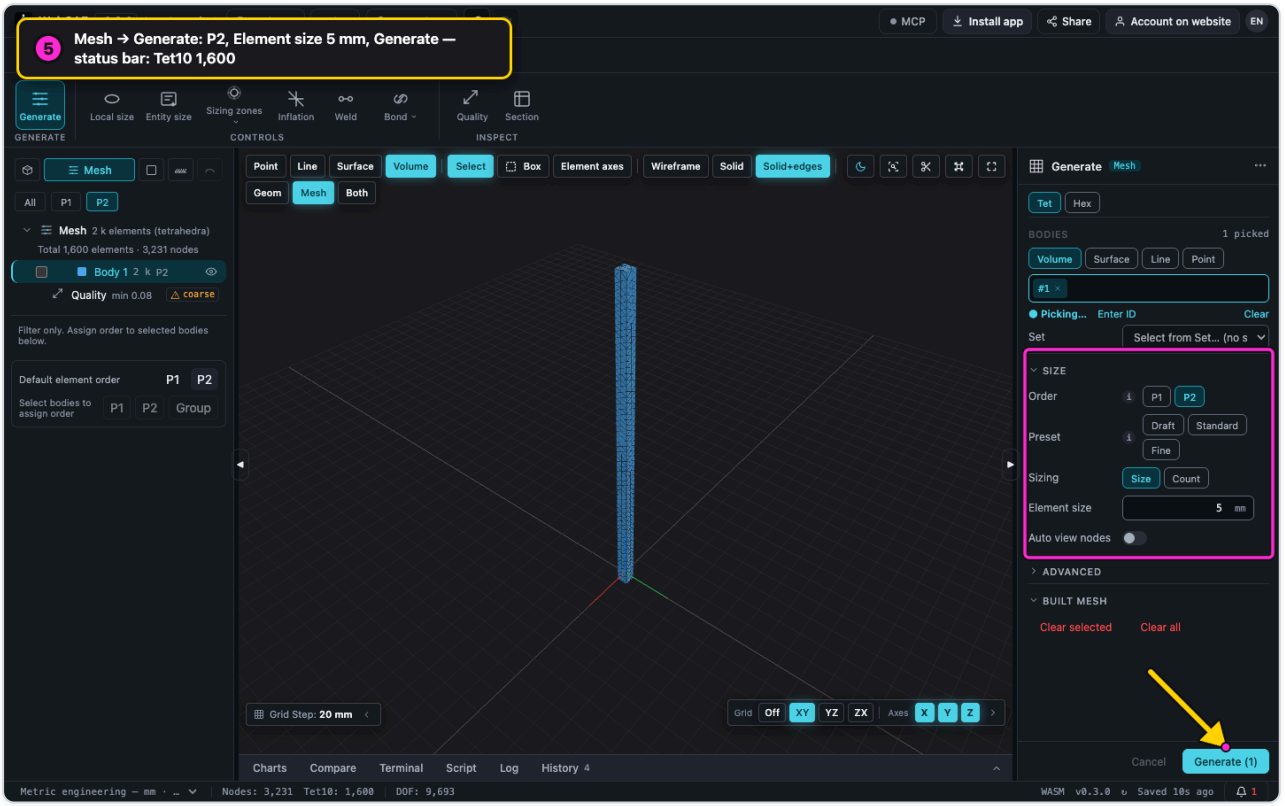
STEP 5

Quadratic tetrahedral mesh

Open **Mesh**, click the **Generate** tile in the ribbon, choose **P2**, type **Element size** `5` mm, click **Generate (1)** at the bottom of the panel (the number in brackets is how many bodies are picked). The status bar shows **Tet10: 1,600**.

The mesh tree may flag the mesh as *coarse*: that is expected for a 5 mm mesh on a 10 mm section.

> **WHY** Quadratic tetrahedra capture the bending curvature without locking.

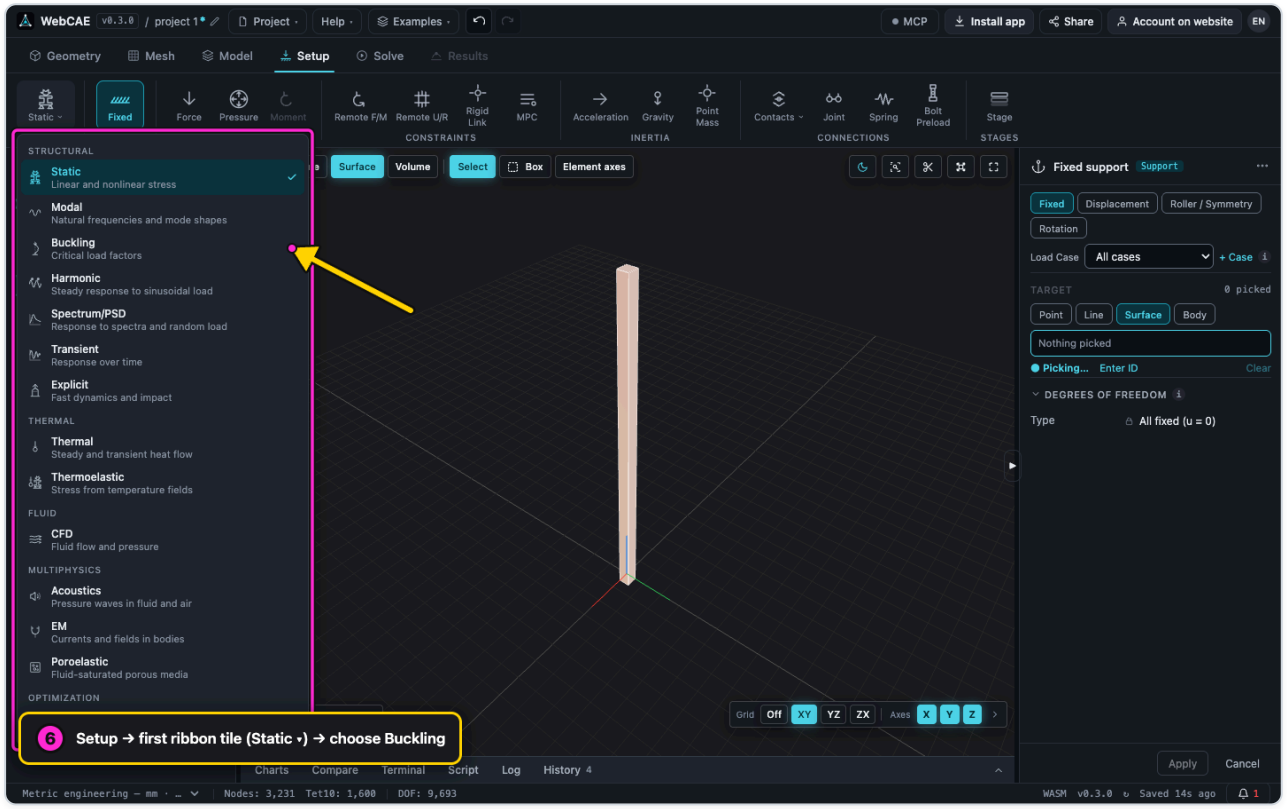


STEP 6

Select Buckling analysis

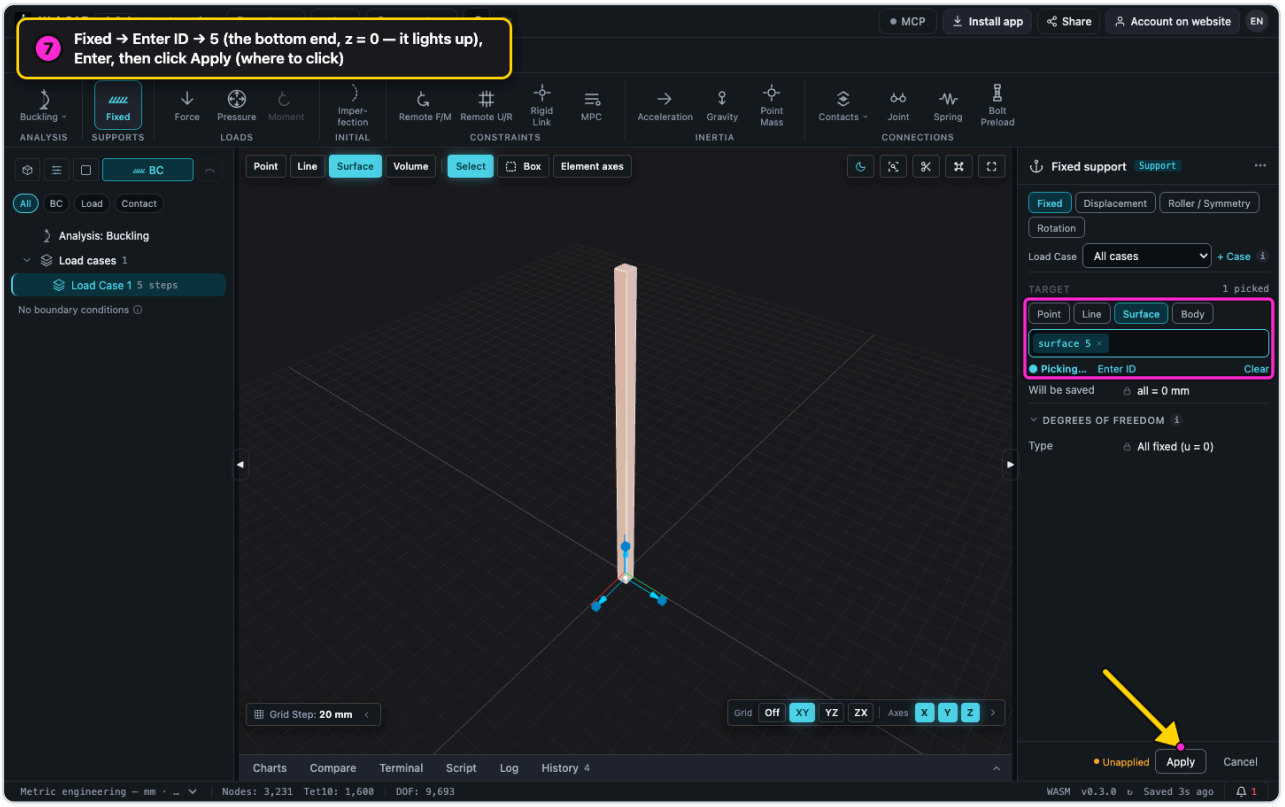
Open **Setup**. The first tile of the ribbon (group **ANALYSIS**) shows the current analysis type: **Static** ▾ by default. Click it and choose **Buckling** (*Critical load factors*).

WHY Buckling solves $Ku = \lambda K_\sigma u$ and needs a reference load.



STEP 7 Clamp the bottom face

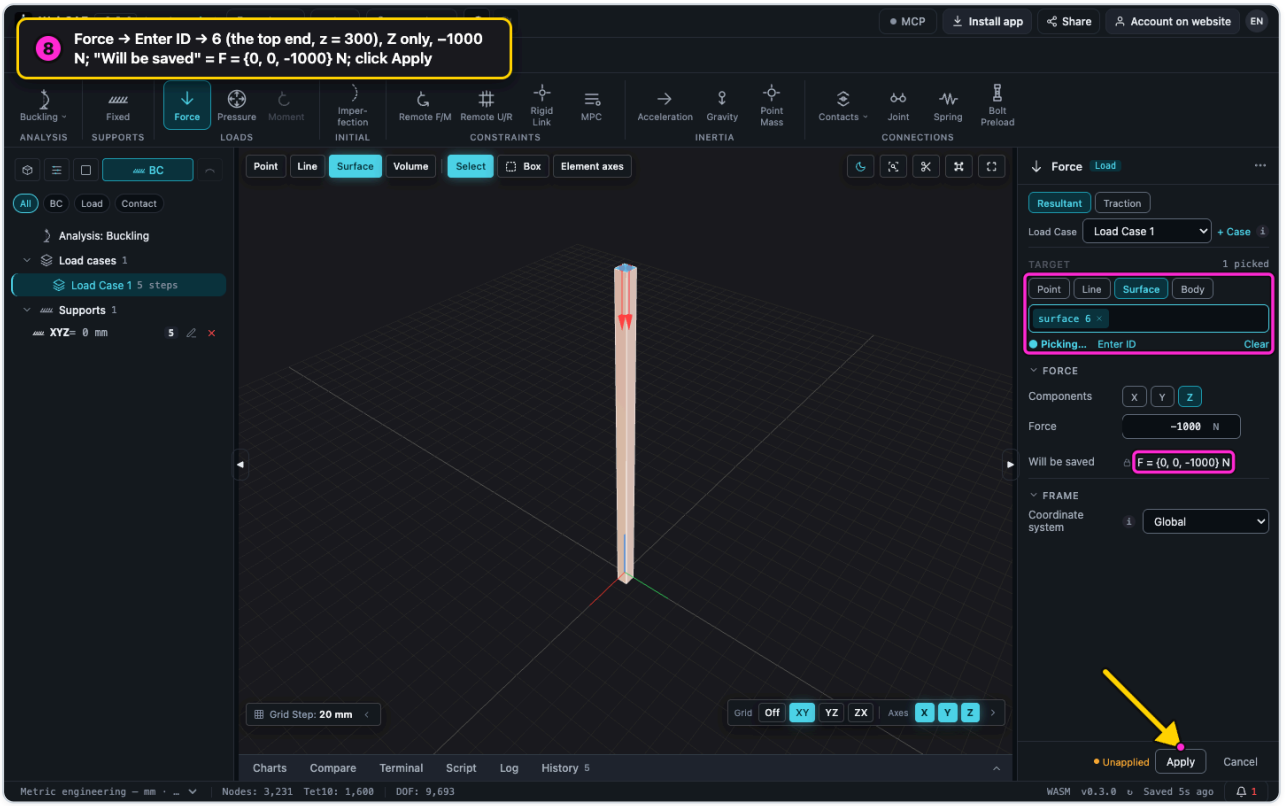
Click **Fixed**, **Enter ID**, type `5`, press **Enter**: the bottom end face lights up. Click **Apply**.



STEP 8 Reference compressive load

Click **Force**, **Enter ID**, type `6`, press **Enter**. Choose the **Z** component only and type `−1000` N. **Will be saved** must read `F = {0, 0, −1000} N`. Click **Apply**.

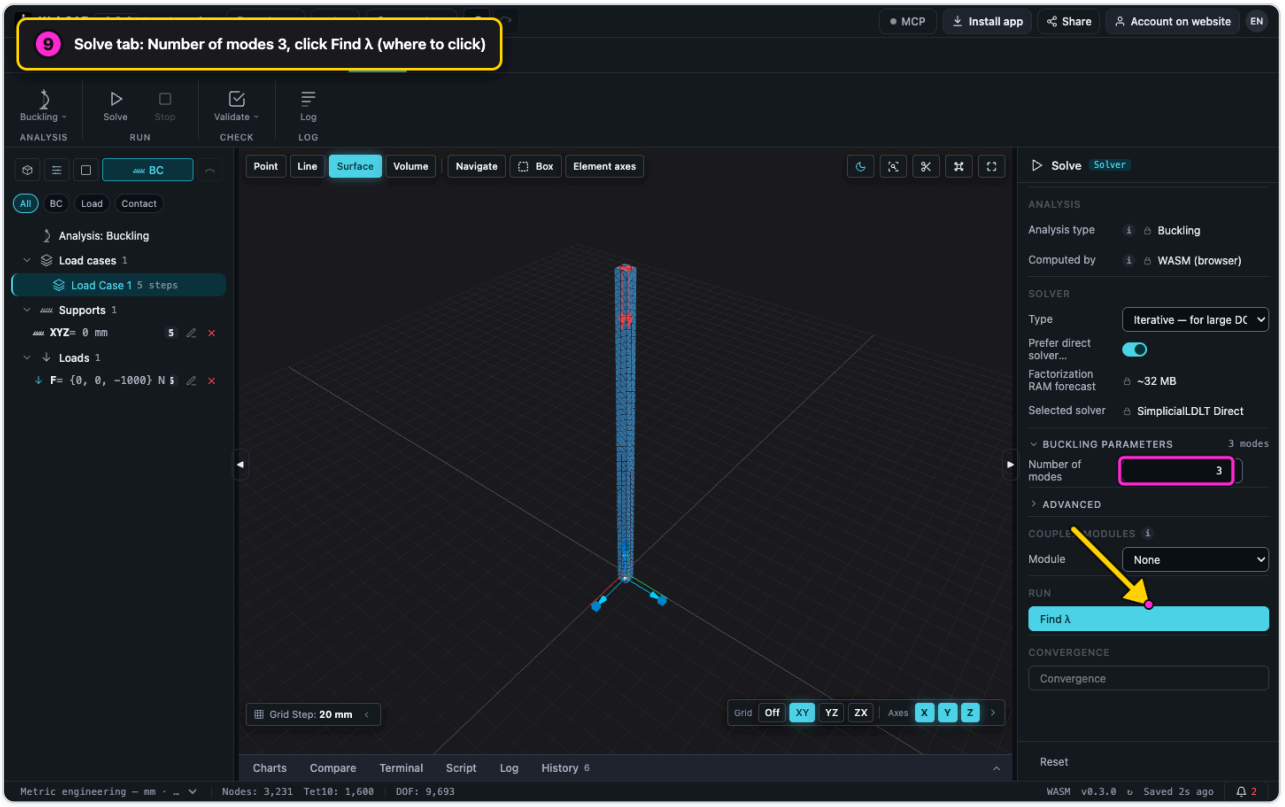
> **WHY** The eigenvalues λ multiply the reference load; pure axial compression along −Z gives the column its geometric stiffness.



STEP 9

Find λ for three modes

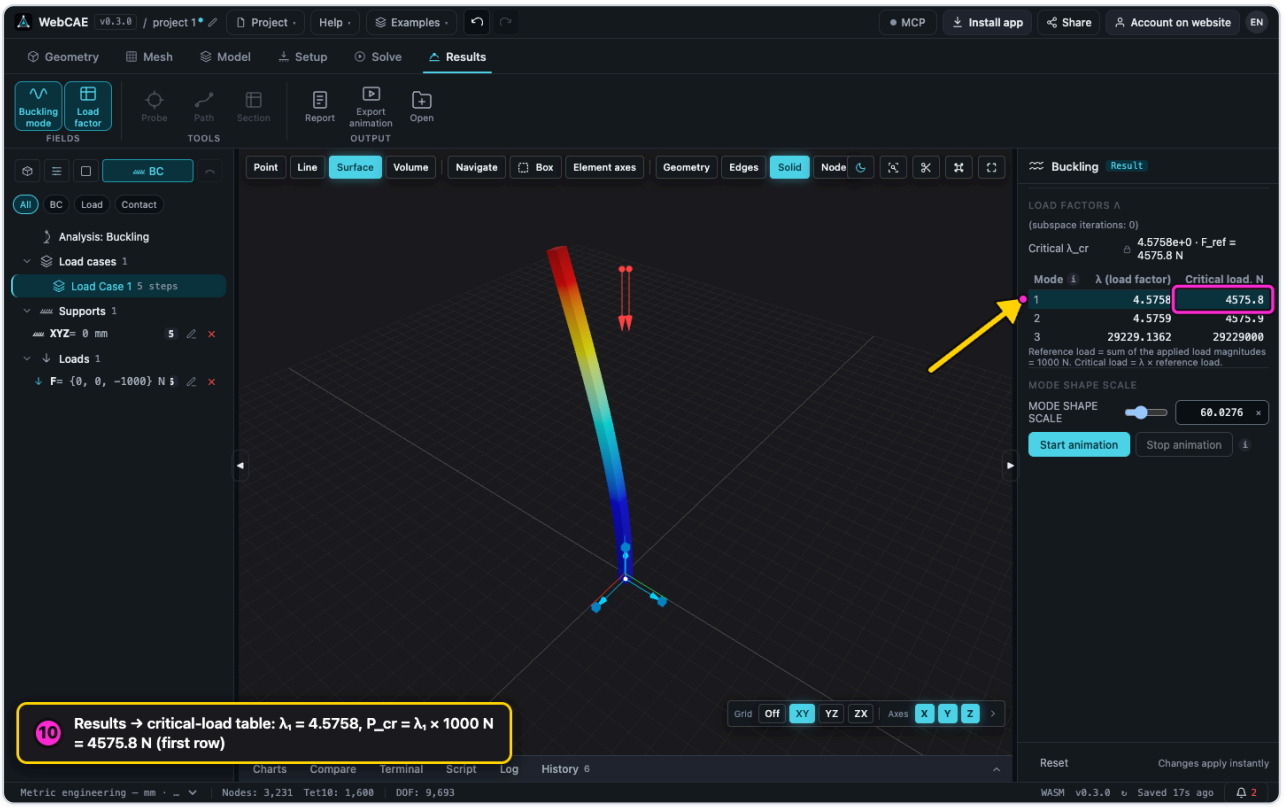
Open **Solve**, set **Number of modes** to `3` and click **Find λ**.



STEP 10

Read the critical load factor

WebCAE jumps to **Results** and shows the critical-load table. The first row gives λ_1 = **4.5758**, so $P_{cr} = \lambda_1 \times 1000\ \mathrm{N}$ = **4575.8 N**. Rows 1 and 2 are nearly equal: a square section buckles equally easily in both planes. The first mode shape is already drawn on the column and row 1 of the table is highlighted. Click row **2**: the column is drawn bent in a different direction (the other bending plane of the square section, with almost the same λ), and row **1** brings the first shape back.



VERIFICATION BENCHMARK

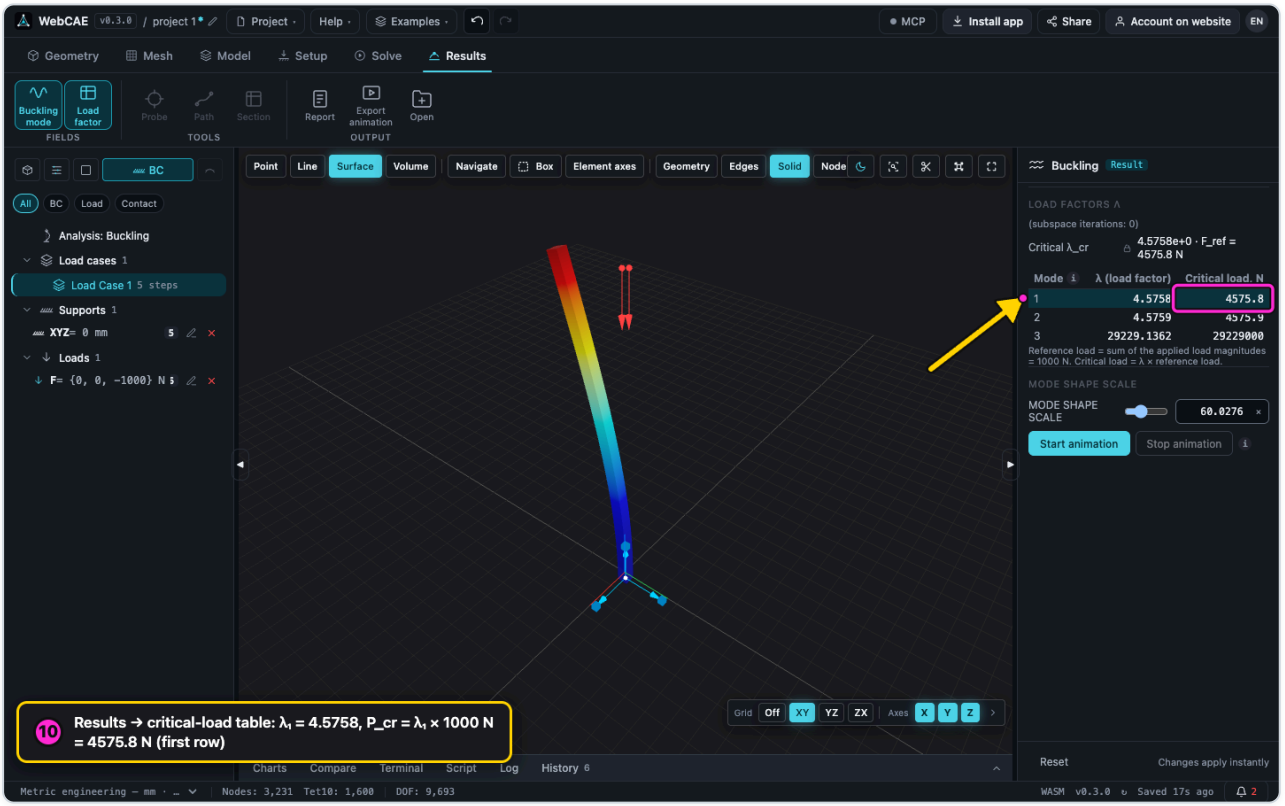
Check against the formula

Quantity	Theory (Euler)	WebCAE web (browser)	WebCAE desktop (native)	Error web / native
Critical load factor λ_1	4.569	4.576 (Step 10)	4.576	0.15 % / 0.15 %
Critical load P_{cr}	4 569.3 N	4575.8 N (Step 10)	4575.9 N	0.15 % / 0.14 %

The desktop column comes from a separate desktop run of the same model (native mesher and solver); it is not on a frame.

FORMULA

$P_{cr} = \frac{\pi^2 EI}{(\mu L)^2} = \frac{\pi^2 \times 200\,000 \times 833.33}{(2\times 300)^2} = 4\,569.3\ \text{N} \rightarrow \lambda_1 = 4.569$ (cantilever column $\mu = 2$: clamped base, free top). Slenderness check: $r = \sqrt{I/A} = 2.89\ \text{mm}$, so $\mu L/r = 600/2.89 \approx 208$, well above the Euler limit of about 100 for mild steel, and $P_{cr}/A = 45.7\ \text{MPa}$ is far below the yield strength, so elastic Euler buckling applies.



TROUBLESHOOTING & NOTES

Common mistakes

- **Wrong force axis:** if the component is left on the default or set along Y instead of Z, the column bends sideways instead of being compressed and the eigenvalues are wrong. Always check `Will be saved: F = {0, 0, −1000} N`.
- **Wrong end faces:** the small ends are the two faces that are 300 mm apart (step 3); the four long faces are not supports.
- **Element order:** linear (P1) tetrahedra are too stiff in slender bending and overestimate the critical load; use quadratic (P2).

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#6 — Steady heat conduction in a cooling fin (F06)**: steady-state heat conduction and convection.

WebCAE 101

LESSON 6 · F06

Steady heat conduction in a cooling fin

Goal Compute the temperature distribution and heat dissipation of a cooling fin, and verify against the classical 1D analytical fin equation.

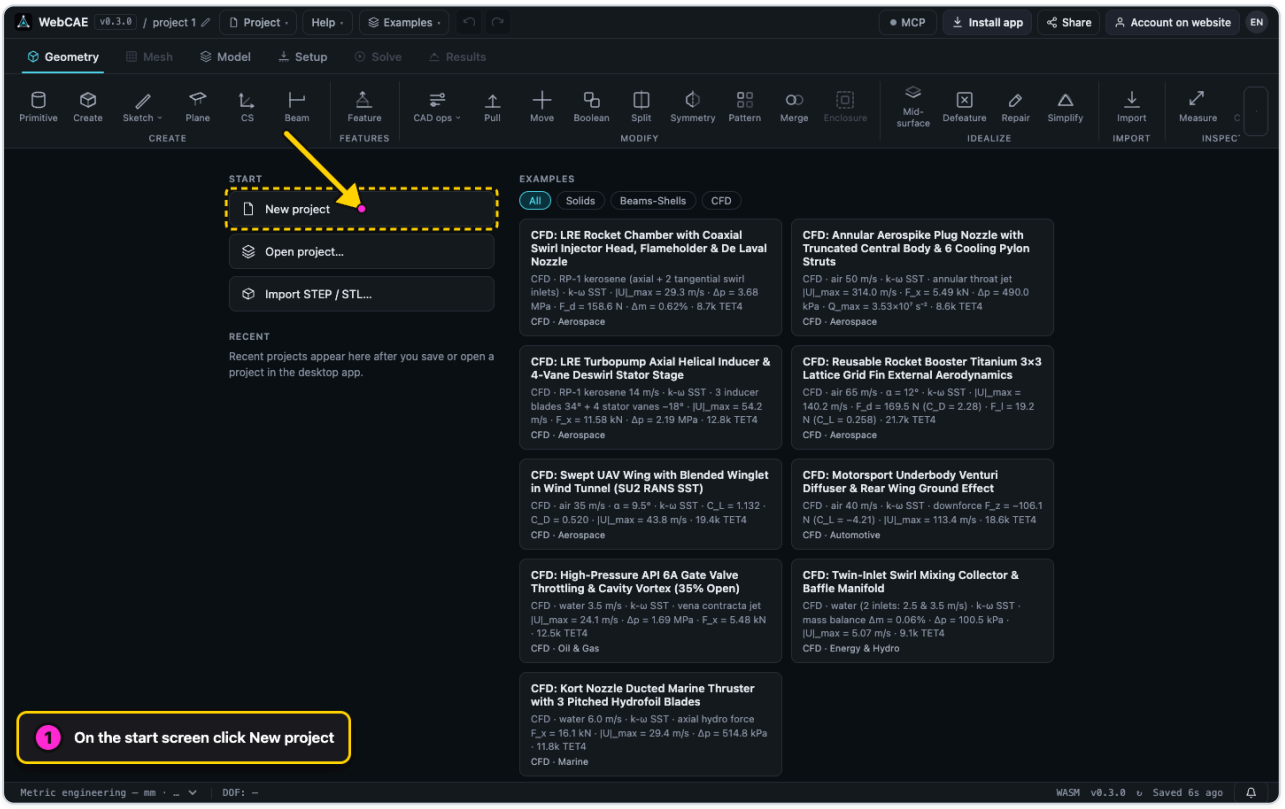
Outcome Tip temperature $T_{\text{tip}} = 71.37\ ^\circ\text{C}$ by hand vs $71.34\ ^\circ\text{C}$ in WebCAE (browser; desktop app: $71.34\ ^\circ\text{C}$); base heat flow $q_{\text{base}} = 12.42\ \text{W}$ by hand vs $12.42\ \text{W}$ in WebCAE (desktop: $12.42\ \text{W}$).

3–4 min | web (WASM) | Run: 2026-11-04-32ffaed2 / F06

STEP 1 Start a new project

Open WebCAE. The centre of the window shows the **Start** screen (a column **START** with **New project**, **Open project...**, **Import STEP / STL...** and a gallery of **EXAMPLES**). Click **New project** (the arrow on the frame): the screen closes and an empty model with the Geometry ribbon opens. If you already see an empty model (**Model is empty**) and no Start screen, skip this step.

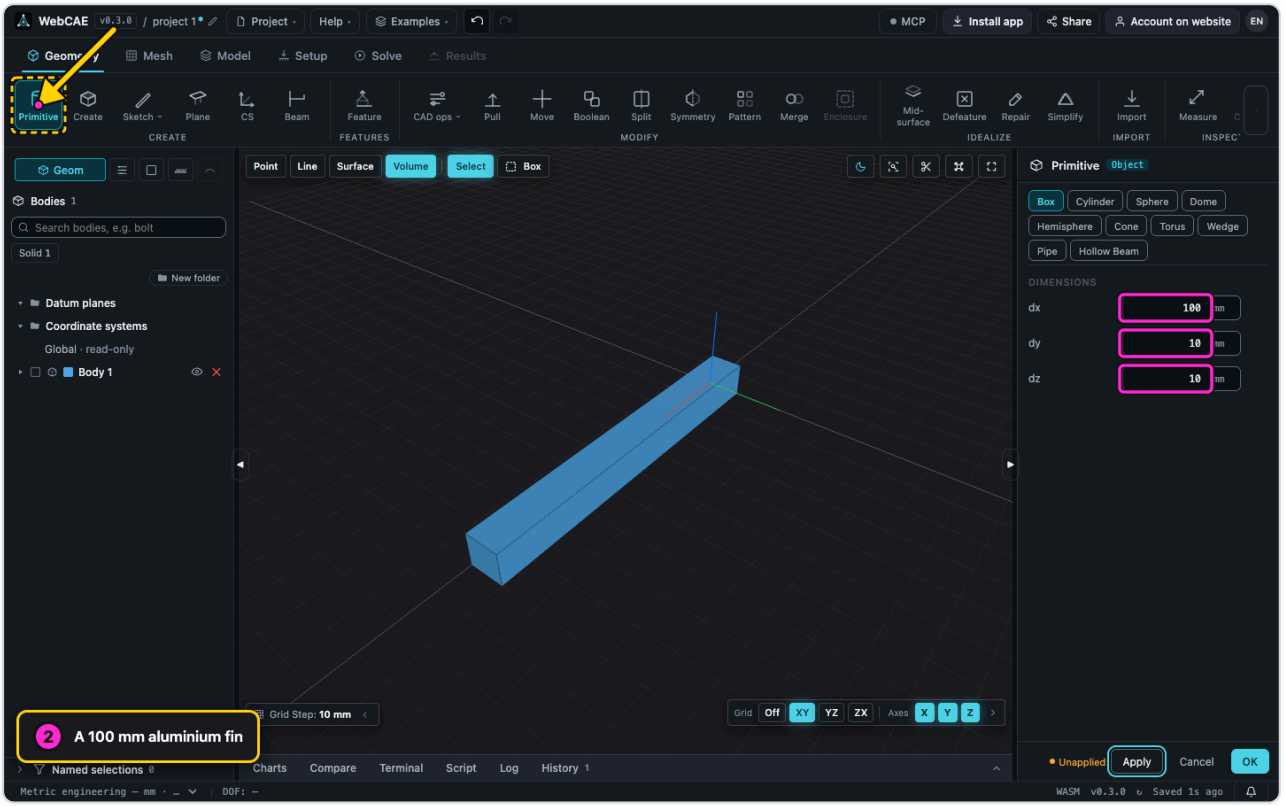
WHY The Start screen covers the viewport and the model tree, and you cannot build anything until you choose what to open; **New project** gives the clean model that this lesson starts from.



STEP 2 Fin geometry

Click **Primitive** in the ribbon, select **Box**, and set **dx** = `100` mm, **dy** = `10` mm, and **dz** = `10` mm, then click **Apply** and then **OK** (the frame is taken after Apply: the body is in the tree and in the viewport).

WHY The fin is a square $10 \times 10\ \text{mm}$ aluminium rod extending $100\ \text{mm}$ along the X-axis ($V = 10\,000\ \text{mm}^3$).

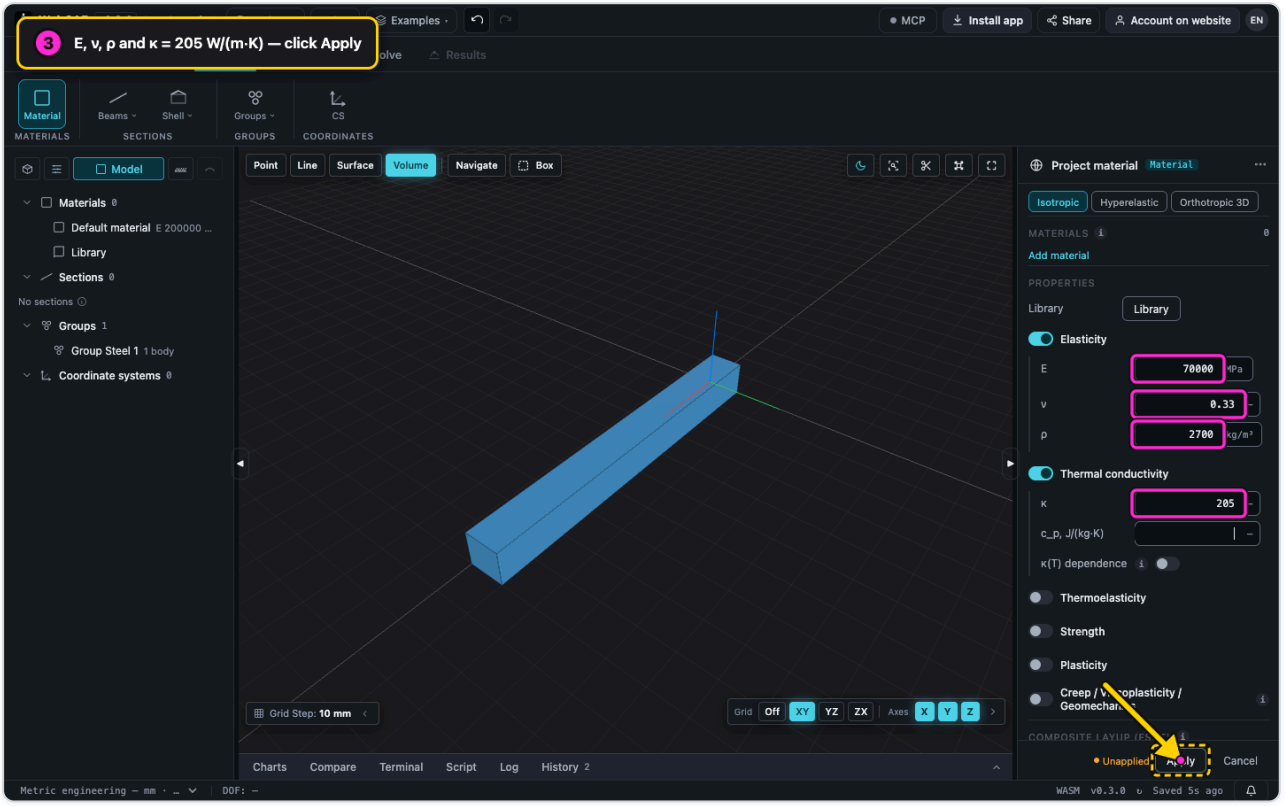


STEP 3

Aluminium thermal conductivity

Switch to **Model** in the ribbon and open **Material**. Set $E = 70\,000\ \mathrm{MPa}$, $\nu = 0.33$, $\rho = 2\,700\ \mathrm{kg/m^3}$. Switch on the **Thermal conductivity** toggle and set $\kappa = 205\ \mathrm{W/(m \cdot K)}$. Click **Apply** (the arrow on the frame).

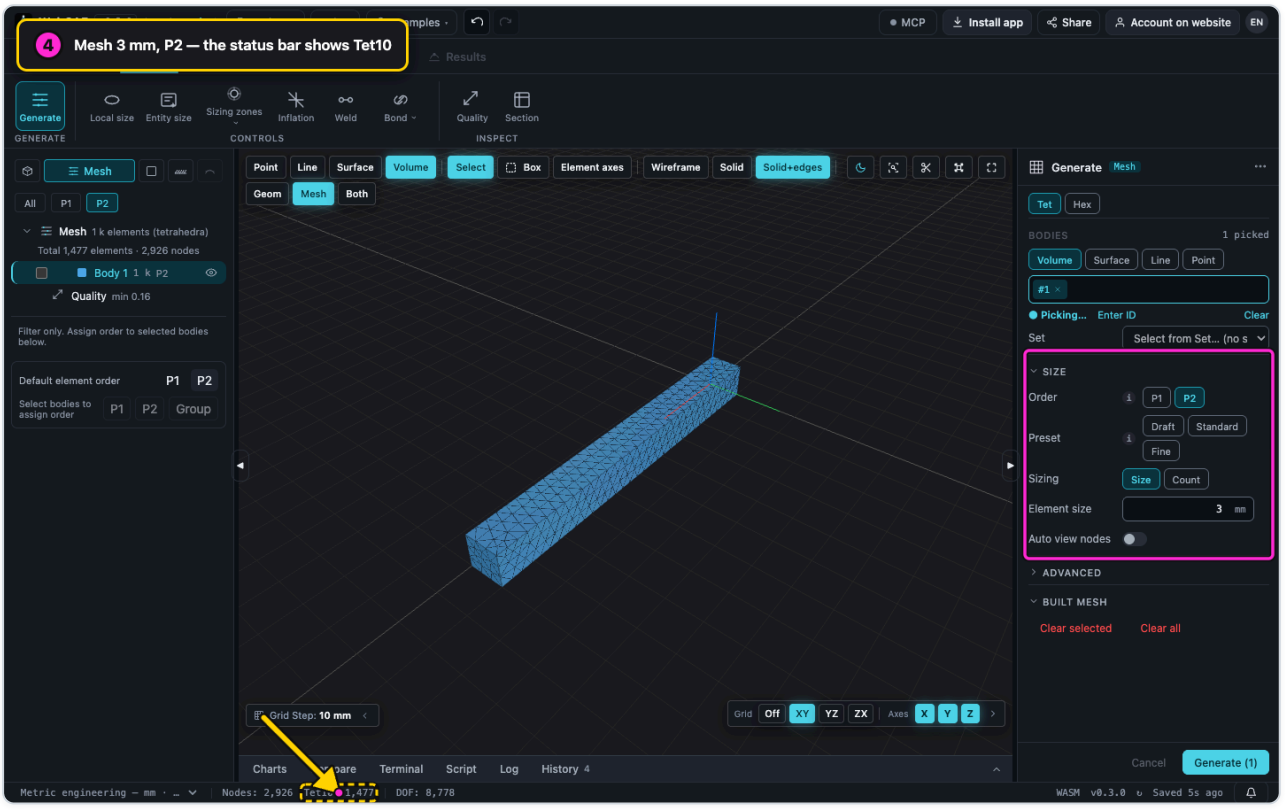
WHY Steady heat conduction depends directly on the thermal conductivity κ of the material.



STEP 4 Quadratic solid mesh

Switch to **Mesh** in the ribbon, open **Generate**, select quadratic **P2** elements, set the element size to `3` mm and click **Generate**. The frame is taken after the mesh is built: the status bar shows **Tet10: 1477** (about 1477 elements in this browser run; the desktop app builds a slightly different mesh, 1357 elements — the meshers differ, the result does not).

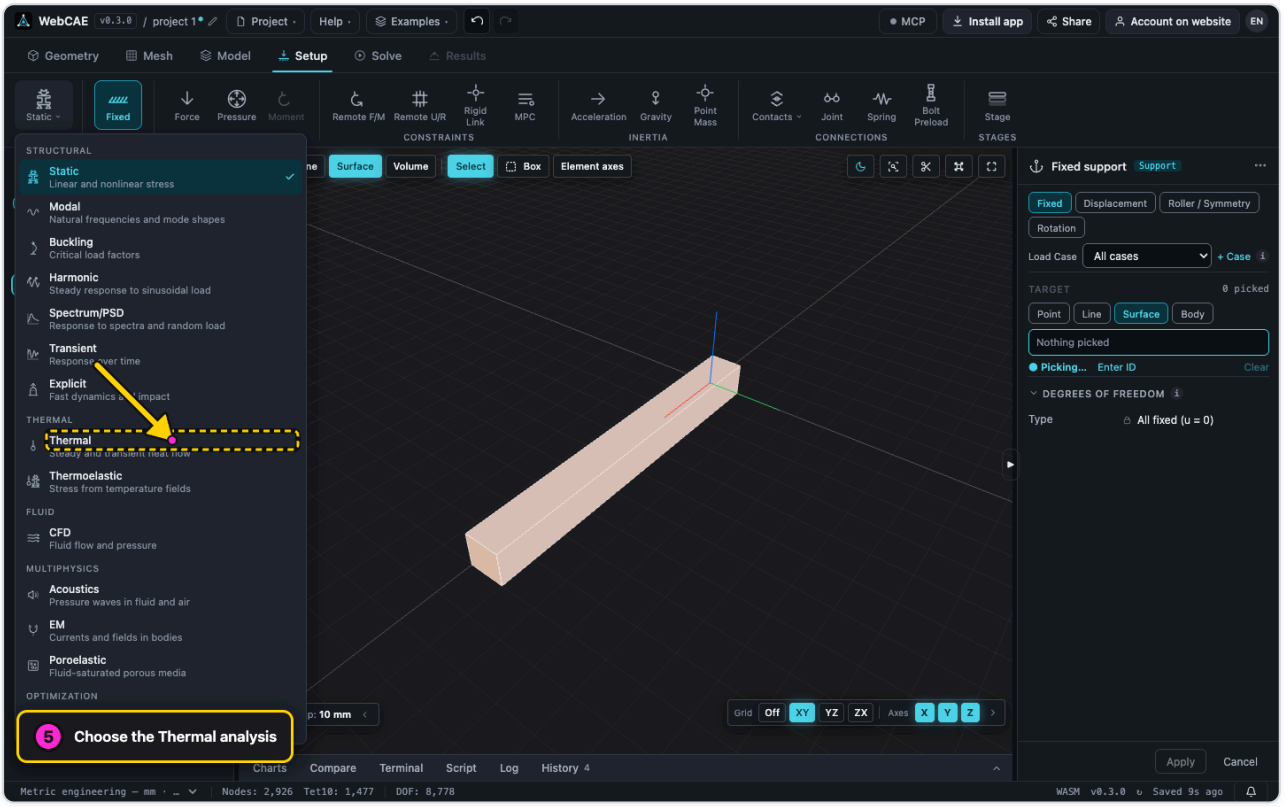
WHY A 3 mm quadratic mesh resolves the temperature gradient across the 10×10 mm section and along the 100 mm length.



STEP 5 Select Thermal analysis

Switch to **Setup** in the ribbon, click the first tile of the ribbon, the **Analysis** dropdown (it shows the current analysis type, **Static**, with a small arrow), and choose **Thermal** (it sits in the **THERMAL** group of the menu; its second line reads *Steady and transient heat flow*, which is just the hint under the name).

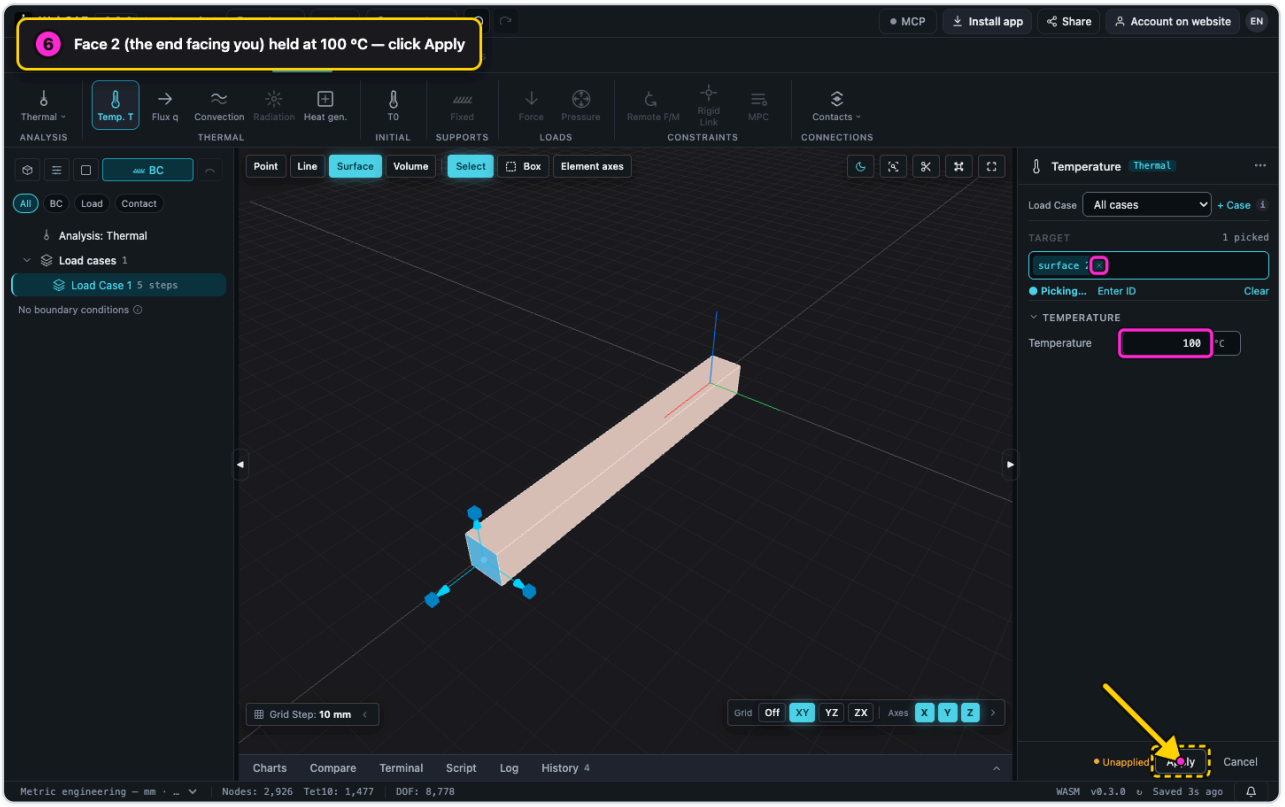
WHY Selecting Thermal analysis switches the ribbon to thermal boundary conditions (**Temp. T**, **Flux q**, **Convection**, **Heat gen.**) and configures the thermal solver ($-\nabla \cdot (\kappa \nabla T) = Q$).



STEP 6 Base temperature boundary condition

Click **Temp. T** in the ribbon, click **Enter ID**, type `2` and press **Enter**. Face **2** is the small square end of the fin at $x = 100\ \mathrm{mm}$ — the end that faces you in the default view; it turns highlighted in the viewport when picked (the number comes from the model tree: **2 Surface** under the body). Check that **Temperature** is `100` °C and click **Apply** (the arrow on the frame).

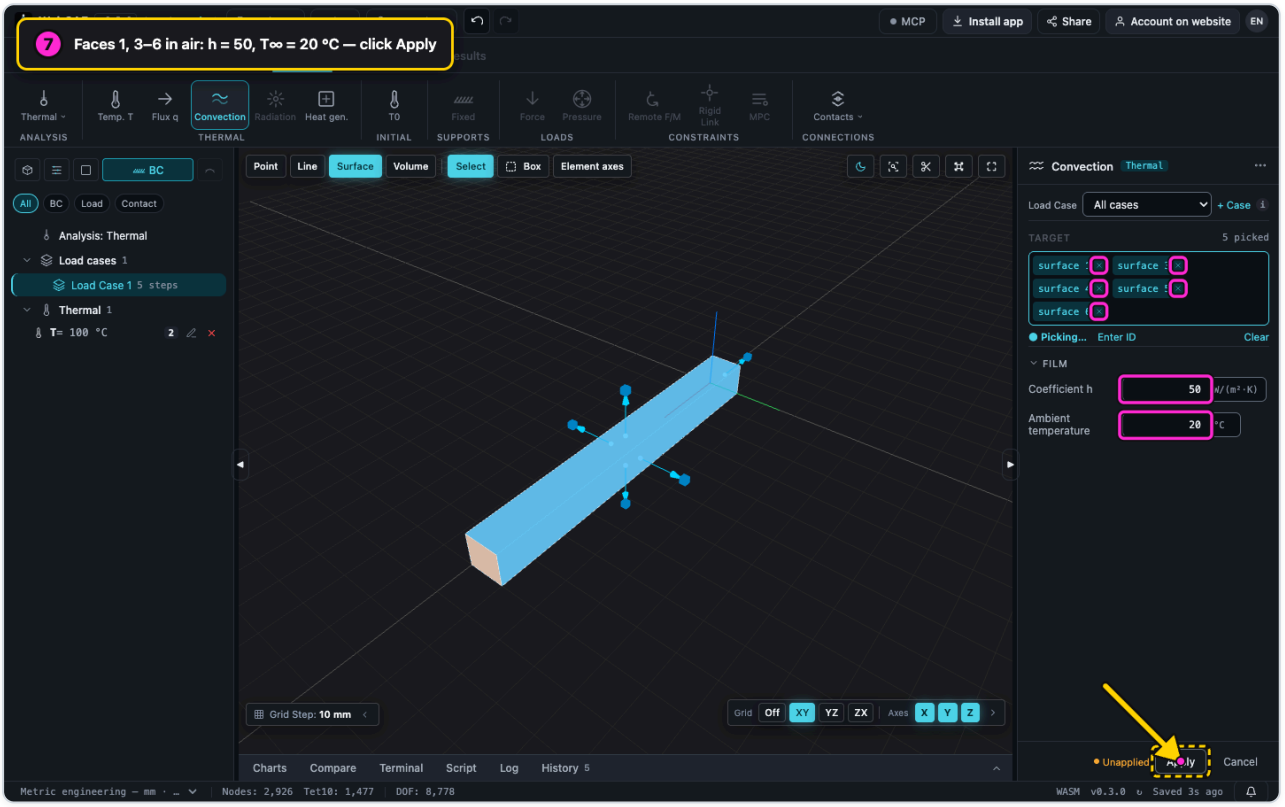
WHY The root of the fin is attached to a hot wall held at $T_{\mathrm{base}} = 100\ ^{\circ}\mathrm{C}$. (Which end is the root does not matter for the physics; here the root is the end facing you, and the far end, face 1 at $x = 0$, is the tip.)



STEP 7 Convective cooling on exposed surfaces

Click **Convection** in the ribbon. Set **Coefficient h** = `50` $\mathrm{W/(m^2 \cdot K)}$ and **Ambient temperature** = `20` °C. Click **Enter ID**, type `1, 3, 4, 5, 6` (the four long lateral faces and the far tip face 1) and press **Enter** — the five faces are highlighted. Click **Apply** (the arrow on the frame).

WHY Air carries heat away from the exposed surfaces at $T_\infty = 20\ ^\circ\mathrm{C}$ by Newton's law of cooling $q = h(T - T_\infty)$.

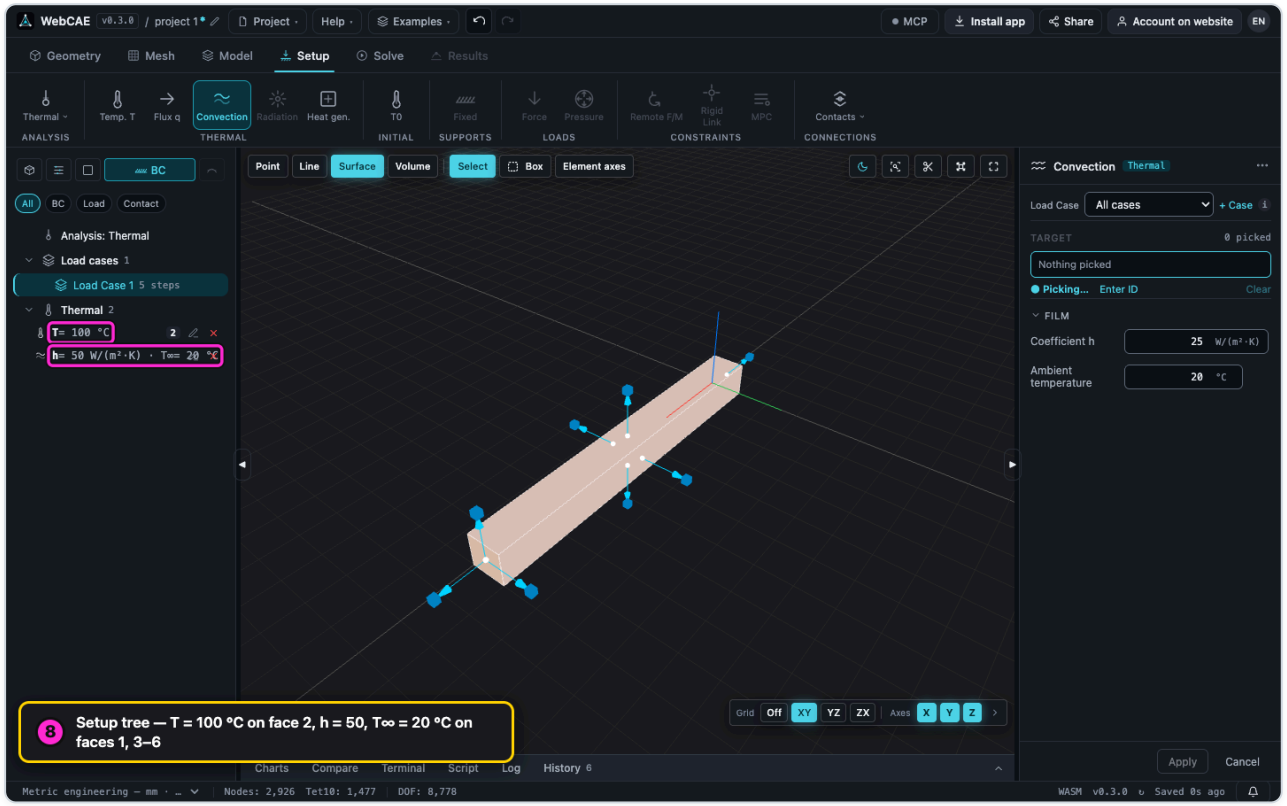


STEP 8

Check the conditions in the Setup tree

Look at the Setup tree on the left: it lists **T = 100 °C** (face 2) and **h = 50 W/(m²·K), T∞ = 20 °C** (faces 1, 3–6). Every one of the six faces has a condition; nothing is left insulated by accident.

WHY A missing condition means an insulated face and a different physics, with no error message.

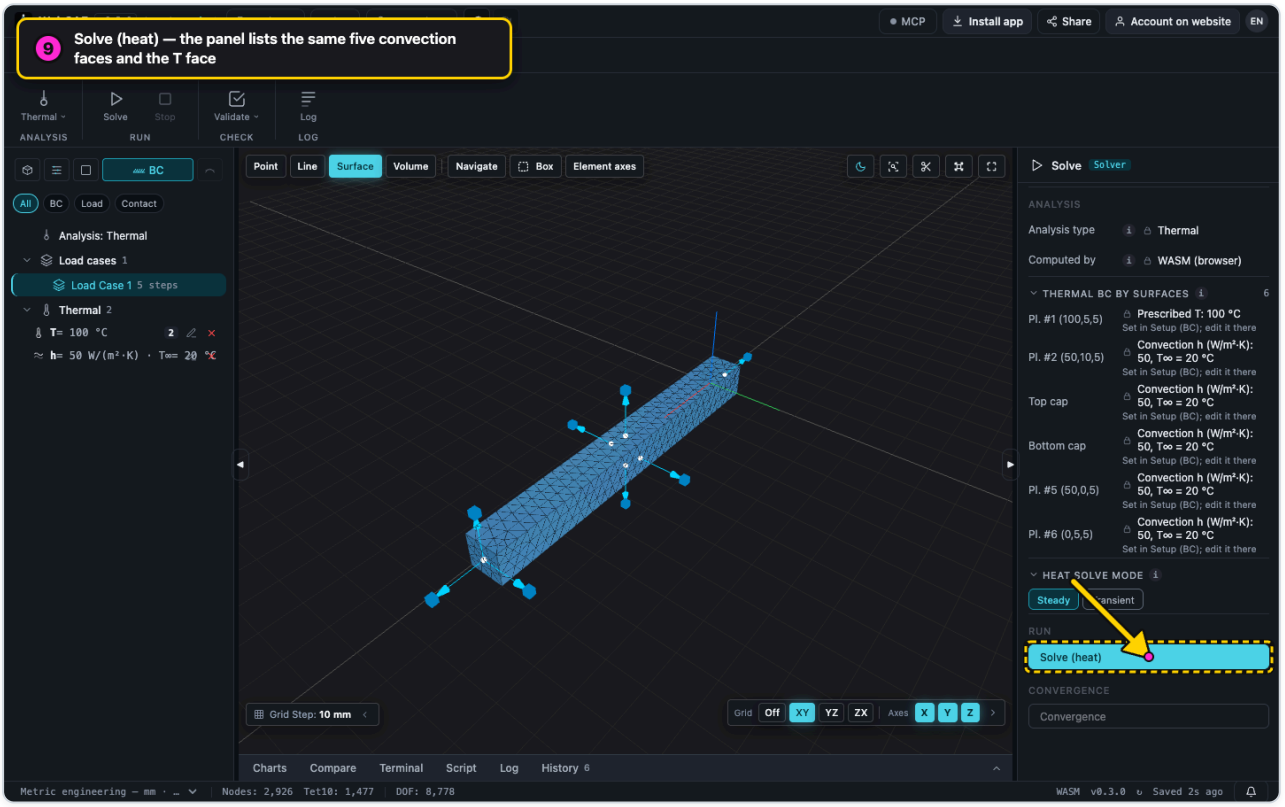


STEP 9

Solve steady heat conduction

Switch to **Solve** in the ribbon. The **Thermal BC by surfaces** list shows the same conditions that you set in Setup (each row is labelled *Set in Setup (BC)*). Its rows are the mesh surfaces named by position — **Pl. #1 (100,5,5)** is the x = 100 end, **Pl. #6 (0,5,5)** the x = 0 end — so the numbers differ from the tree numbers of the faces; the conditions are the same. Click **Solve (heat)** (the arrow on the frame). The solve takes about a second, and the app then switches to **Results** by itself.

WHY The solver assembles the conductivity matrix and the convective boundary terms and finds the nodal temperature field $T(\mathbf{x})$.

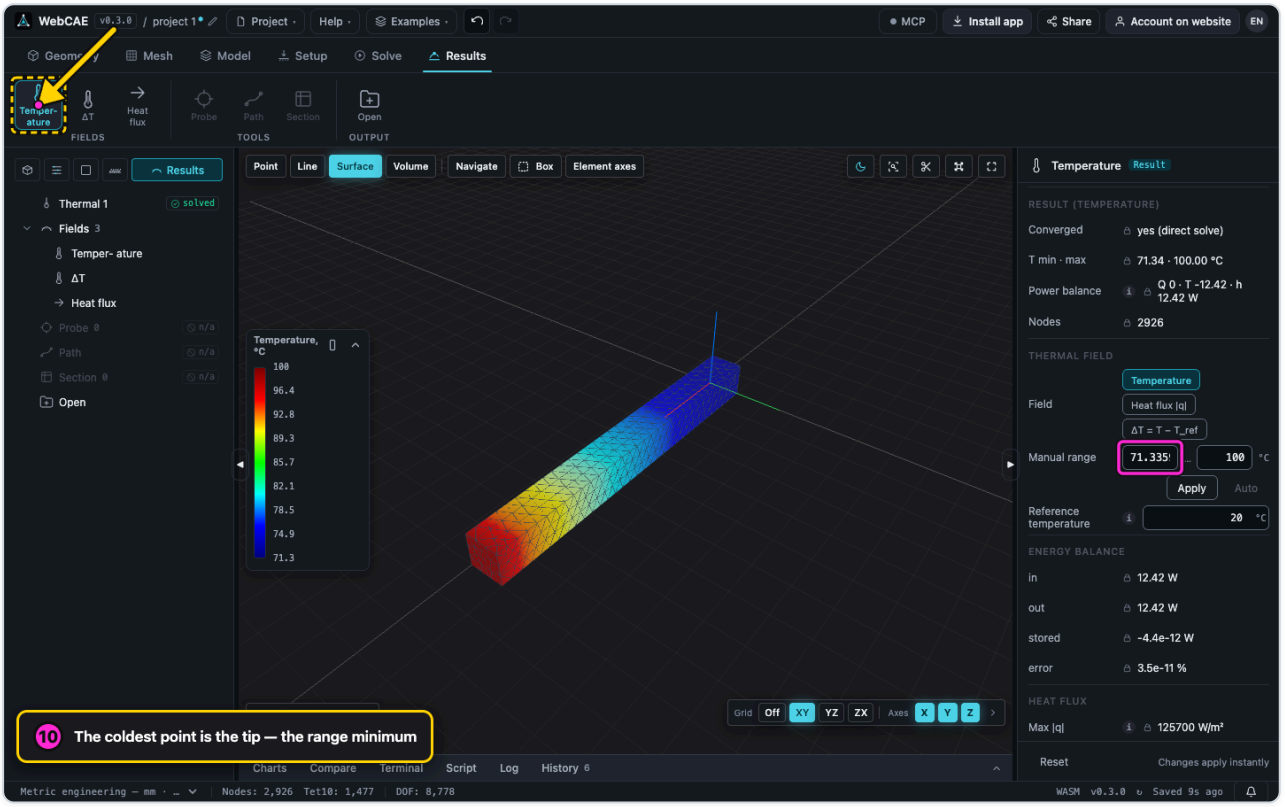


STEP 10

Inspect the minimum temperature at the tip

The app has already opened **Results** with the **Temperature** field (if you are still on Solve, switch to **Results** in the ribbon and click **Temperature**; the tile name wraps as *Temper-ature* — that is only the line break). The minimum of the colour range (the framed field) is the coldest point — the tip.

> **WHY** Heat flows from the base ($x = 0$) to the tip ($x = 100\ \mathrm{mm}$) and leaves through the sides, so the tip is the coldest point ($T_{\min} = T_{\mathrm{tip}} \approx 71.3\ ^{\circ}\mathrm{C}$).

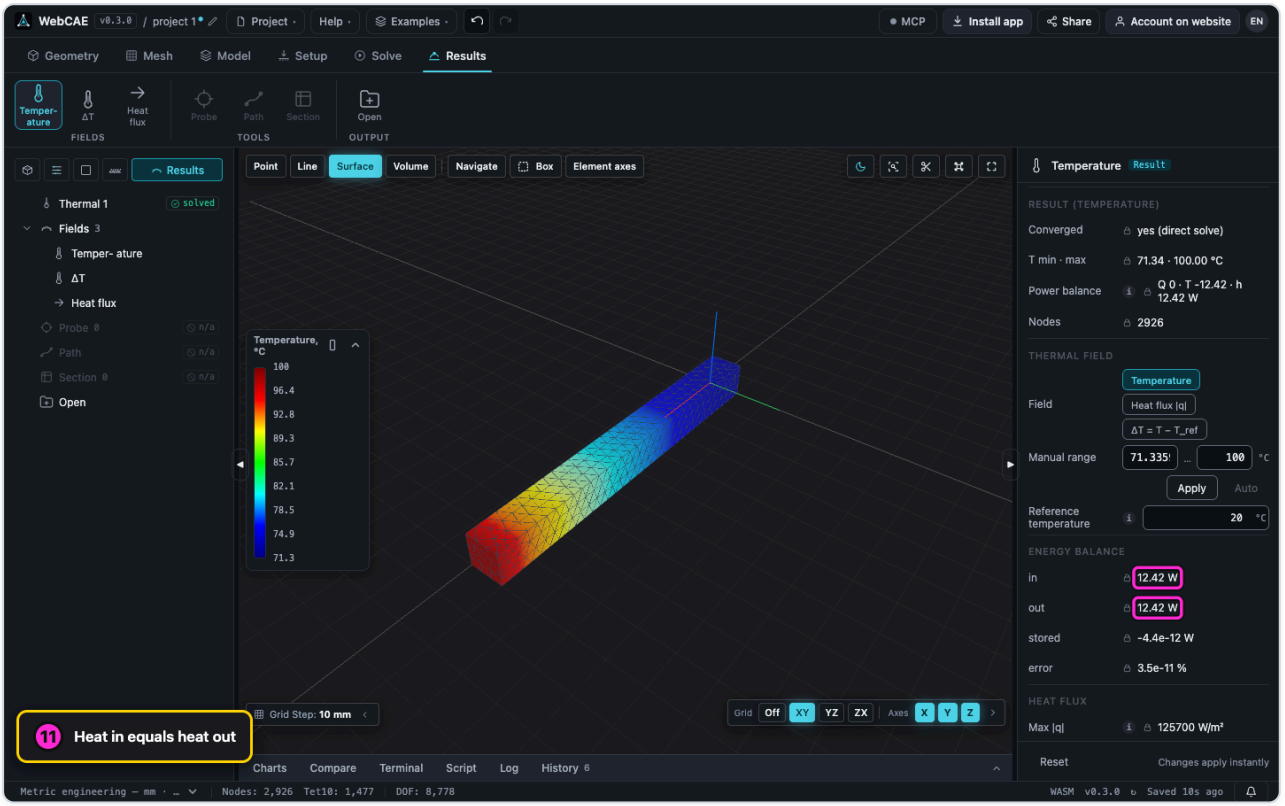


STEP 11

Energy balance: heat in equals heat out

In the right panel find the energy balance and read **Heat in** (q_{in}) and **Heat out** (q_{out}).

WHY In steady state without internal heat sources the heat entering through the base equals the heat leaving through the surfaces ($\sum Q_{\text{in}} = \sum Q_{\text{out}}$).



VERIFICATION BENCHMARK

Check against the formula

Quantity	Theory (fin equation)	WebCAE web (browser)	WebCAE desktop (native)	Error (web)
Tip temperature T_{tip} (range minimum, Step 10)	$71.37\ ^\circ\text{C}$	$71.34\ ^\circ\text{C}$	$71.34\ ^\circ\text{C}$	0.04 % of the 80 K span
Heat flow through base q_{base} (Heat in, Step 11)	$12.42\ \text{W}$	$12.42\ \text{W}$	$12.42\ \text{W}$	0.00 %
Heat balance $\lvert q_{\text{in}} - q_{\text{out}}\rvert / q_{\text{base}}$	0%	0.00 %	0.00 %	0.00 %
Tet10 elements (not a check)	—	1477	1357	—

ANALYTICAL FORMULA

- Fin parameter $m = \sqrt{\frac{hP}{\kappa A}}$ where perimeter $P = 4 \times 10\ \text{mm} = 0.04\ \text{m}$, area $A = 10 \times 10\ \text{mm}^2 = 10^{-4}\ \text{m}^2$, $\kappa = 205\ \text{W/(m}\cdot\text{K)}$, $h = 50\ \text{W/(m}^2\cdot\text{K)}$:

$$m = \sqrt{\frac{50 \times 0.04}{205 \times 10^{-4}}} = 9.877\ \text{m}^{-1}, \quad mL = 9.877 \times 0.1 = 0.9877$$

- Excess temperature ratio at tip ($x = L$) with convective tip condition:

$$\frac{\theta(L)}{\theta_b} = \frac{1}{\cosh(mL) + \frac{h}{m\kappa}\sinh(mL)} = 0.6421$$

- With $\theta_b = T_{\text{base}} - T_\infty = 100 - 20 = 80\ \text{K}$:

$$\theta_{\text{tip}} = 80 \times 0.6421 = 51.37\ \text{K} \implies T_{\text{tip}} = 20 + 51.37 = 71.37\ ^\circ\text{C}$$

- Total heat dissipated through the fin:

$$q_{\text{base}} = \sqrt{hP\kappa A}\,\theta_b \frac{\sinh(mL) + \frac{h}{m\kappa}\cosh(mL)}{\cosh(mL) + \frac{h}{m\kappa}\sinh(mL)} = 12.42\ \text{W}$$

- Biot number validation: $\text{Bi} = \frac{h(t/2)}{\kappa} = \frac{50 \times 0.005}{205} = 1.22 \times 10^{-3} \ll 0.1$, confirming the 1D fin approximation matches the full 3D finite element simulation within $< 1\%$. Note on the error column: the tip-temperature error is taken relative to the temperature span $\theta_b = 80$ K, because a percentage of a Celsius value depends on the arbitrary zero of the scale.

TROUBLESHOOTING & NOTES

Common mistakes

- **Forgetting to enable Thermal conductivity:** In the Material panel, the "Thermal conductivity" property group must be switched on so that κ is included in the model state and solver input.
- **Applying convection to the base:** The root face (face 2) is held at a fixed temperature of $100\ ^{\circ}\mathrm{C}$. Convection goes only on the remaining five surfaces (1, 3, 4, 5, 6); the **Thermal BC by surfaces** list on the Solve tab must show no insulated (adiabatic) face.
- **Using axis directions instead of entity numbers:** Entities in WebCAE are strictly addressed by model tree surface numbers (here face 2 for the temperature and faces 1, 3, 4, 5, 6 for the convection).

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#7 — CAD import and multi-surface boundary conditions (F07)** to import a STEP part, heal it and fix several faces at once.

WebCAE 101

LESSON 7 · F07

CAD import and multi-surface boundary conditions

Goal Import a STEP part, check that it arrived whole, run Repair, find faces by number and fix three faces with one boundary condition.

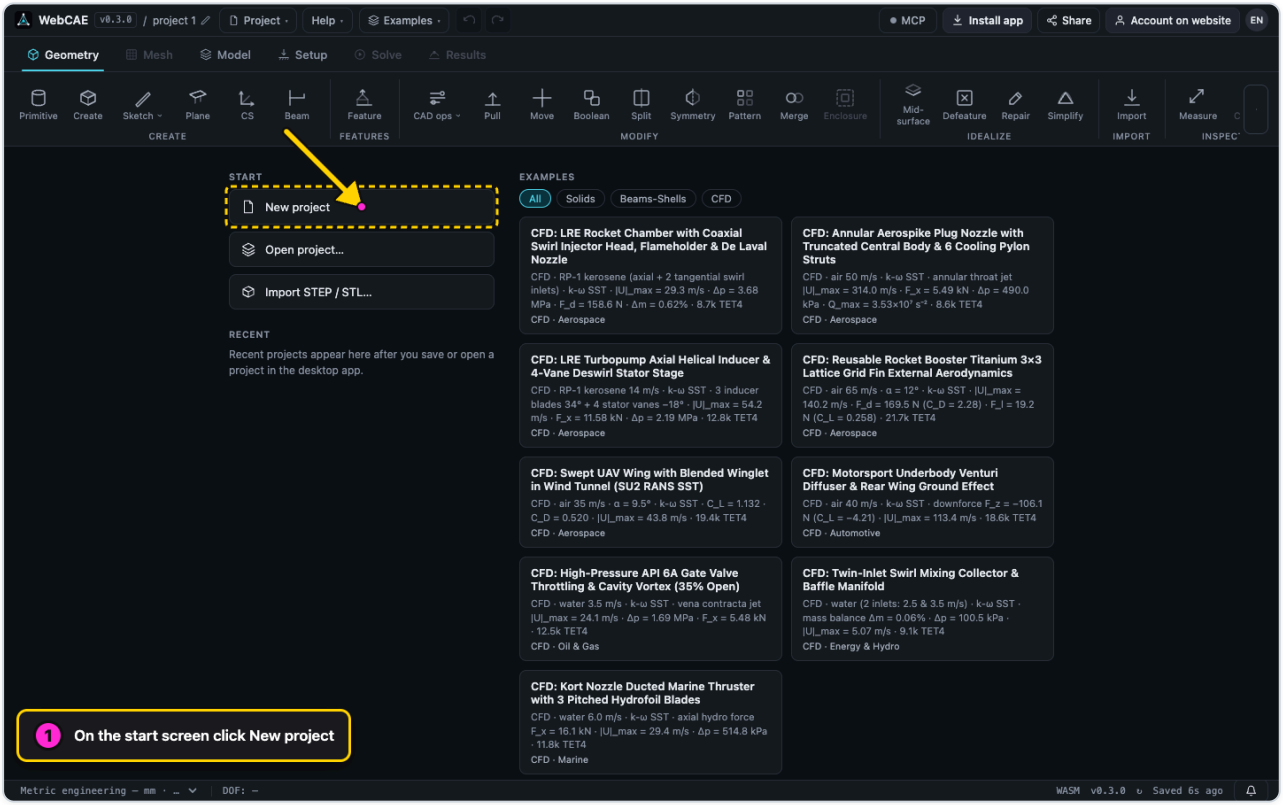
Outcome 15 numbered faces, a quadratic solid mesh (about 1885 Tet10 in the browser, 2310 in the desktop app) and one Fixed support on three faces (3, 10, 11).

5–6 min | web (WASM) with the desktop (native) column in the table | Run: 2026-11-04-32ffaed2 / F07

STEP 1 Start a new project

Open WebCAE. The centre of the window shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and the gallery of **EXAMPLES**). Click **New project** (the arrow on the frame): the screen closes and the empty model opens. If you already see the empty model and no Start screen, skip this step.

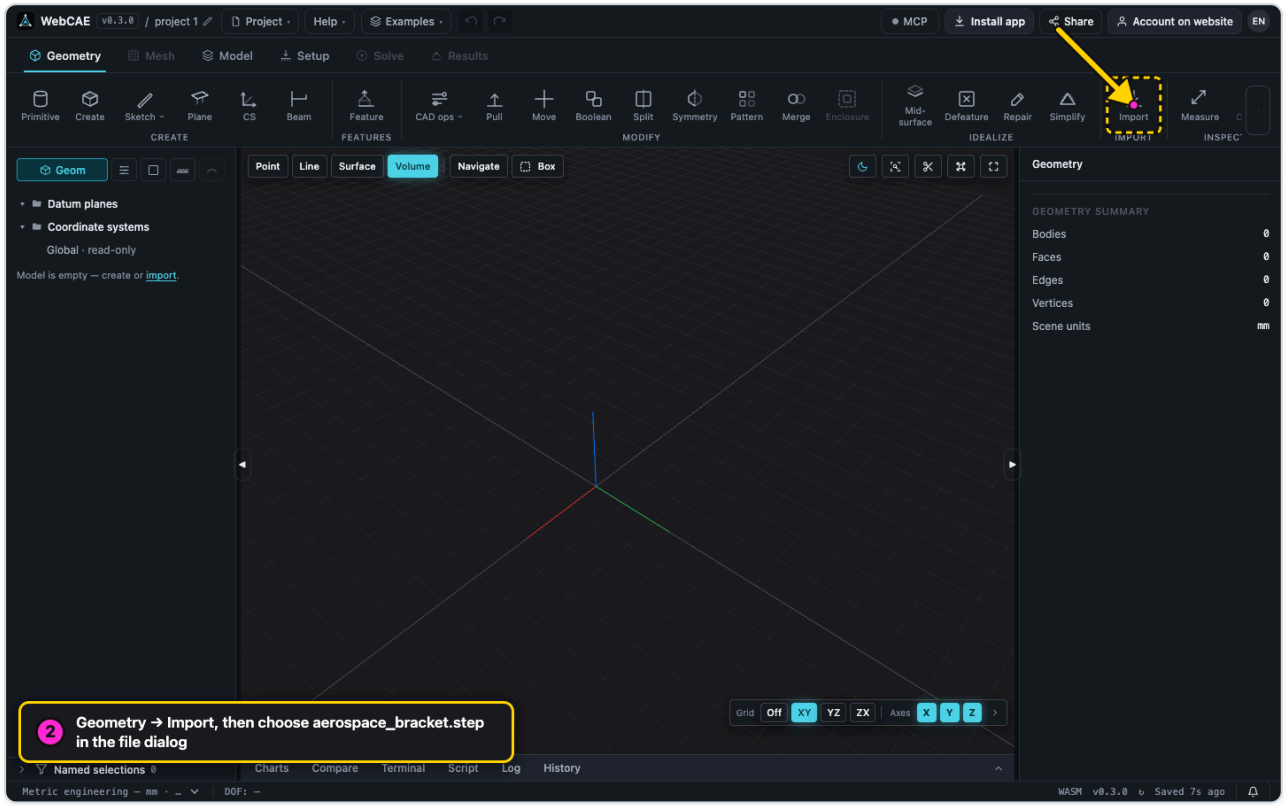
WHY The Start screen covers the viewport and the tree until you choose what to open; this lesson starts from an empty model and brings its own part in the next step.



STEP 2 Import the STEP file

In the **Geometry** tab click **Import** in the ribbon (the arrow on the frame). The file dialog opens: choose `aerospace_bracket.step`. (You can also drag the file onto the viewport.)

> **WHY** The part is read by the CAD kernel in the app; no upload to a server.

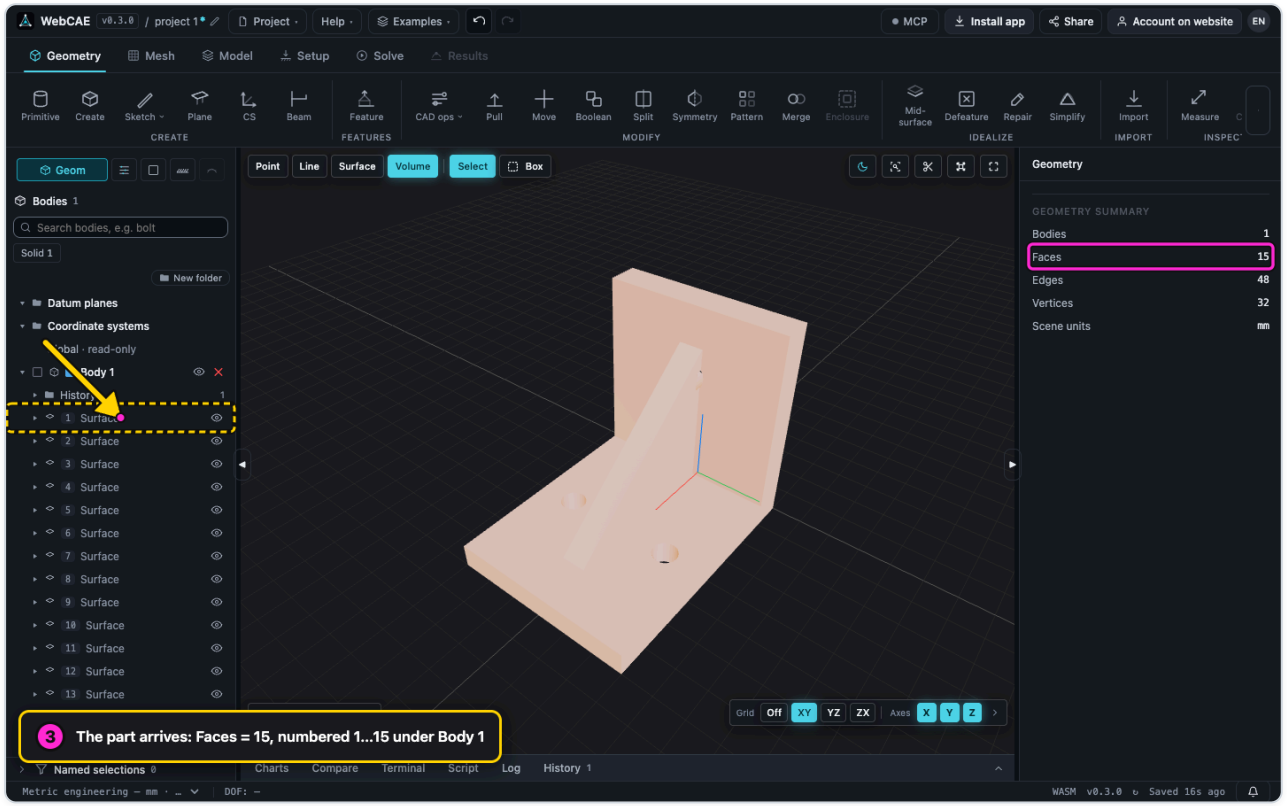


STEP 3

The part arrives with numbered faces

The bracket appears in the viewport. Expand **Body 1** in the tree: its faces are numbered **1 Surface … 15 Surface**, and the Geometry summary on the right reads **Faces 15**, **Edges 48**, **Vertices 32**.

> **WHY** Every boundary condition from now on is addressed by these numbers.

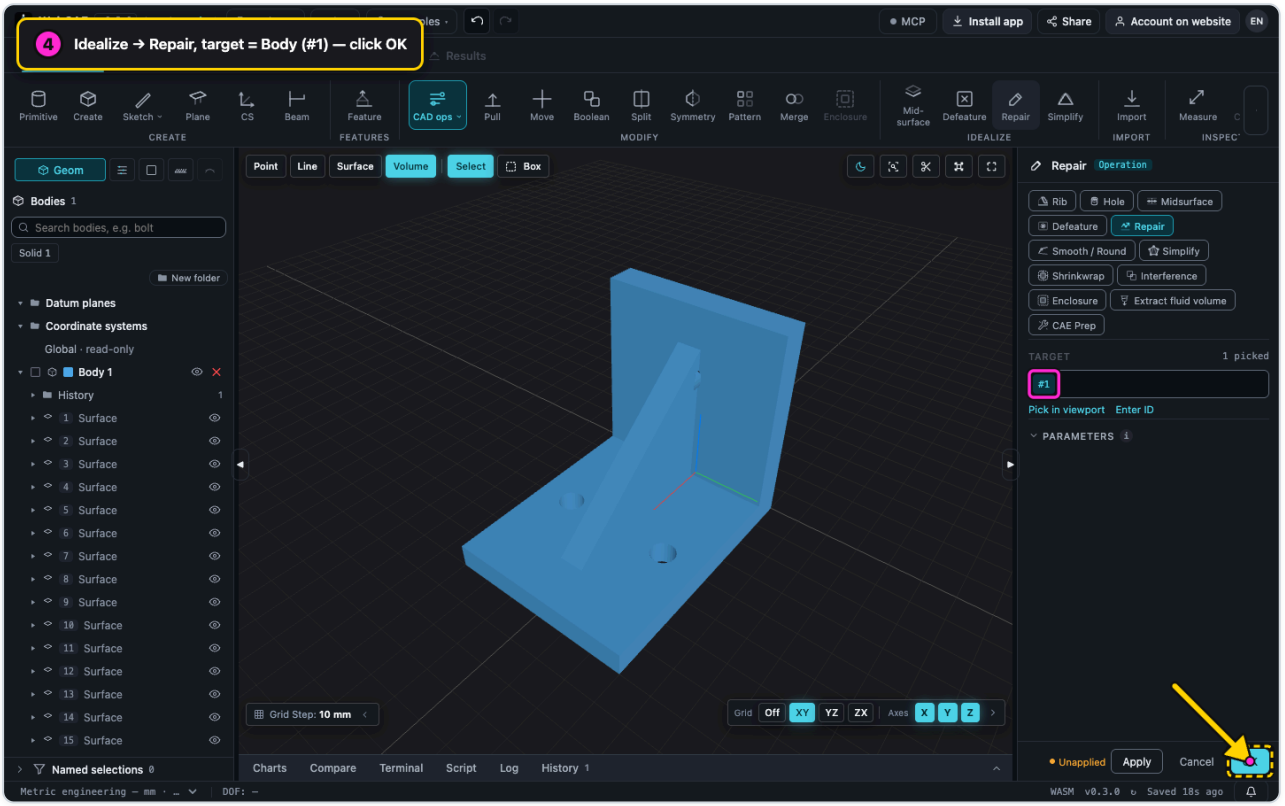


STEP 4

Repair

Click **Repair** in the **IDEALIZE** group of the Geometry ribbon (this tile is called *Repair*; there is no tile called *Prepare*). The target is the body (**#1**). Click **OK** (the arrow on the frame): OK applies the operation and closes the panel (do not press Apply and then OK — that runs Repair twice).

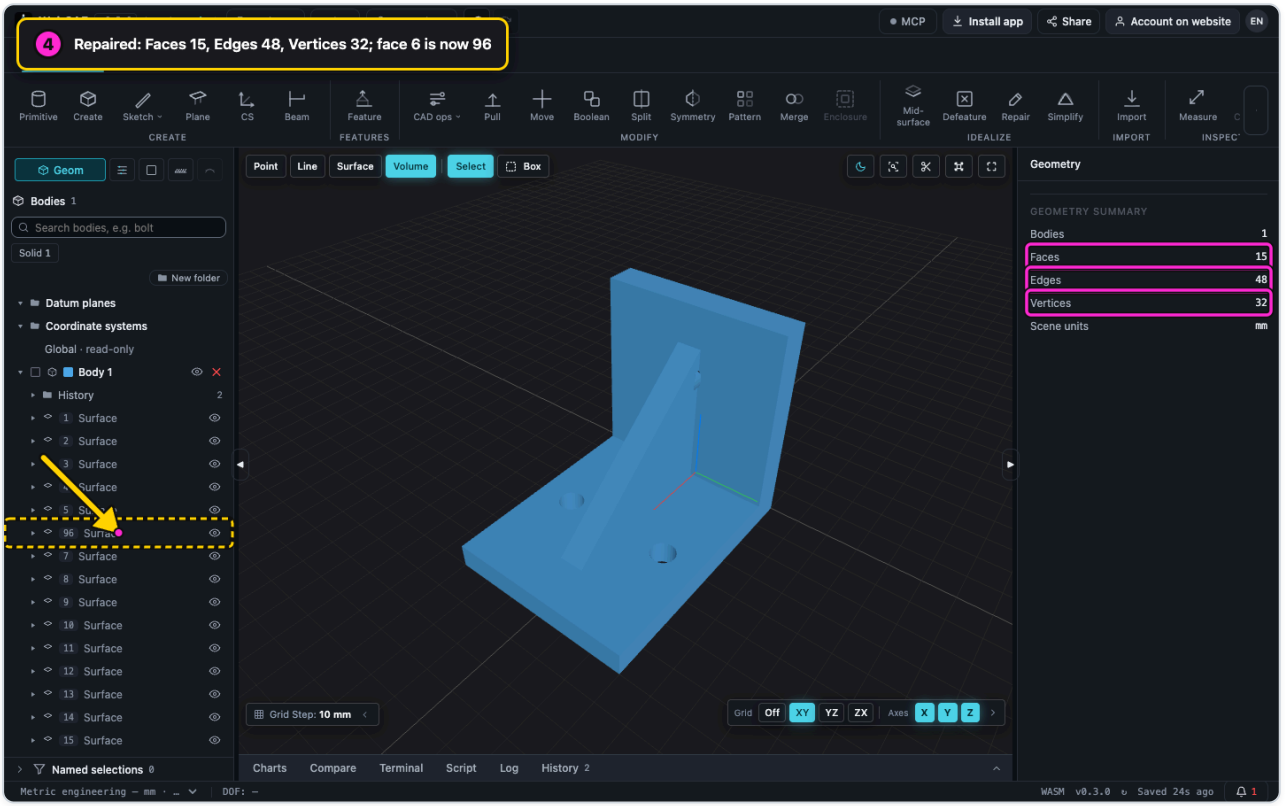
> **WHY** Repair heals small topology defects before meshing.



STEP 5 Read the result of Repair

The Geometry summary after Repair: **Faces 15**, **Edges 48**, **Vertices 32** — the same as before: this part is clean and there was nothing to heal. A notification **Geometry operation applied** appears in the bell (the app gives no separate healing report). One thing did change: the face that was number **6** is now number **96**. WebCAE never re-uses a number, and Repair rebuilt that face, so it received a fresh one; every other face keeps its number (so 3, 10 and 11 below are the same as before Repair).

> **WHY** Numbers belong to the document history, they are not re-counted from the geometry: a number in a script or a boundary condition keeps pointing to the same face.

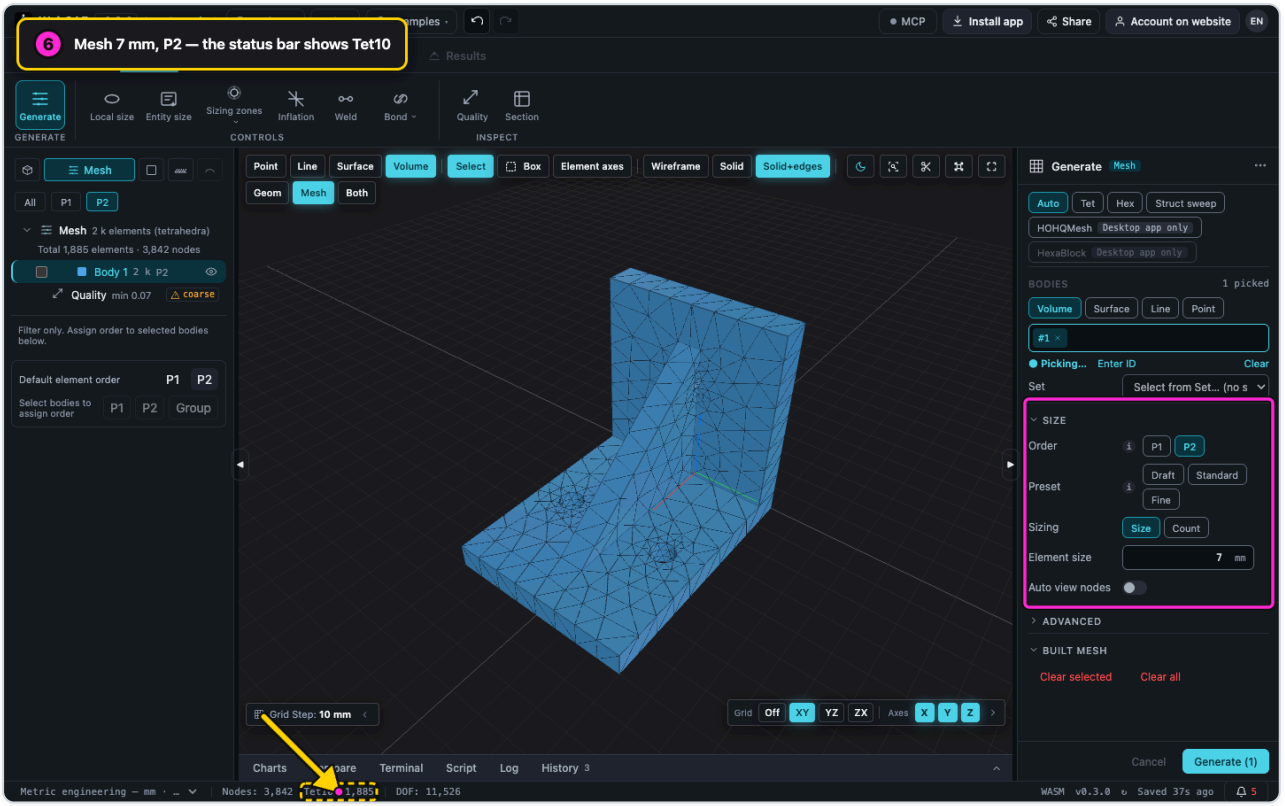


STEP 6

Mesh the part

Open the **Mesh** tab, click **Generate**, choose **P2** (quadratic) and set the element size to `7` mm, then click **Generate**. The frame is taken after the mesh is built: the status bar shows **Tet10: 1885** in the browser (the desktop app builds a slightly different mesh: 2310 elements — the two meshers differ).

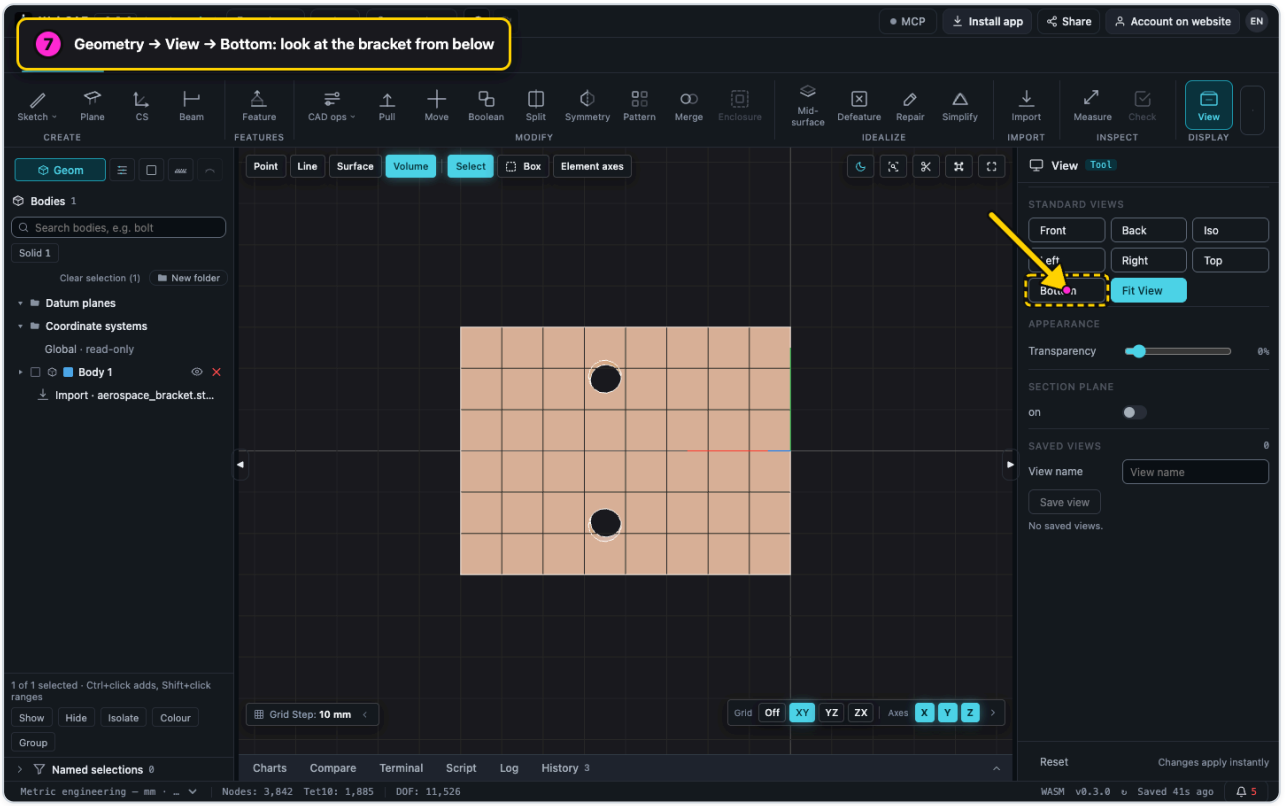
> **WHY** 7 mm resolves the fillets and the holes with a few thousand quadratic tetrahedra.



STEP 7 Look at the part from below

Go back to the **Geometry** tab, click **View** (the last tile of the ribbon, the **DISPLAY** group: it is hidden at the right edge on a 1440 px window — click the **2 more groups** button at the right end of the ribbon or scroll the ribbon to the right) and press **Bottom** in the panel (the arrow on the frame).

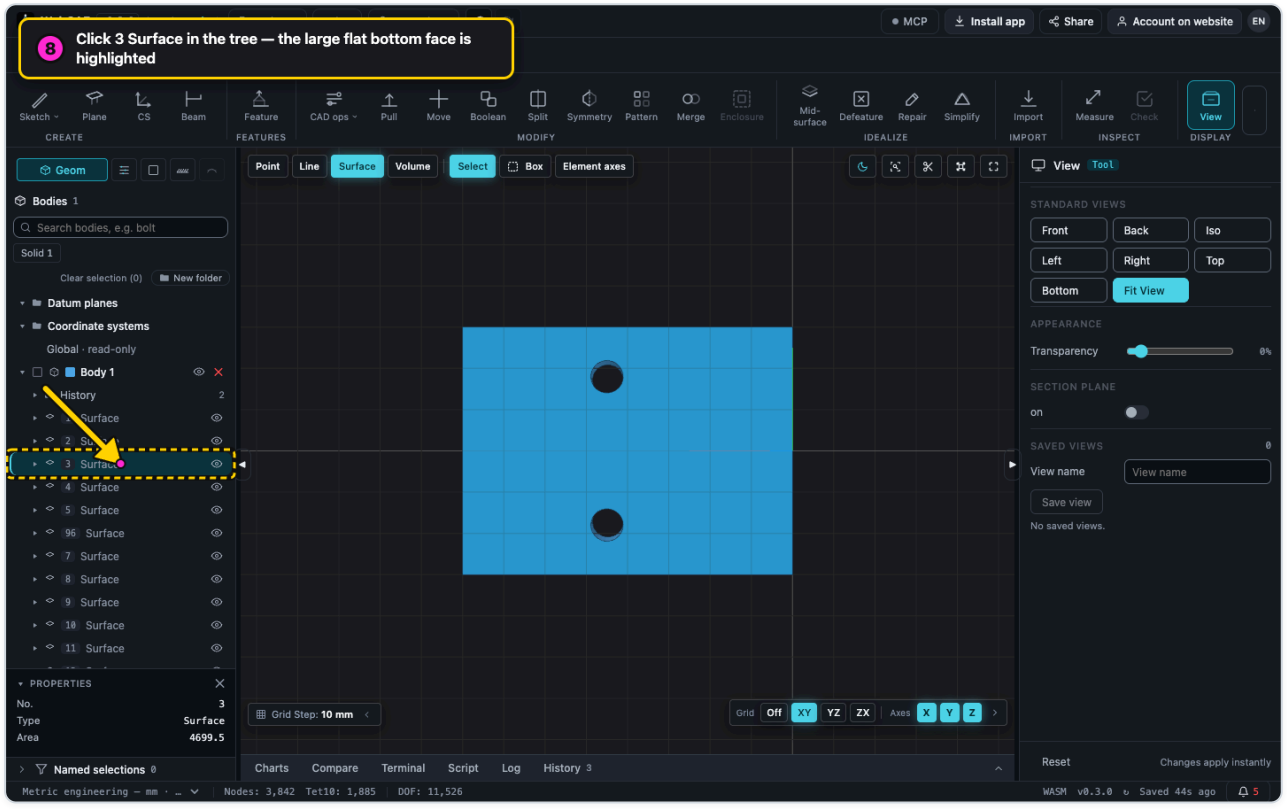
> **WHY** The mounting base is the bottom face of the bracket; from the default view it is hidden.



STEP 8 Find the base face

In the viewport toolbar switch the selection to **Surface**, then click **3 Surface** in the tree. The large flat face — the mounting base — is highlighted; the **Properties** block under the tree shows **No. 3**, **Type Surface** and **Area 4699.5** (mm², one decimal; the script computes 4,699.47 mm² from the CAD data). If **Body 1** is collapsed (the tree folds again when you come back from the **Mesh** tab), expand it with the arrow to the left of **Body 1** first; to see the Properties block you may need to scroll the left panel.

> **WHY** The tree and the viewport use one numbering: click a number to see which face it is.

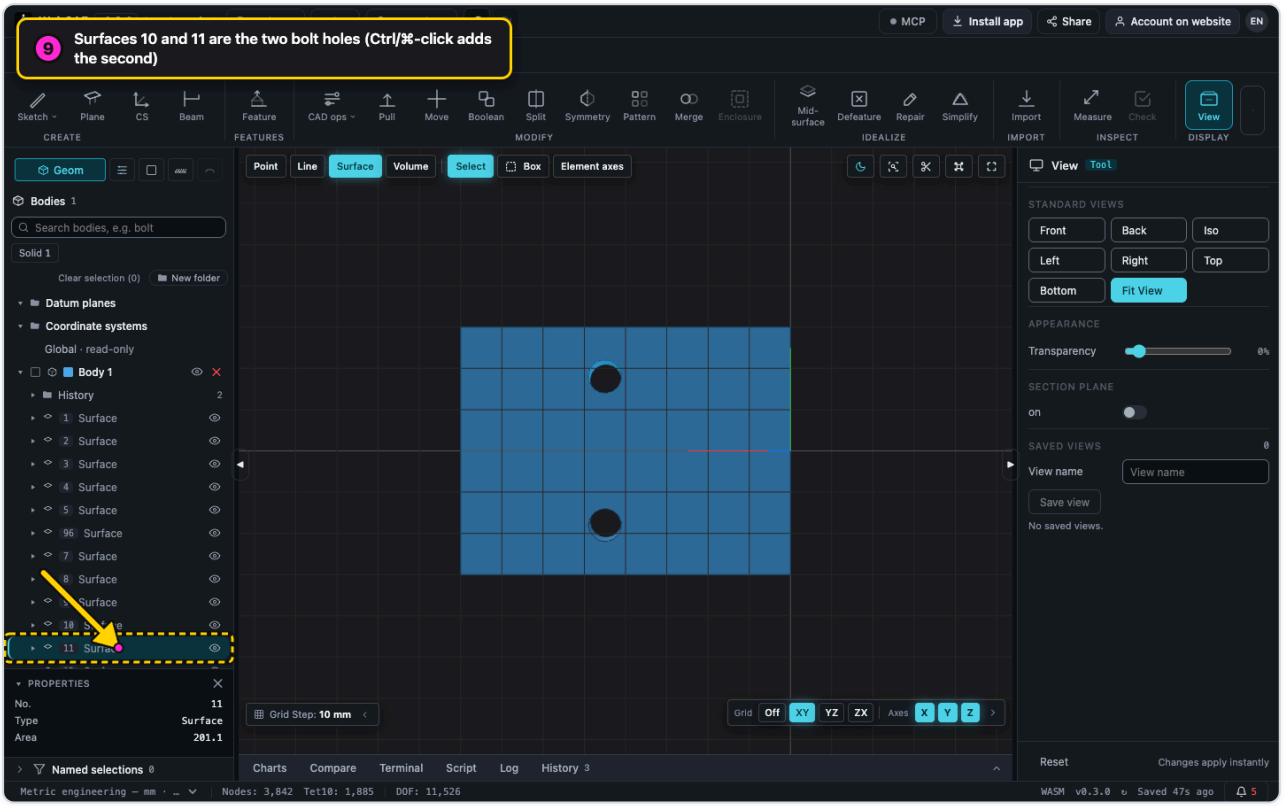


STEP 9

Find the bolt holes

Click **10 Surface**, then Ctrl/⌘-click **11 Surface**: the cylindrical walls of the two bolt holes are highlighted. The **Properties** block under the tree shows the face you clicked last: after the click on 10 it reads **Area 201.1**, after the click on 11 it reads **Area 201.1** (mm²; the script computes 201.06 and 201.06 mm²).

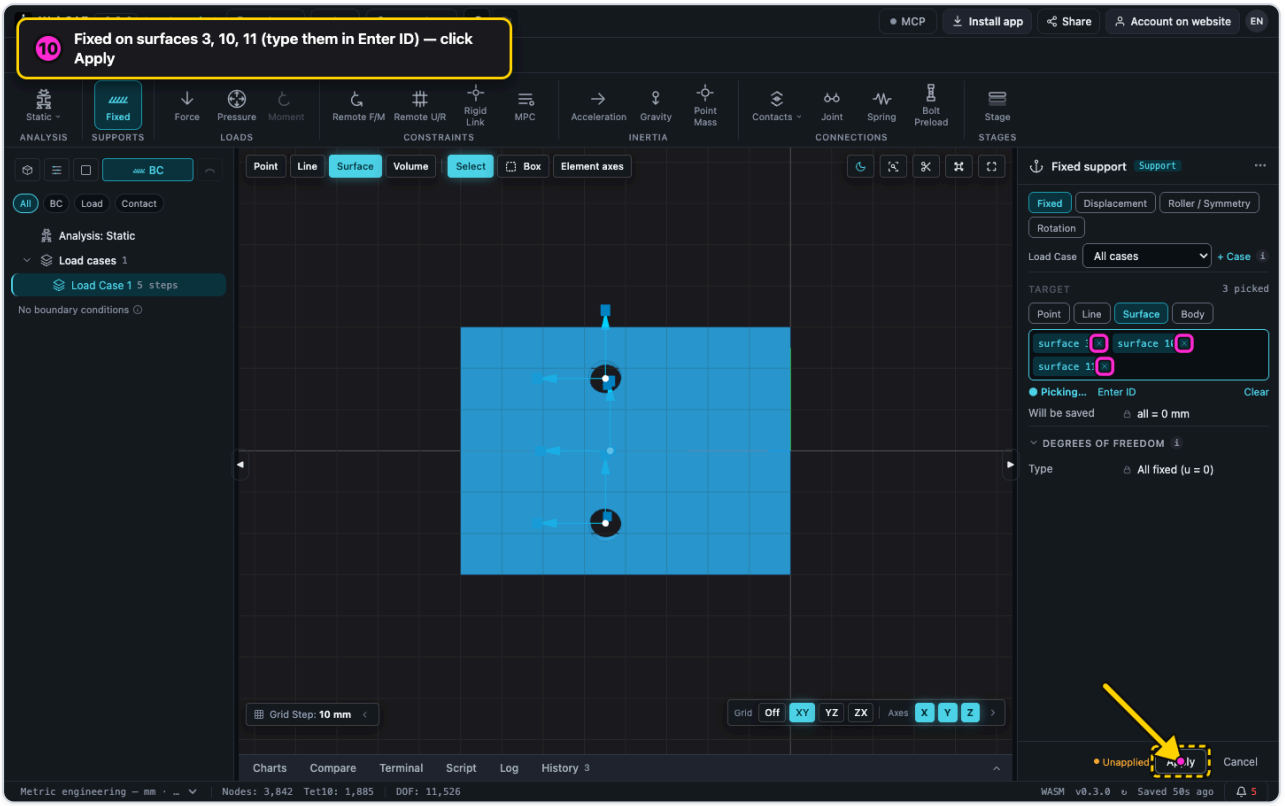
> **WHY** The base and both holes will be fixed together.



STEP 10 Fix the base and the holes with one condition

Open the **Setup** tab, click **Fixed**, click **Enter ID**, type `3 10 11` and press **Enter**: the three faces are picked and highlighted. Click **Apply** (the arrow on the frame).

WHY Several face numbers separated by spaces give one condition on all of them.

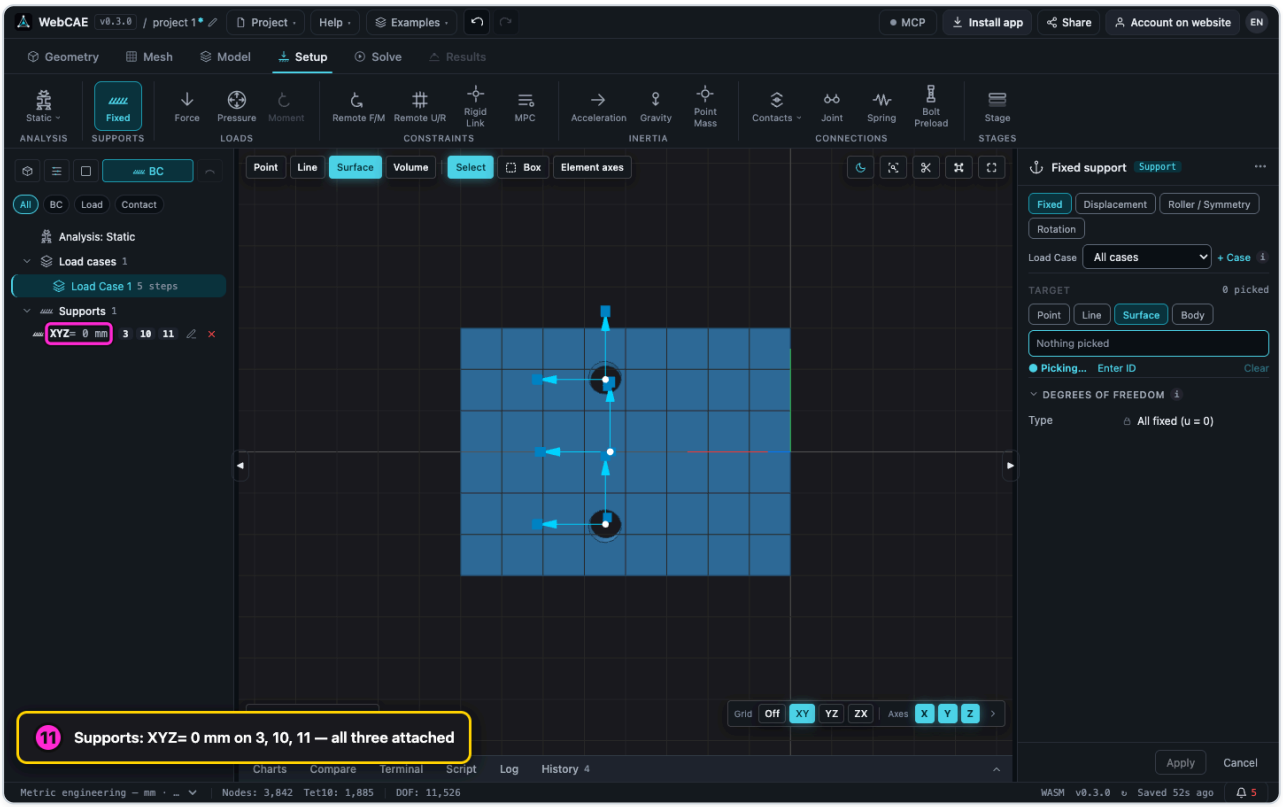


STEP 11

Check the Setup tree

The Setup tree now lists, under **Supports**, one entry **XYZ= 0 mm** (all three displacements fixed at zero) with the targets **3 10 11**; the entry is not marked as orphaned.

WHY A boundary condition that lost its face would silently change the physics; the tree is the place to see that all of them are attached.



VERIFICATION BENCHMARK

Check against the CAD facts

Quantity	CAD reference	WebCAE web (browser)	WebCAE desktop (native)	Where you see it
Number of B-Rep faces	15	15	15	Geometry summary, Step 3
Body volume, mm³	83 009.6 (CAD data sheet)	83,024.44	83,024.44	computed by the script from the boundary mesh (not shown in the UI)
Base face area, mm²	4 699.5	4,699.47 (the Properties block shows 4699.5)	4699 (4 significant digits)	Properties block under the tree, Step 8
Tet10 elements, h = 7 mm (not a check)	—	1885	2310	status bar, Step 6
Mesh volume vs CAD volume $\lvert\Delta V\rvert$	≤ 5 %	0.10 %	0.10 %	computed by the script (not shown in the UI)
Fixed targets	3	3	3	Setup tree, Step 11

TROUBLESHOOTING & NOTES

Common mistakes

- **Typing axis names instead of face numbers:** conditions are addressed by the tree numbers (`3 10 11`), not by directions such as "bottom".
- **Pressing Apply and then OK in Repair:** the operation runs twice (History grows by two). Press OK once.
- **Taking the numbers from another model:** the numbers 3, 10 and 11 belong to this file; with another part click the faces in the tree and read them off.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#8 — Two bodies, one load path: assembly contact (F08)** to place two bodies, bond their interface and check the stiffness of the pair.

WebCAE 101

LESSON 8 · F08

Two bodies, one load path: assembly contact

Goal Model a two-part assembly touching along a planar interface, bond the interface (exact MPC) and check that the pair is as stiff as a single 200 mm bar.

Outcome Axial deflection $\delta = 0.100\ \mathrm{mm}$ under a $10\,000\ \mathrm{N}$ pull by the bar formula $\delta = \frac{PL}{AE}$; WebCAE web (browser) gives 0.102914 mm (+2.9 %), WebCAE desktop (native) gives 0.09985 mm (-0.15 %); both inside the 5 % tolerance.

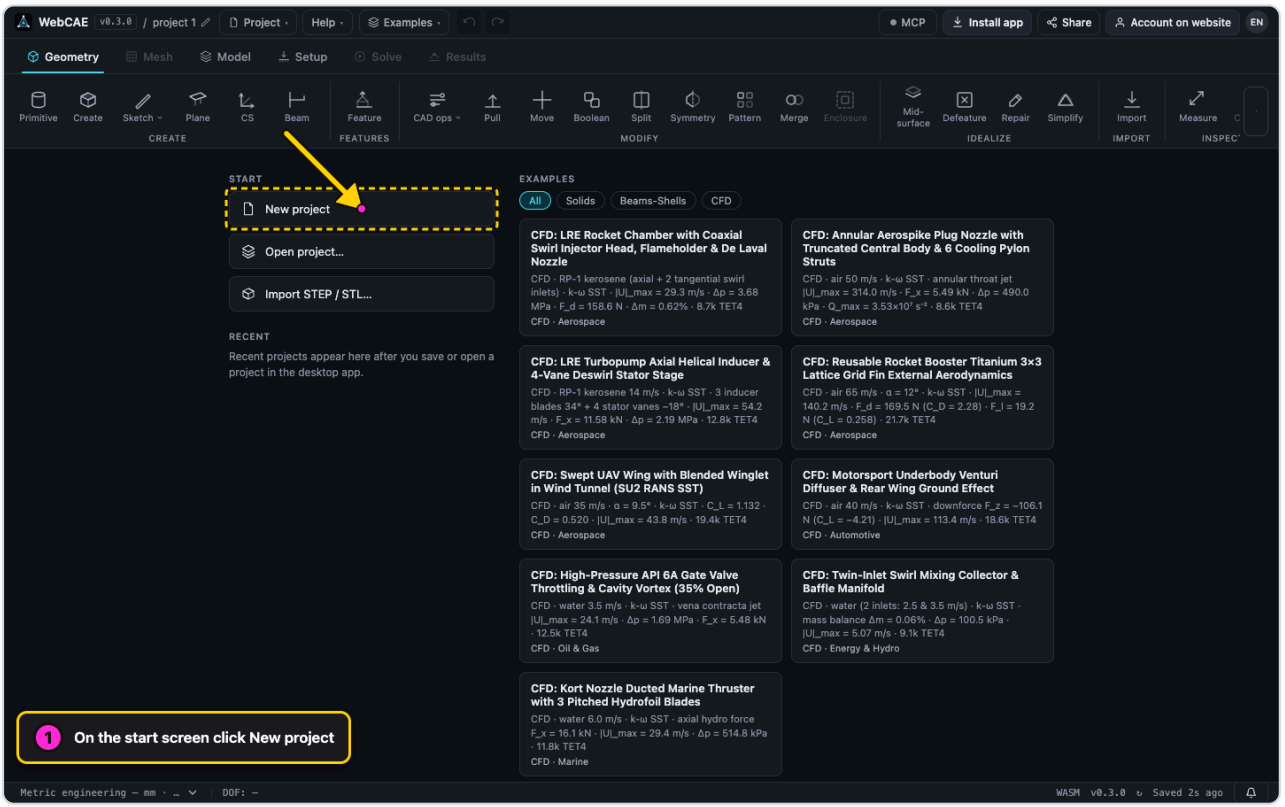
5–6 min | web (WASM) and desktop (native), shown in separate columns

Run: 2026-11-04-32ffaed2 / F08

STEP 1 Start a new project

Open WebCAE. The centre of the window shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and the gallery of **EXAMPLES**). Click **New project** (the arrow on the frame): the screen closes and the empty model opens. If you already see the empty model and no Start screen, skip this step.

WHY The Start screen covers the viewport and the tree until you choose what to open; this lesson builds its two bars from scratch.

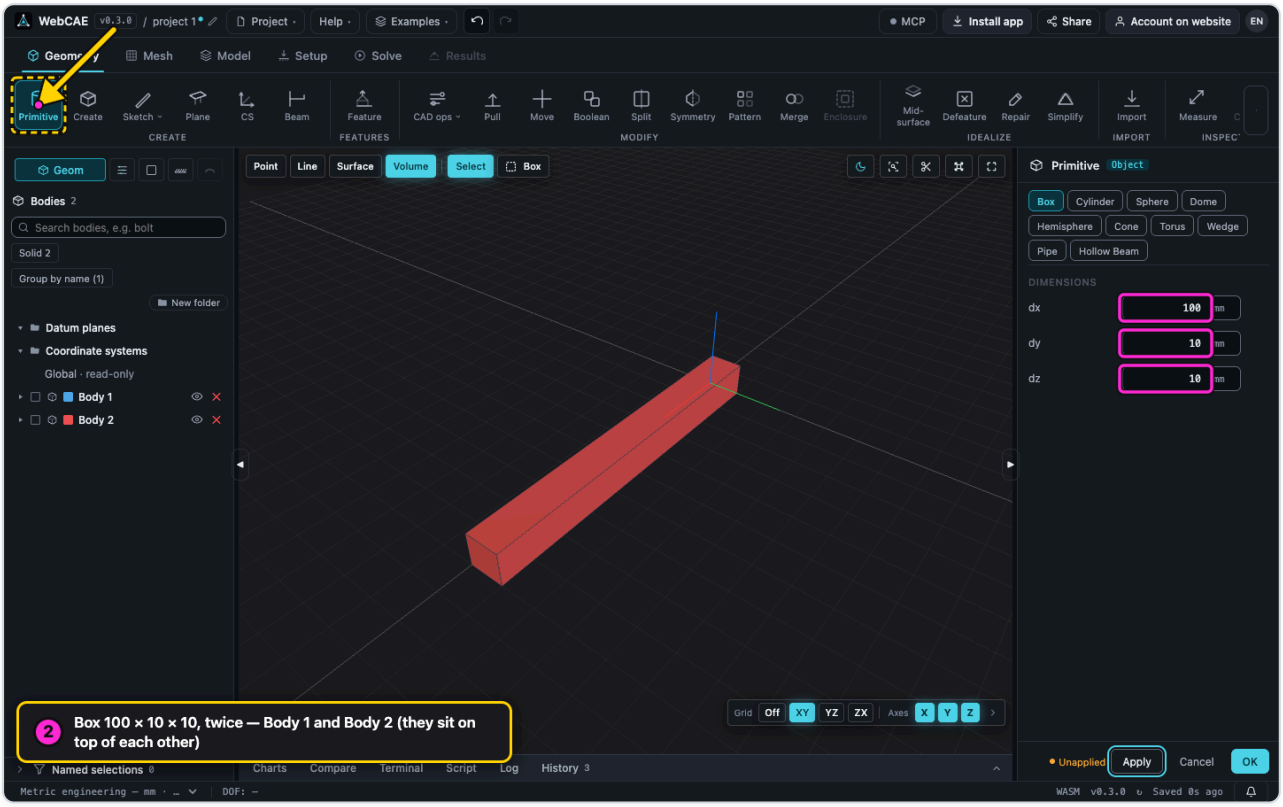


STEP 2

Two identical bars

In the **Geometry** tab open **Primitive**, choose **Box**, set **dx** = `100`, **dy** = `10`, **dz** = `10` mm, click **Apply** and **OK**. Repeat once more for the second bar (a second **Apply** in the same open panel does not add a second body, so close the panel with **OK** and open **Primitive** again). The frame is taken after the second **Apply**: the tree lists **Body 1** and **Body 2**, but you see one bar — both are created at the origin, on top of each other.

WHY Two separate bodies, each with its own faces and numbering.

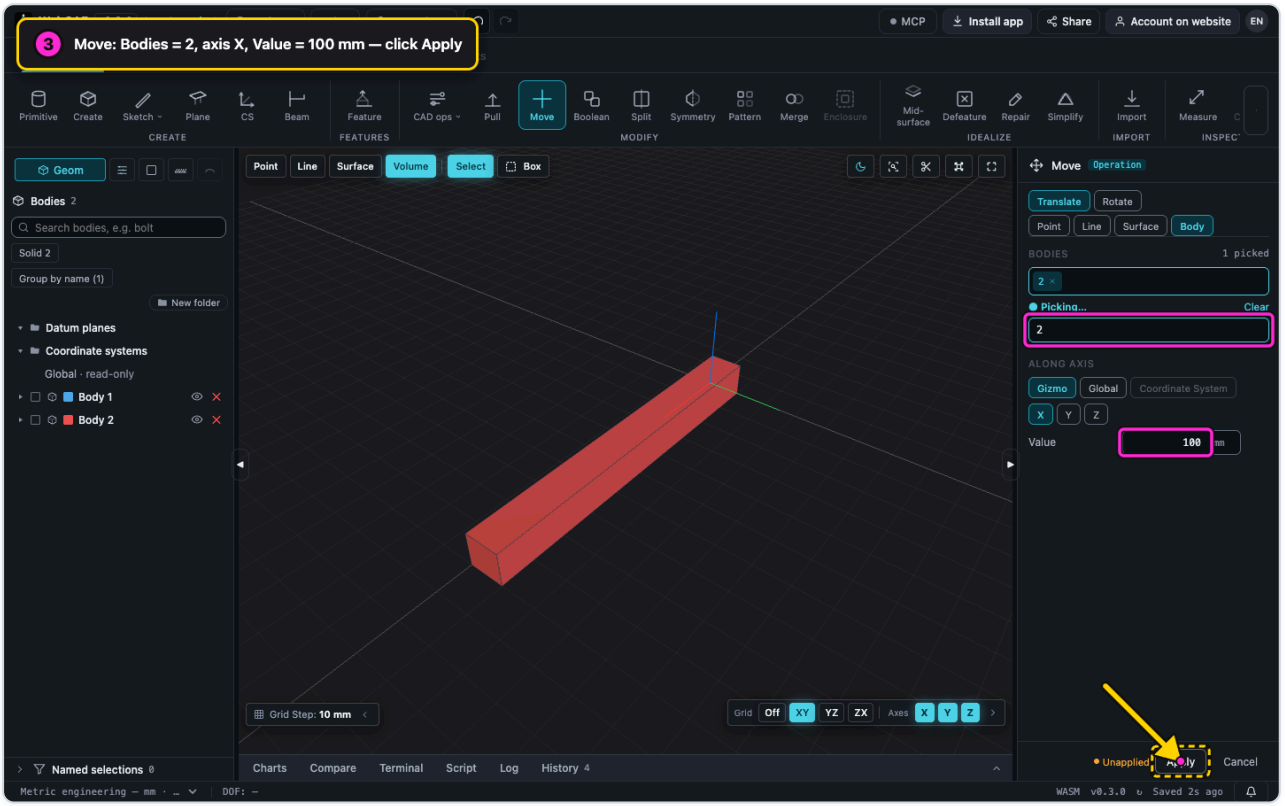


STEP 3

Move Body 2 by 100 mm along X

Click **Move**. Leave **Translate** and **Body**, type `2` into **Bodies** and press **Enter**, press the **X** axis button and type `100` into **Value** (mm). The frame shows the filled panel; click **Apply** (the arrow on the frame).

WHY Body 1 spans $x \in [0, 100]$; Body 2 will span $[100, 200]$ — the two end faces coincide at $x = 100\ \text{mm}$.

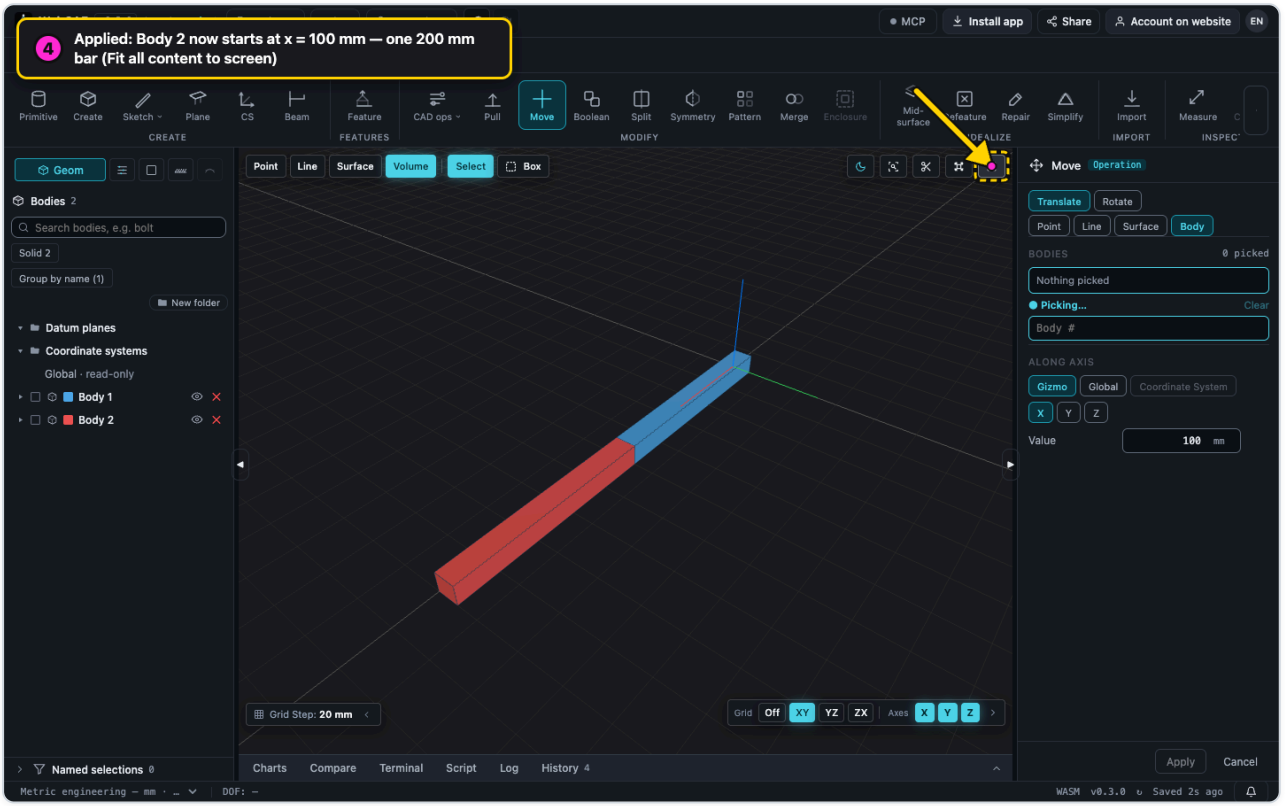


STEP 4

Check that the bars touch

After **Apply** the second bar moves; click **Fit all content to screen** (the button at the top right of the viewport, the arrow) to see both: Body 1 (blue) and Body 2 (red) now form one 200 mm bar. In the model state Body 2 starts at x = 100 mm.

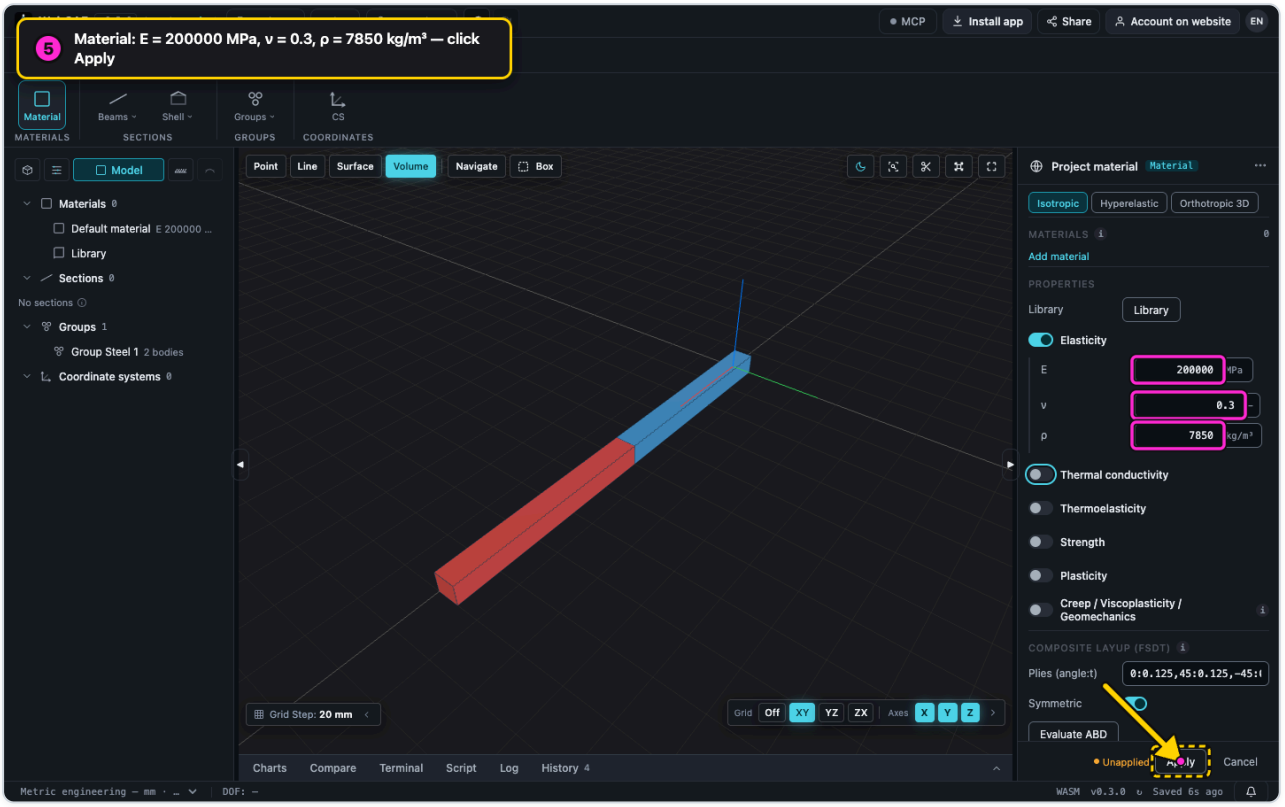
> **WHY** The faces must coincide within the contact tolerance (0.2 mm by default) to be detected as touching.



STEP 5 Material

Open the **Model** tab and **Material**. Set $E = 200\,000$ MPa, $\nu = 0.3$, $\rho = 7\,850$ kg/m^3 and click **Apply** (the arrow on the frame). One material applies to both bodies.

WHY Steel; the bar formula needs E.

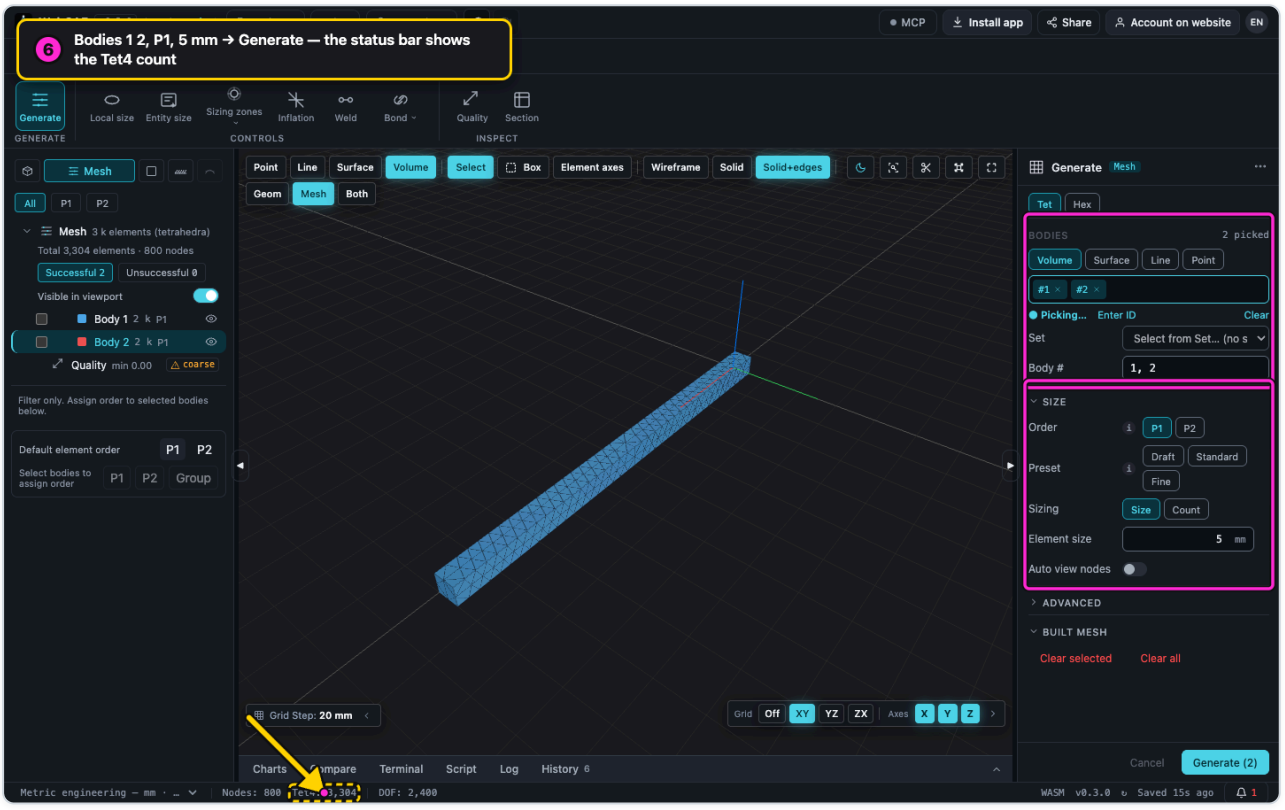


STEP 6

Mesh both bodies

In the **Mesh** tab click **Generate**, pick both bodies (**Enter ID**, type `1 2`, press **Enter**), choose element order **P1** (linear tetrahedra) and set the element size to `5` mm; click **Generate**. The frame is taken after the mesh is built: the status bar shows **Tet4: 3304** in the browser (the desktop app builds a different mesh, 2748 elements).

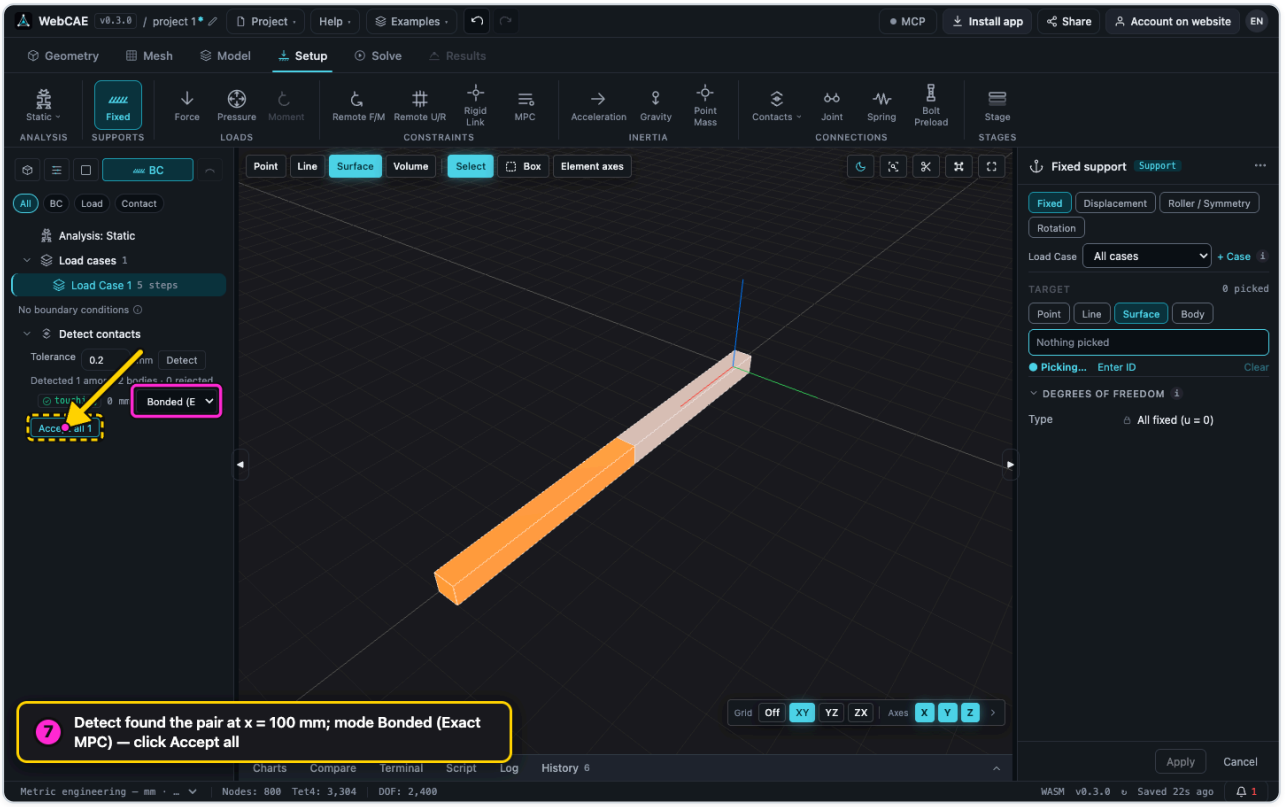
> **WHY** P1 keeps the web solve small and quick; the two bodies are meshed separately, so the nodes on the two sides of the interface do not have to match.



STEP 7 Detect the contact and bond it

In the **Setup** tab click **Detect contacts** in the tree, then **Detect**. The touching pair at $x = 100\ \mathrm{mm}$ is found; in the mode list of the found pair choose **Bonded (Exact MPC)** (framed; the list is narrow and shows the name shortened, *Bonded (E…)*) and click **Accept all 1** (the arrow on the frame; the number on the button is the count of found pairs).

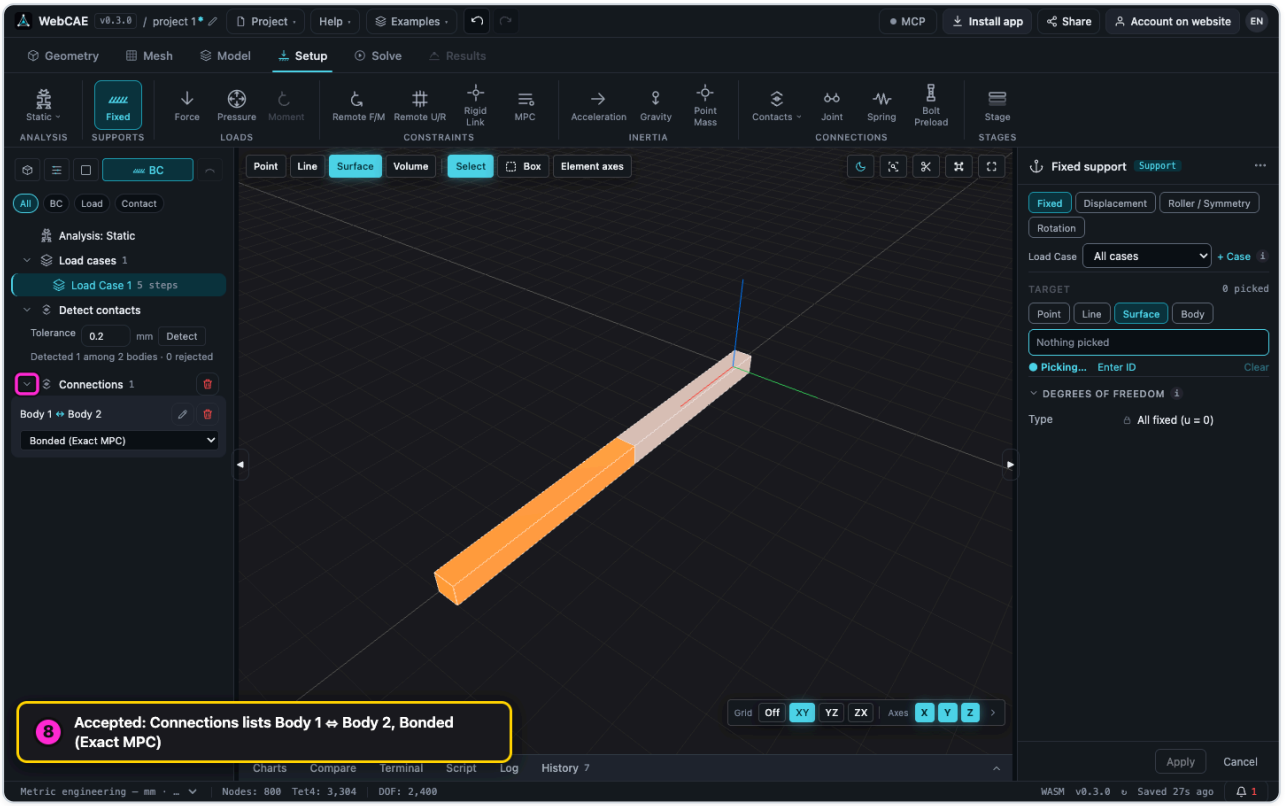
WHY Bonded (Exact MPC) ties all three displacement components across the interface by exact constraints, with no spring and no slip.



STEP 8 Check the accepted pair

Under **Connections** in the Setup tree the pair **Body 1 ⇔ Body 2** with **Bonded (Exact MPC)** is listed (framed).

WHY Without it the two bars would be separate and the solver would report a rigid-body motion.

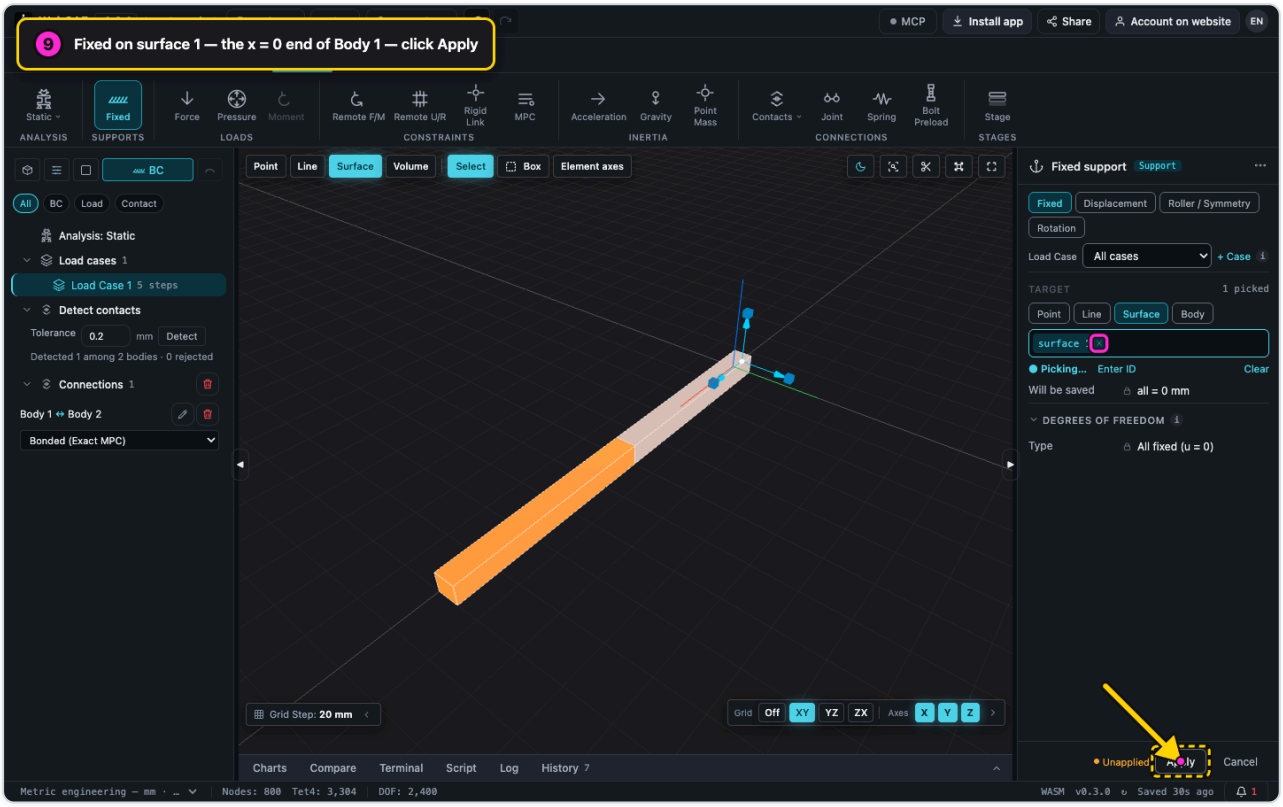


STEP 9

Clamp one end

Click **Fixed**, **Enter ID**, type `1` and press **Enter**: the end face of Body 1 at $x = 0$ is picked. It is the far end of the bar, turned away from this view, so the face itself is hidden: the blue glyphs at that end mark it. Click **Apply** (the arrow on the frame). (The number comes from the tree: **1 Surface** under Body 1.)

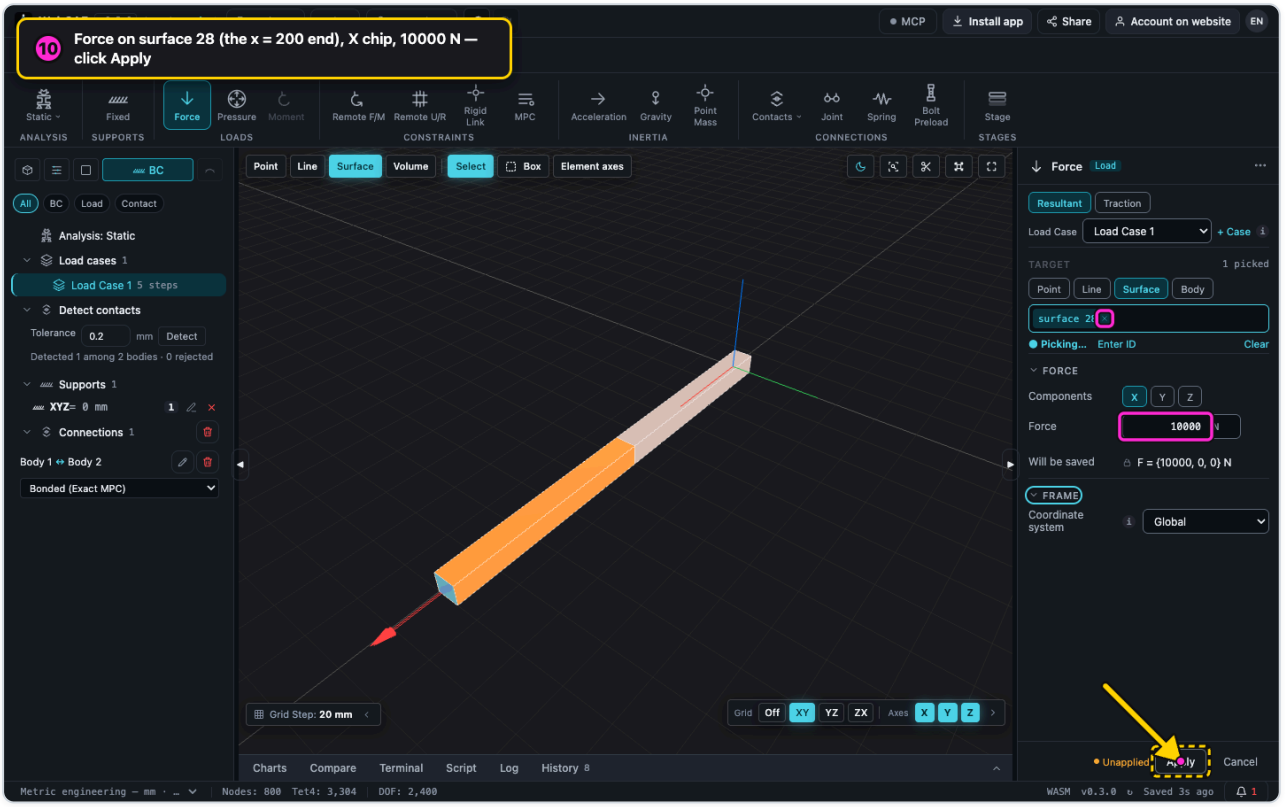
> **WHY** Removes all rigid-body motions.



STEP 10 Pull the other end

Click **Force**, **Enter ID**, type `28` and press **Enter**: the end face of Body 2 at $x = 200$ mm (**28 Surface** under Body 2) is picked. Press the **X** chip, type `10000` into **Force** (the framed field; the panel confirms *Will be saved: F = (10000, 0, 0) N*) and click **Apply** (the arrow on the frame).

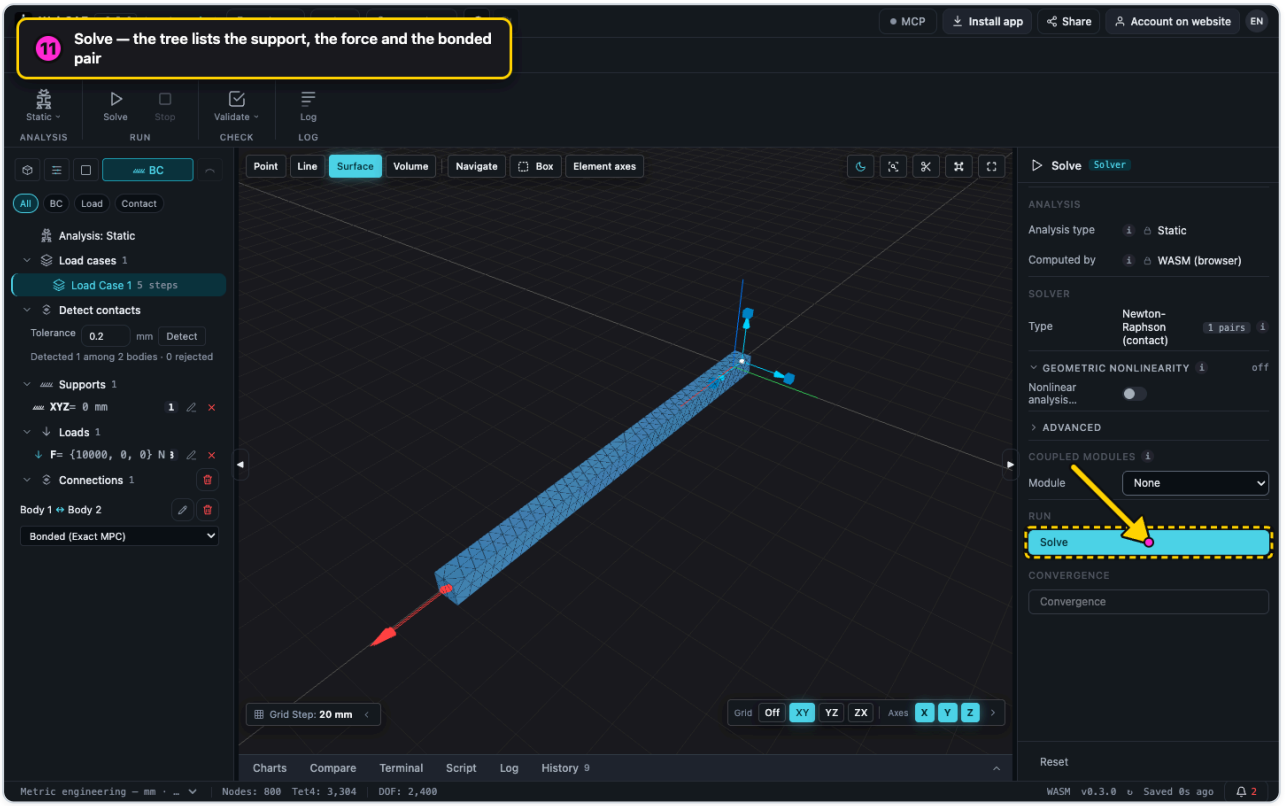
> **WHY** Pure axial tension through the bonded interface; not on the interface face at $x = 100$.



STEP 11 Solve

Open the **Solve** tab and click **Solve** (the arrow on the frame). The Setup tree on the left already lists the support, the force and the bonded pair.

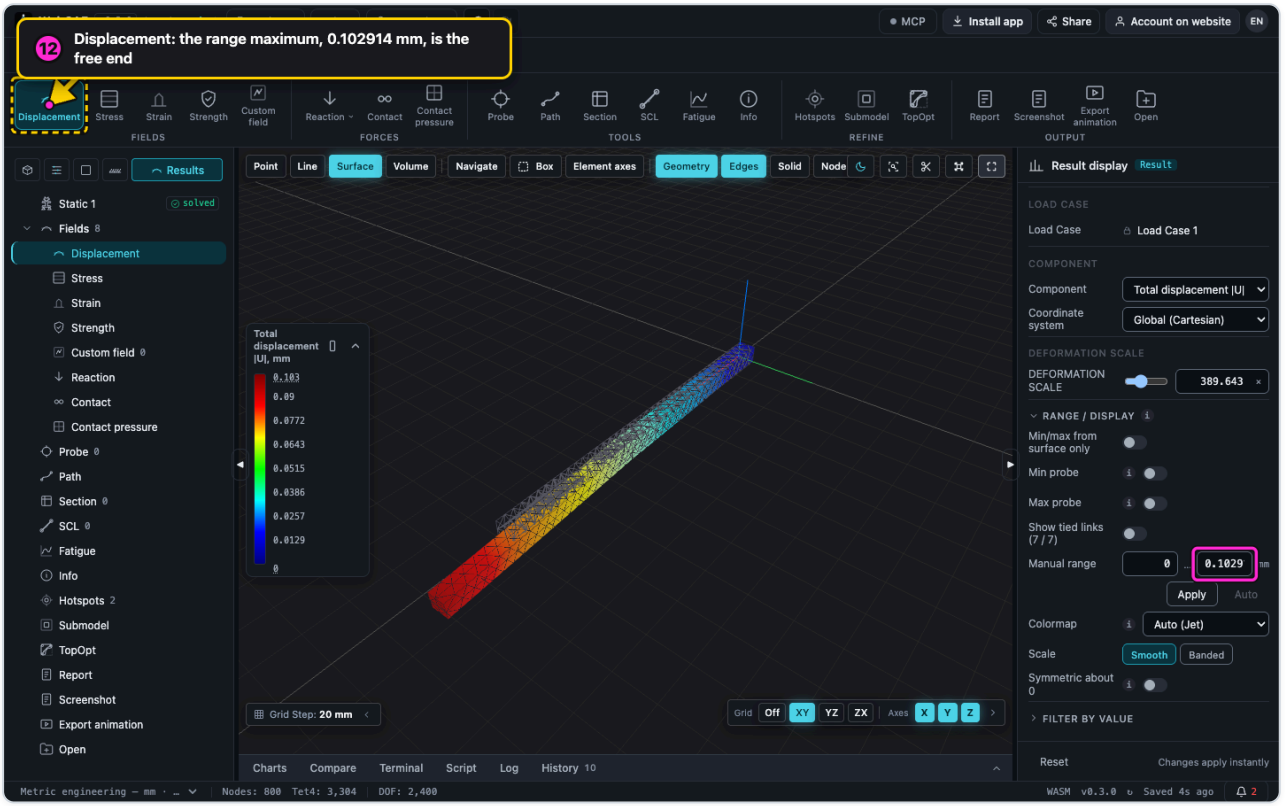
WHY One linear static solve of the 2-body assembly.



STEP 12 Read the displacement

Open **Results**, click **Displacement**. The **Manual range** maximum (the framed field) is the largest displacement, at the free end: **0.102914 mm** (web; the field is narrow and shows the first digits). The bar formula gives 0.100 mm.

> **WHY** A bonded interface makes the pair deform like one bar; the small excess is discretization.

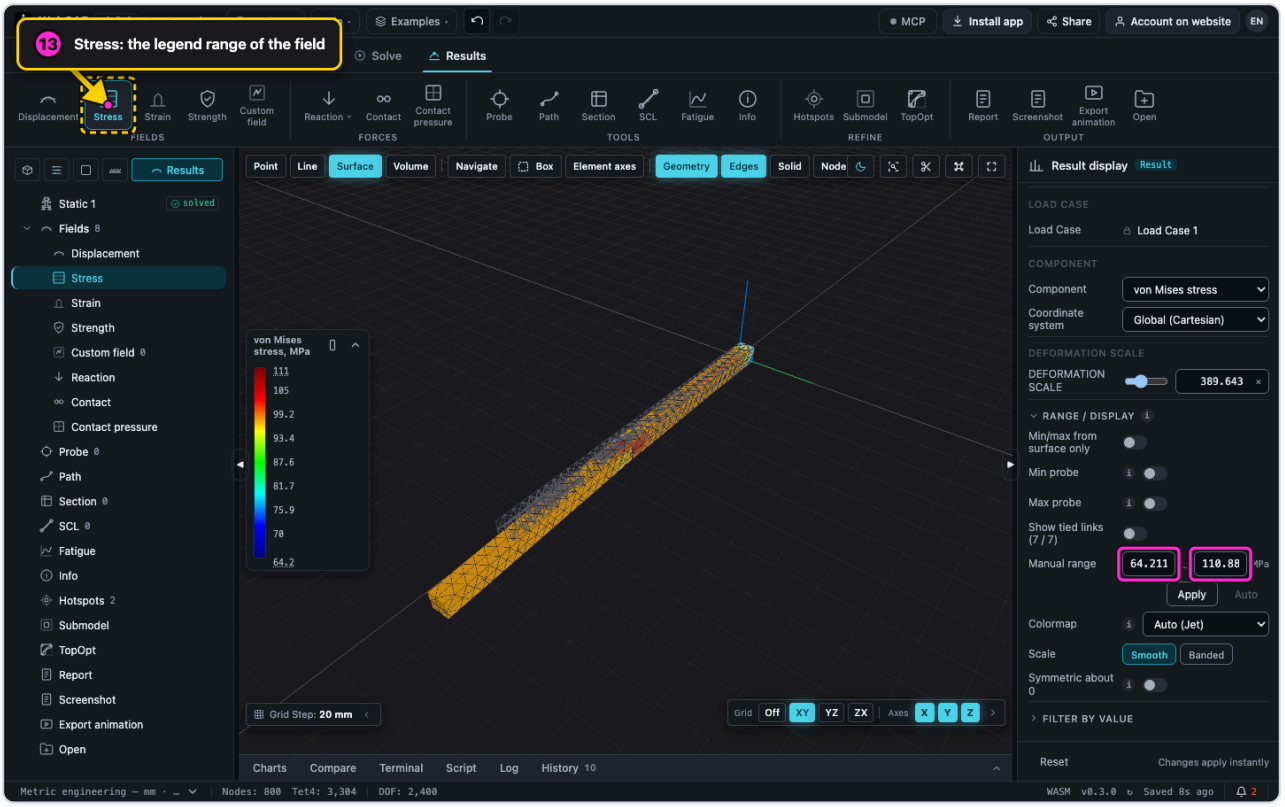


STEP 13

The stress field

Click **Stress**. The legend range of the field is framed on the frame (64.2117 … 110.888): the stress is nearly uniform along the bar, the highest values are the end effects at the clamp and at the loaded face. The average von Mises stress below (99.73 MPa; for pure tension it equals σ_{xx}) is computed by the script as the mean of the nodal von Mises values — the app shows no such average, so this number is not on a frame; the range ends 64.21 and 110.89 MPa are the same numbers as the framed legend.

WHY For pure tension $\sigma = P/A = 100\ \mathrm{MPa}$ along the whole bar.



Verification against analytical formulas

Quantity	Formula	WebCAE web (browser)	WebCAE desktop (native)	Error web / native	Where you see it
Axial displacement δ	$\frac{P(L_1+L_2)}{AE} = \frac{10\,000\times200}{100\times200\,000} = 0.100\ \mathrm{mm}$	0.102914 mm	0.09985 mm	+2.91 % / -0.15 %	Step 12 (Manual range)
Average axial stress σ_{xx} (von Mises, equal for pure tension)	$\frac{P}{A} = \frac{10\,000}{100} = 100\ \mathrm{MPa}$	99.73 MPa (mean of nodal von Mises, script)	100.00 MPa (probe at the middle of Body 1, a different reduction)	-0.27 % / 0.00 %	computed by the script (Step 13 shows the field legend only)
Tetrahedra (not a check)	—	3304	2748	—	Step 6 (status bar)

Analytical derivation.
For two prismatic bars of area $A = 100\ \mathrm{mm}^2$ and lengths $L_1 = L_2 = 100\ \mathrm{mm}$ in series under $P = 10\,000\ \mathrm{N}$:

$$\delta = \frac{PL_1}{AE} + \frac{PL_2}{AE} = \frac{10\,000 \times 200}{100 \times 200\,000} = 0.1000\ \mathrm{mm}$$

The web result is 0.102914 mm (+2.9 %). The model is not exactly the 1D bar: it uses linear P1 elements on a 5 mm mesh and the two bodies are meshed separately, so the tied interface is only approximately conforming; the stress field is not perfectly uniform (legend 64.2 … 110.9 MPa). The desktop result (0.09985 mm, -0.15 %) comes from a different mesh of the same model, so the two platforms differ by about 3 percentage points, mesh against mesh; the share of the interface constraint was not separated. Both are inside the 5 % tolerance, neither is zero.

TROUBLESHOOTING & NOTES

Common mistakes

- **Overlapping or disjoint bodies:** the translation must be exactly 100 mm along X so that the faces meet; otherwise no pair is detected.
- **Forgetting Accept all 1:** detected proposals do nothing until they are accepted; the Connections entry of Step 8 is the proof.
- **Loading the interface face instead of the end face:** the Fixed support goes on surface 1 ($x = 0$) and the Force on surface 28 ($x = 200$), not on the faces at $x = 100$.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#9 — Load cases: two loadings on one model (F09)**, or to lesson **#10 — From clicks to code: the DSL (F10)** to rebuild such models from a script.

WebCAE 101

LESSON 9 · F09

Load cases: two loadings on one model

Goal Build a cantilever bar, set up two independent load cases with a shared clamp and verify the result of each case.

Outcome Maximum deflection $\delta_1 = 1.595\ \text{mm}$ for Case 1 (100 N down), $\delta_2 = 0.797\ \text{mm}$ for Case 2 (50 N sideways) and the ratio $\delta_1/\delta_2 = 2.001$ (theory 2.0) — WebCAE web (browser); the desktop app gives 1.596 mm, 0.798 mm and 2.000.

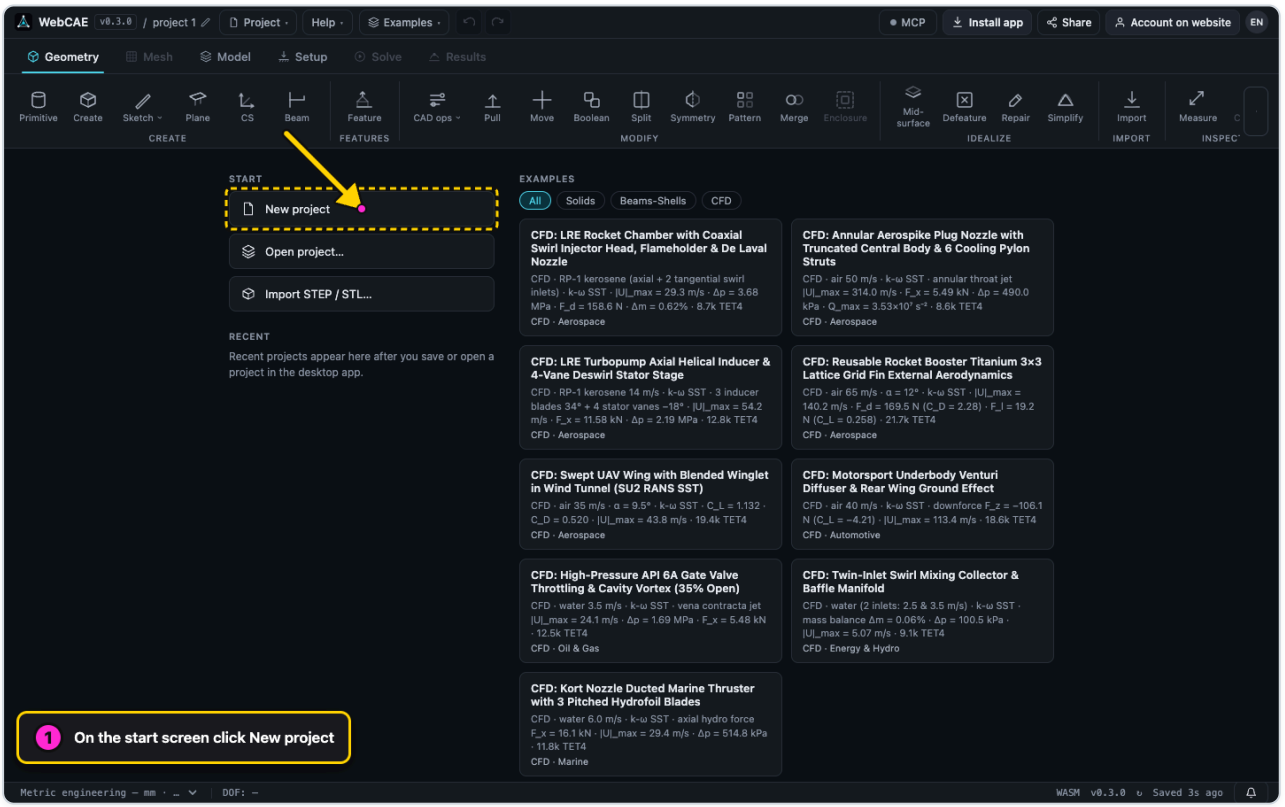
6–8 min | web (WASM) and desktop (native), shown in separate columns

Run: 2026-11-04-32ffaed2 / F09

STEP 1 Start a new project

Open WebCAE. The centre of the window shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and the gallery of **EXAMPLES**). Click **New project** (the arrow on the frame): the screen closes and the empty model opens. If you already see the empty model and no Start screen, skip this step.

WHY The Start screen covers the viewport and the tree until you choose what to open; this lesson builds its bar from scratch.

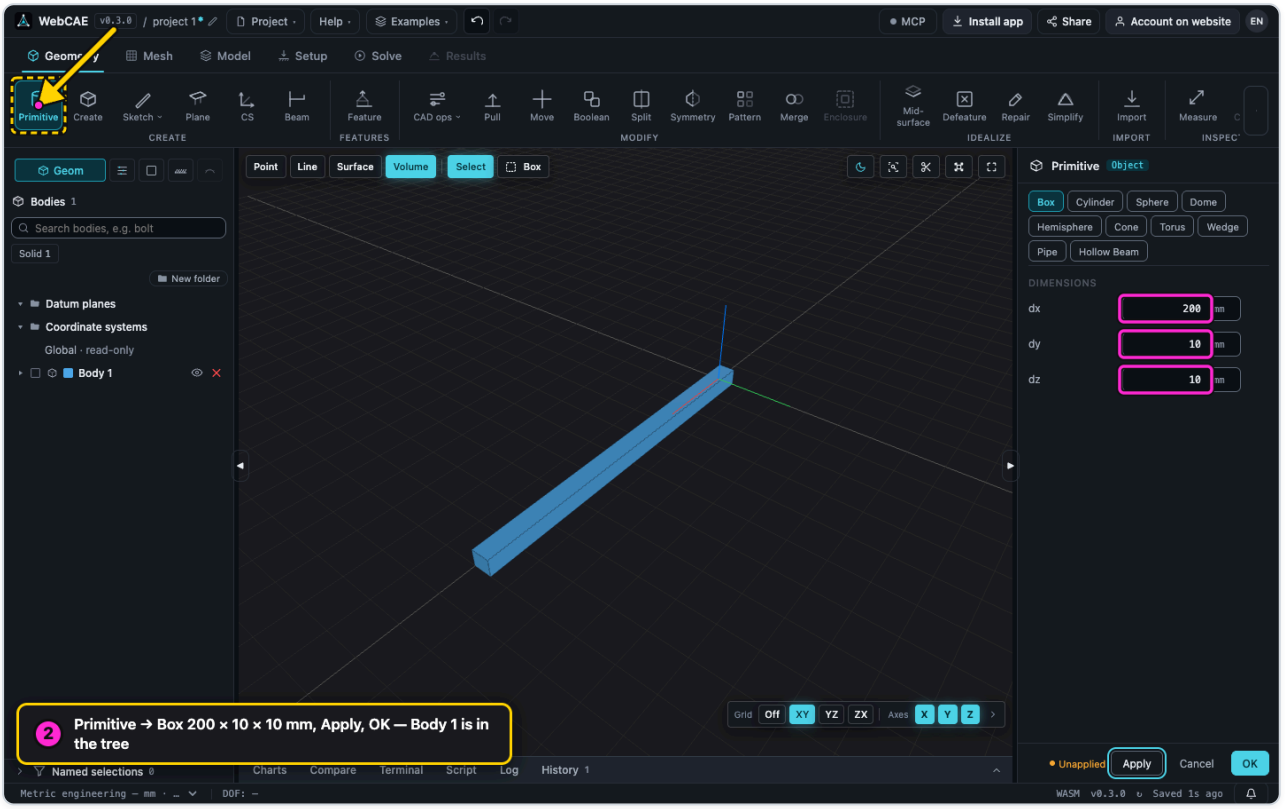


STEP 2

The bar

In the **Geometry** tab open **Primitive**, choose **Box** and set **dx** = `200`, **dy** = `10`, **dz** = `10` mm; click **Apply** and then **OK**. The frame is taken after Apply: **Body 1** is in the tree.

WHY A $200 \times 10 \times 10$ mm cantilever; face **1** is the end at $x = 0$, face **2** the end at $x = 200$ mm.

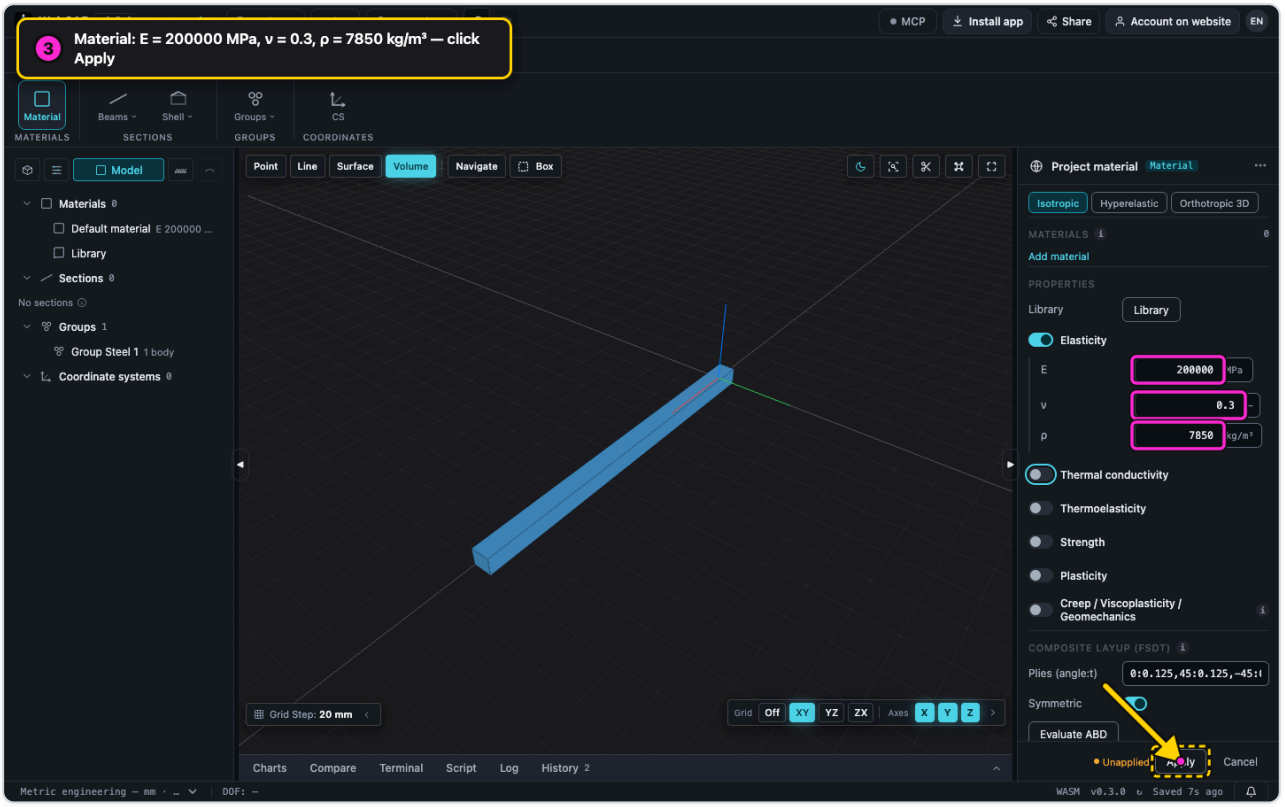


STEP 3

Material

Open the **Model** tab, **Material**: $E = 200\,000$ MPa, $\nu = 0.3$, $\rho = 7\,850$ kg/m^3; click **Apply** (the arrow on the frame).

WHY Steel; the deflection formula needs E.

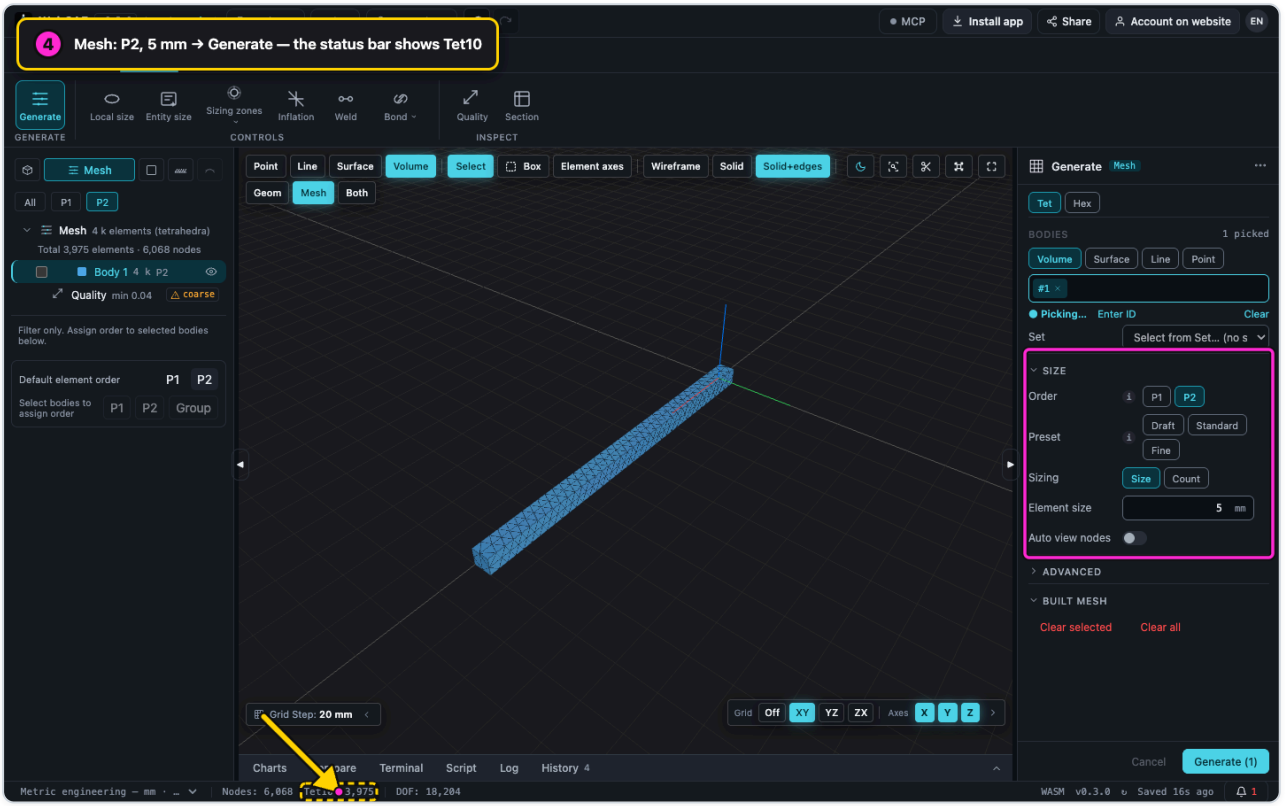


STEP 4

Mesh

Open the **Mesh** tab, click **Generate**, choose **P2** (quadratic) and set the element size to `5` mm; click **Generate**. The frame is taken after the mesh is built: the status bar shows **Tet10: 3975** in the browser (the desktop app builds 3090 elements).

> **WHY** Quadratic tetrahedra give bending deflection without shear locking. One mesh serves both load cases.

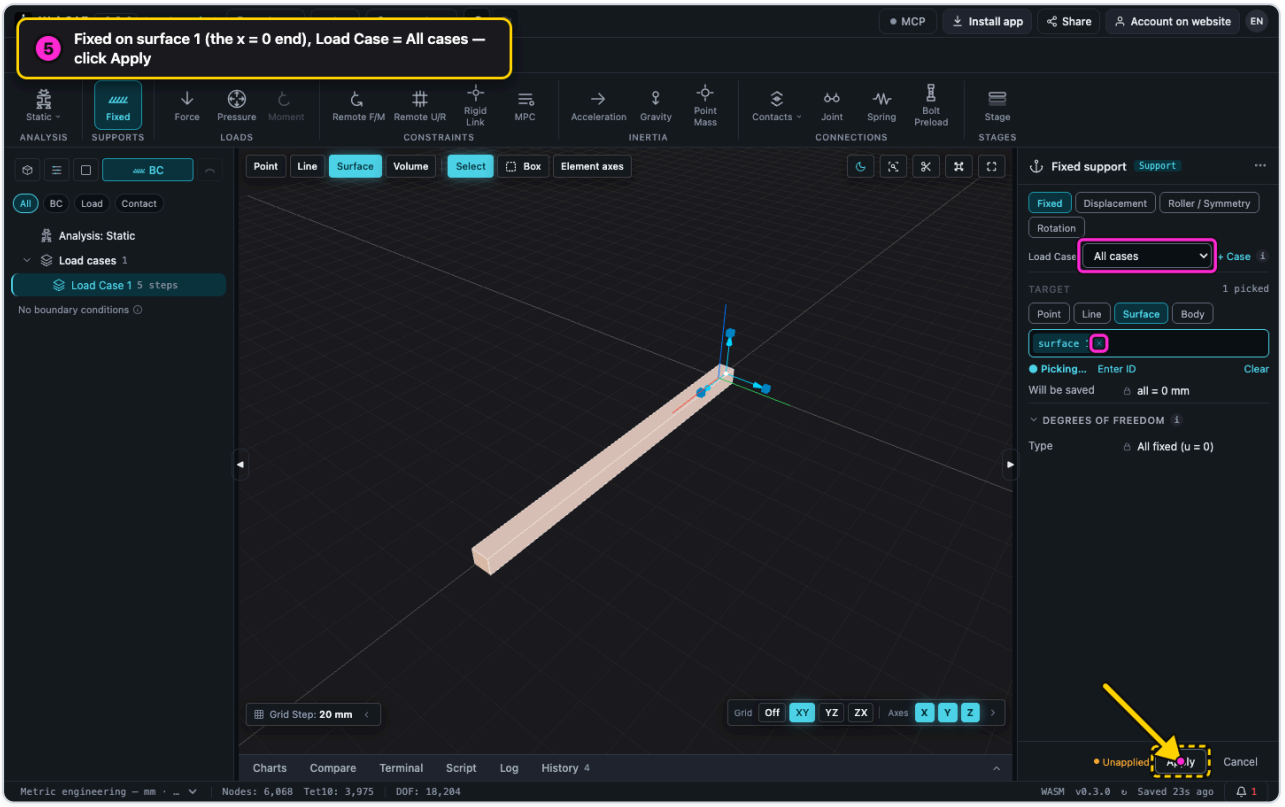


STEP 5

A support that belongs to every case

Open the **Setup** tab, click **Fixed**, check that **Load Case** says **All cases** (the framed list; it is preset, so just leave it), click **Enter ID**, type `1` and press **Enter** (surface 1 is picked: it is the far end of the bar, turned away from this view, so the face itself is hidden and the blue glyphs at that end mark it). Click **Apply** (the arrow on the frame).

> **WHY** A condition without a case tag applies to all cases, so the clamp is not copied.

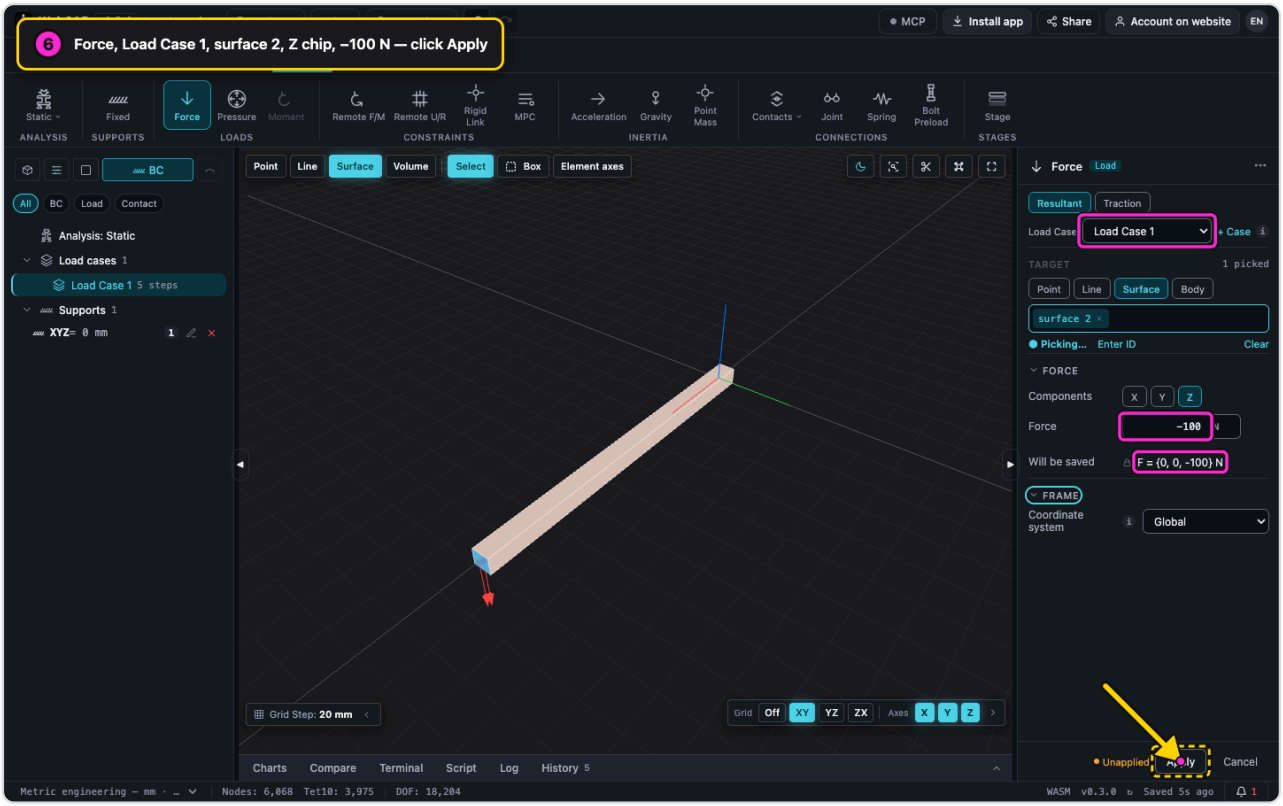


STEP 6

Load Case 1: 100 N down

Click **Force**, check that **Load Case** says **Load Case 1** (preset for the first force), click **Enter ID**, type `2` and press **Enter** (the near end of the bar is highlighted), press the **Z** chip and type `−100` into **Force**. The panel confirms **F = {0, 0, -100} N**. Click **Apply** (the arrow on the frame).

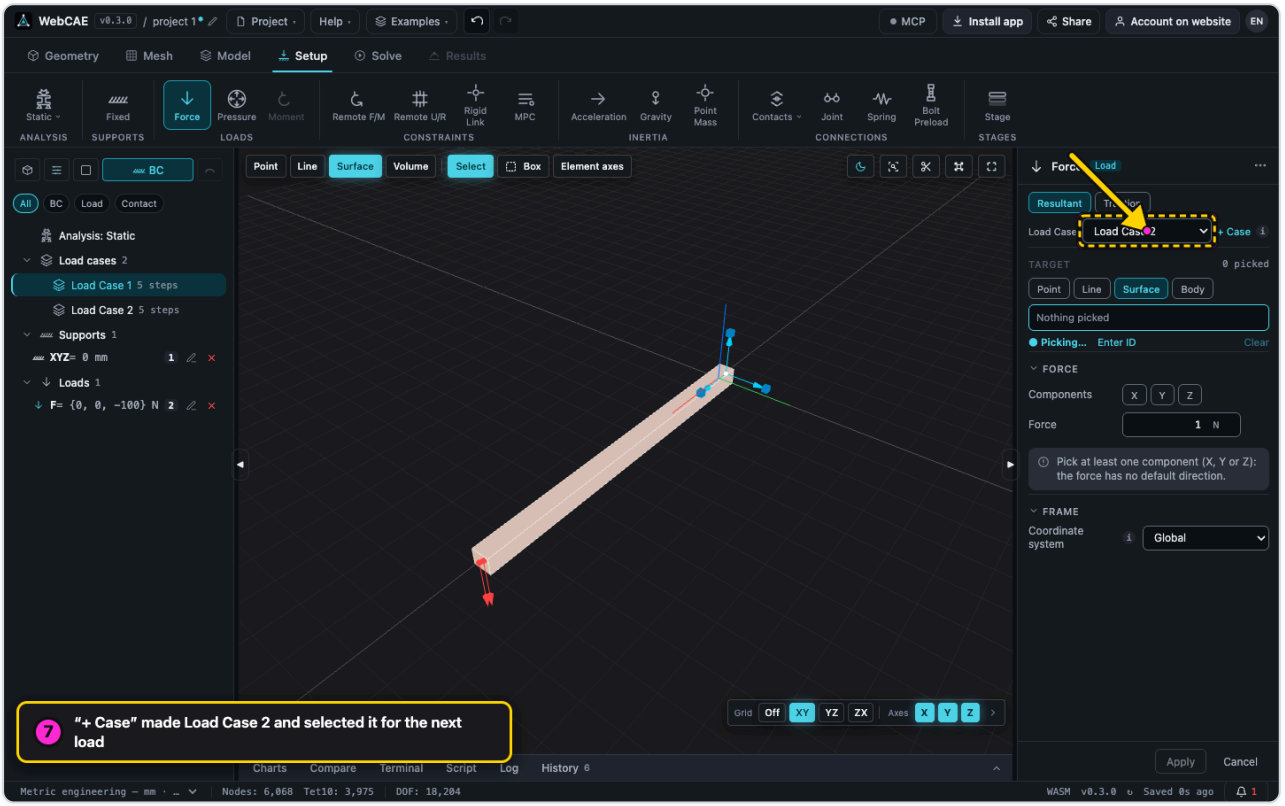
> **WHY** The confirmation line shows the vector that will be stored, so a wrong axis is seen before Apply.



STEP 7 Add a second case

Click **Force** again and press **+ Case** next to the **Load Case** list. **Load Case 2** appears in the tree and is selected in the list (the framed field) for the next load.

WHY Several loadings can be solved on the same mesh without re-meshing.

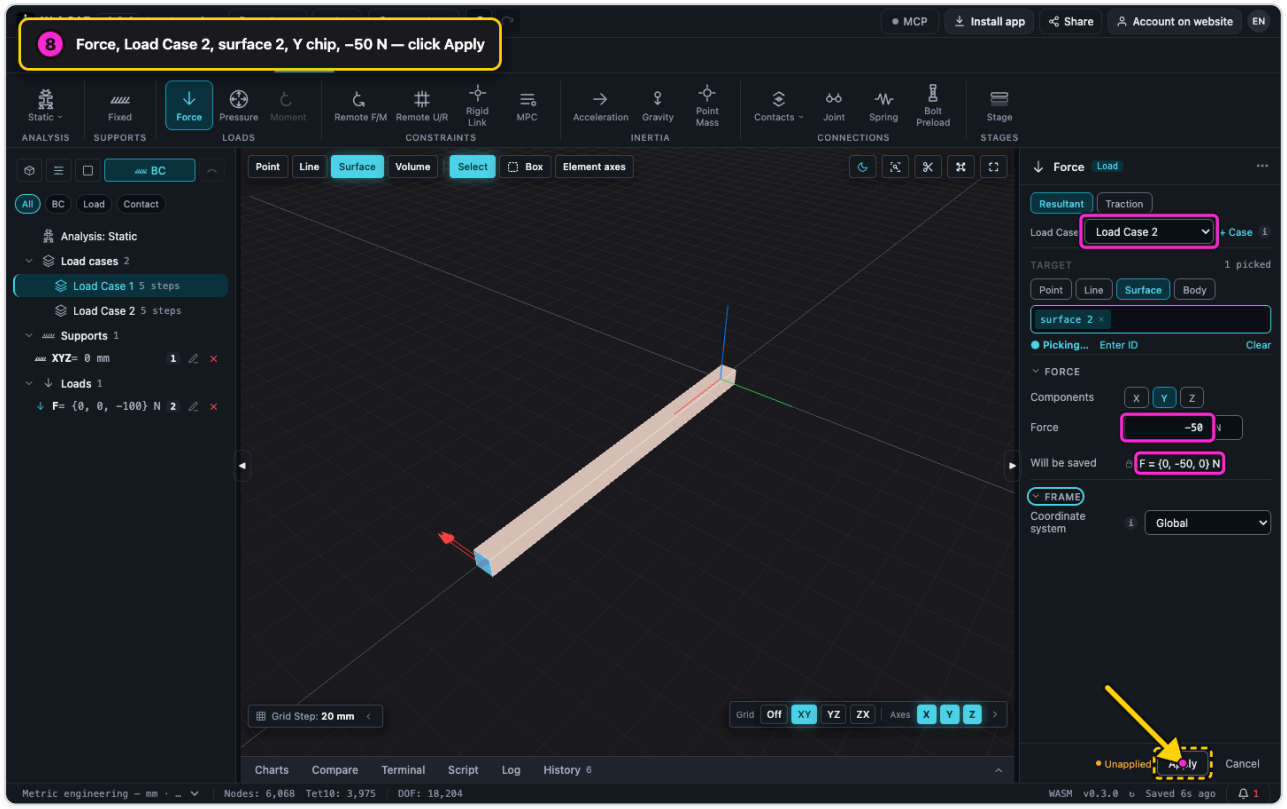


STEP 8

Load Case 2: 50 N sideways

With **Load Case 2** selected: **Enter ID**, type `2`, press **Enter**, press the **Y** chip and type `−50` into **Force**. The panel confirms **F = {0, -50, 0} N**. Click **Apply** (the arrow on the frame).

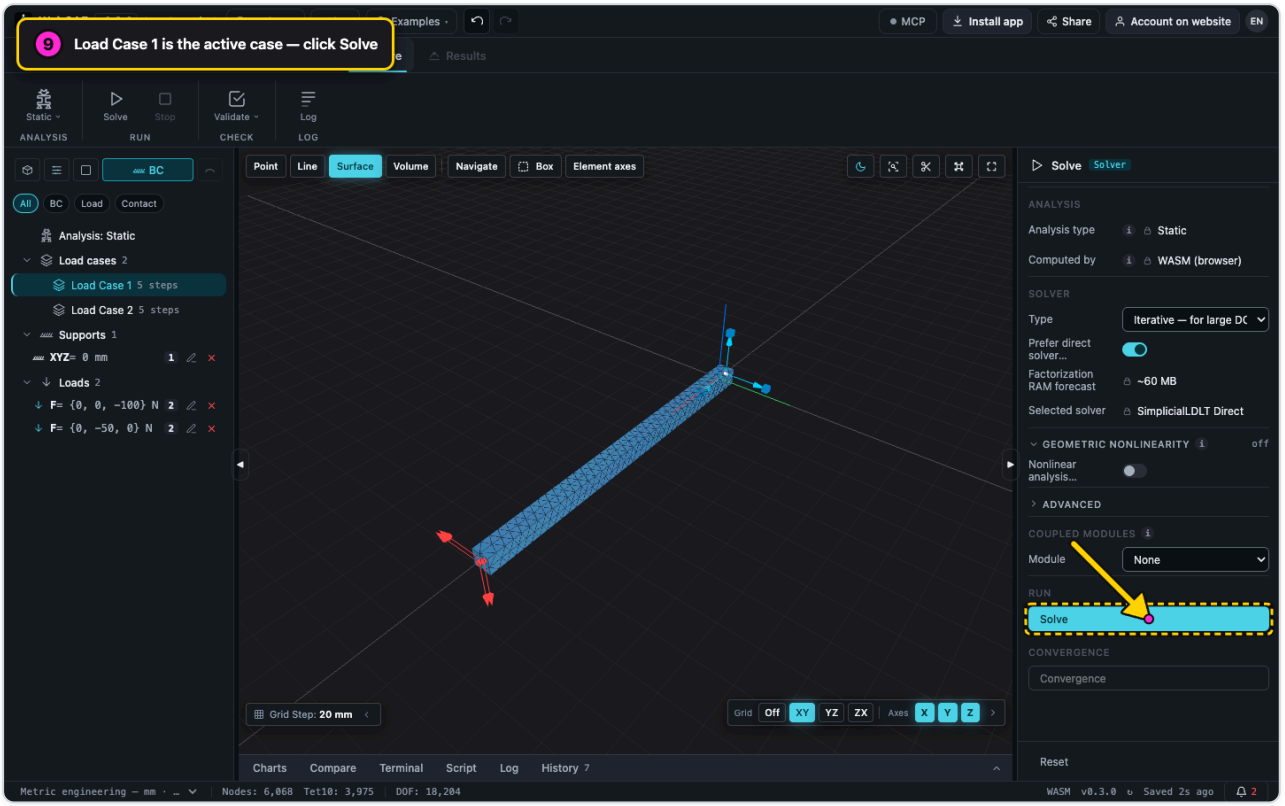
WHY The cross-section is square ($I_y = I_z = 833.33\ \text{mm}^4$), so half the force gives half the deflection.



STEP 9 Solve Load Case 1

In the Setup tree click **Load Case 1** (it becomes the active case), open the **Solve** tab and click **Solve** (the arrow on the frame).

WHY Only the active case is solved.

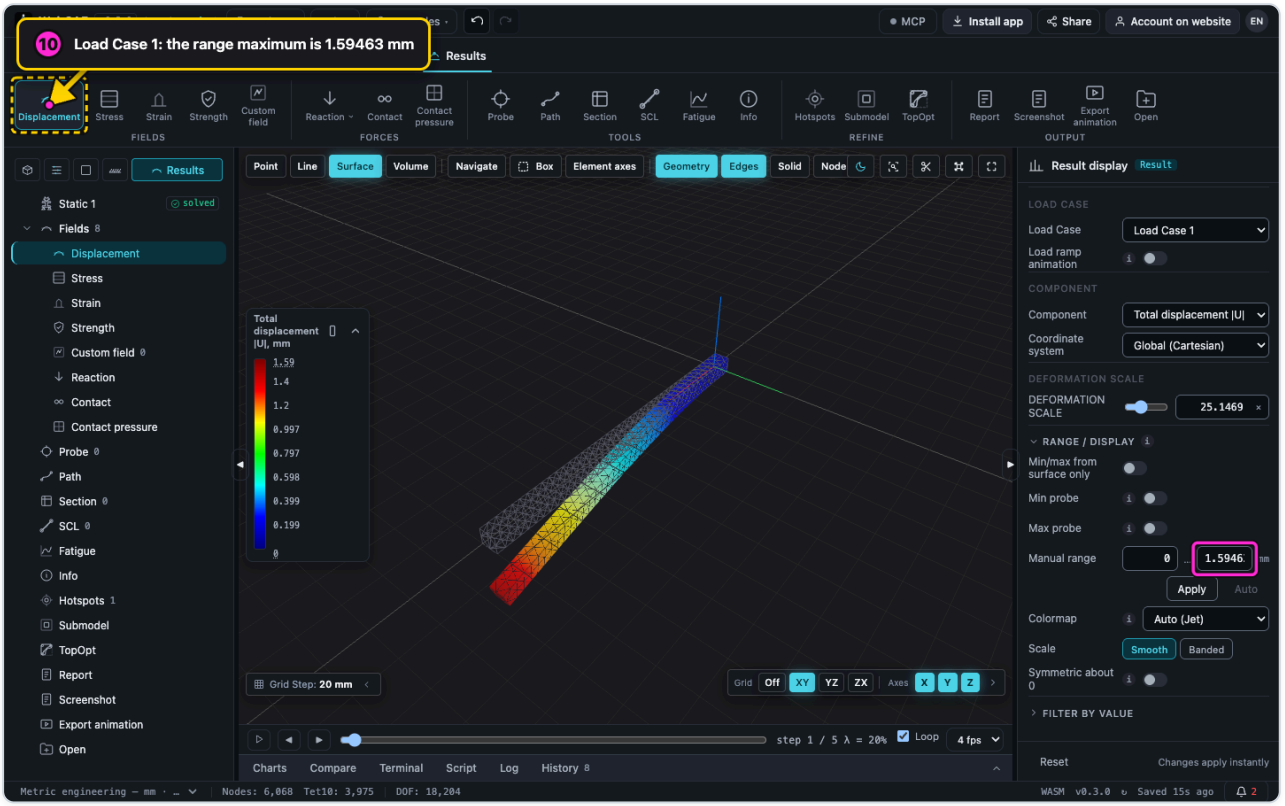


STEP 10

Results of Load Case 1

Open **Results** and click **Displacement**. The **Manual range** maximum (framed) is the largest displacement: **1.59463 mm** (web).

WHY Results belong to the active case.

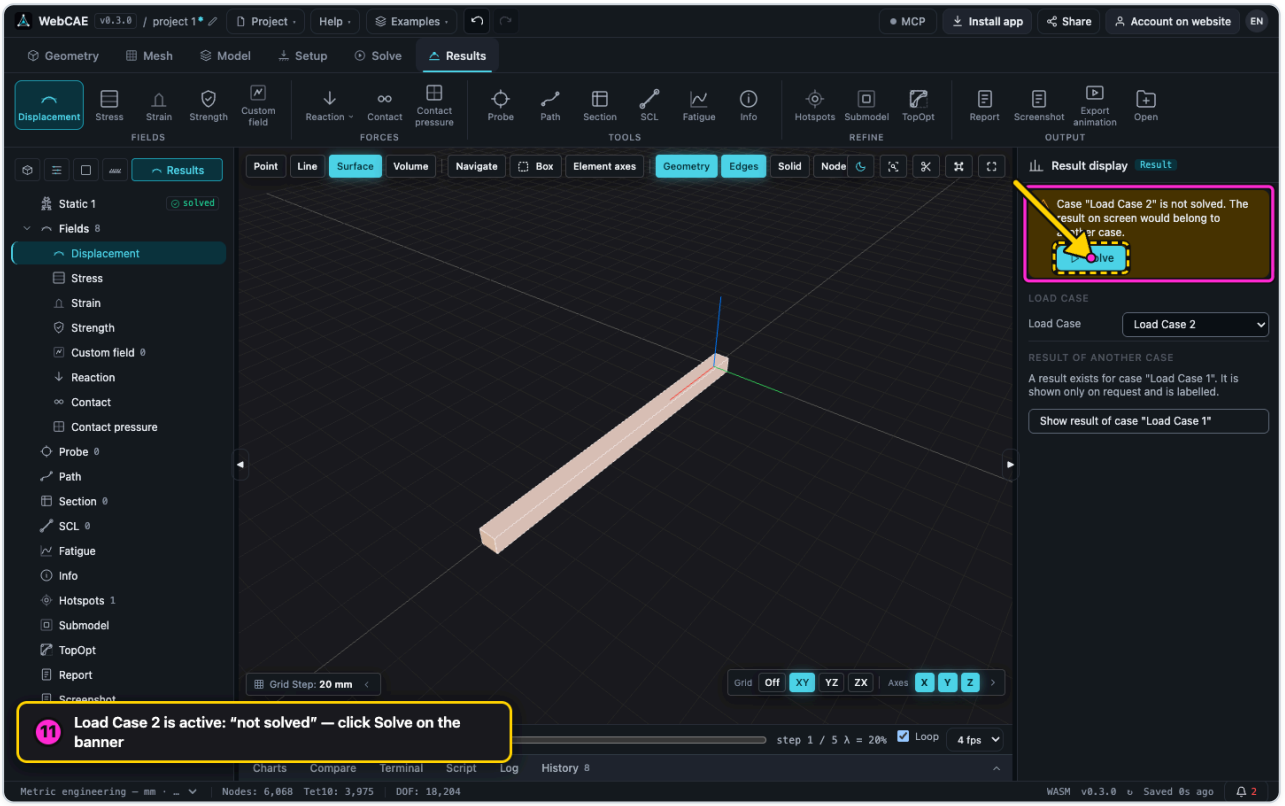


STEP 11

The other case is honestly “not solved”

Go back to **Setup**, click **Load Case 2** in the tree, and open **Results** again. WebCAE does not show Case 1 results under the name of Case 2: the banner says **Case "Load Case 2" is not solved. The result on screen would belong to another case.**. Click **Solve** on the banner (the arrow on the frame).

WHY An unsolved case is never masked by the field of another case.

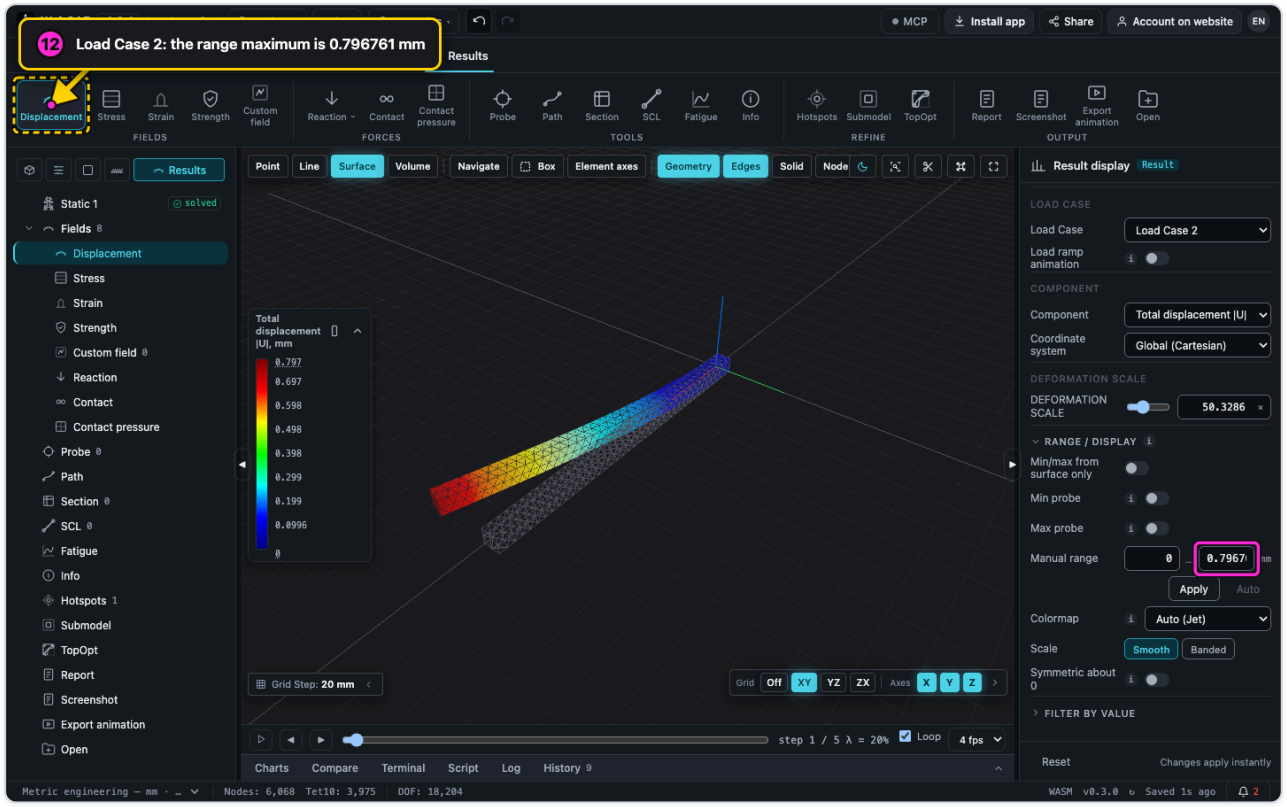


STEP 12

Results of Load Case 2

After the solve click **Displacement**: the **Manual range** maximum (framed) is **0.796761 mm** (web). The ratio of the two maxima is $1.59463/0.796761 = 2.0014$.

WHY Linear problem: half the force, half the deflection.



Verification against analytical formulas

Quantity	Formula	WebCAE web (browser)	WebCAE desktop (native)	Error web / native	Where you see it
Case 1 δ_1 ($F_z = -100$ N)	$\frac{FL^3}{3EI} = \frac{100\times200^3}{3\times200\,000\times833.33} = 1.600$ mm	1.59463 mm	1.5958 mm	-0.34 % / -0.26 %	Step 10 (Manual range)
Case 2 δ_2 ($F_y = -50$ N)	$\frac{50\times200^3}{3\times200\,000\times833.33} = 0.800$ mm	0.796761 mm	0.7977 mm	-0.40 % / -0.28 %	Step 12 (Manual range)
Ratio δ_1/δ_2	$\frac{100}{50} = 2.000$	2.0014	2.0004	+0.070 % / 0.019 %	computed from Steps 10 and 11
Tet10 elements (not a check)	—	3975	3090	—	Step 4 (status bar)

TROUBLESHOOTING & NOTES

Common mistakes

- **Leaving the support in a single case:** if **Fixed** is assigned to Load Case 1 instead of All cases, Load Case 2 has no support and its solve fails with a singular matrix.
- **Wrong axis:** the Z chip for Case 1 and the Y chip for Case 2; read the **Will be saved** line before Apply.
- **Reading a result under the wrong case:** the active case is the one selected in the Setup tree; an unsolved case shows the banner of Step 11, not a field.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#10 — From clicks to code: the DSL (F10)** to generate the same models, supports and probes from a script.

WebCAE 101

LESSON 10 · F10

From clicks to code: the DSL

Goal See that every condition you set with the mouse is a line of the script language (DSL), and rebuild, solve and check three models — static, modal and buckling — from text.

Outcome Tip deflection 1.5957 mm vs 1.600 mm, first natural frequency 204.14 Hz vs 203.85 Hz, buckling factor 4.5759 vs 4.569 — WebCAE web (browser); desktop (native) values in the table below.

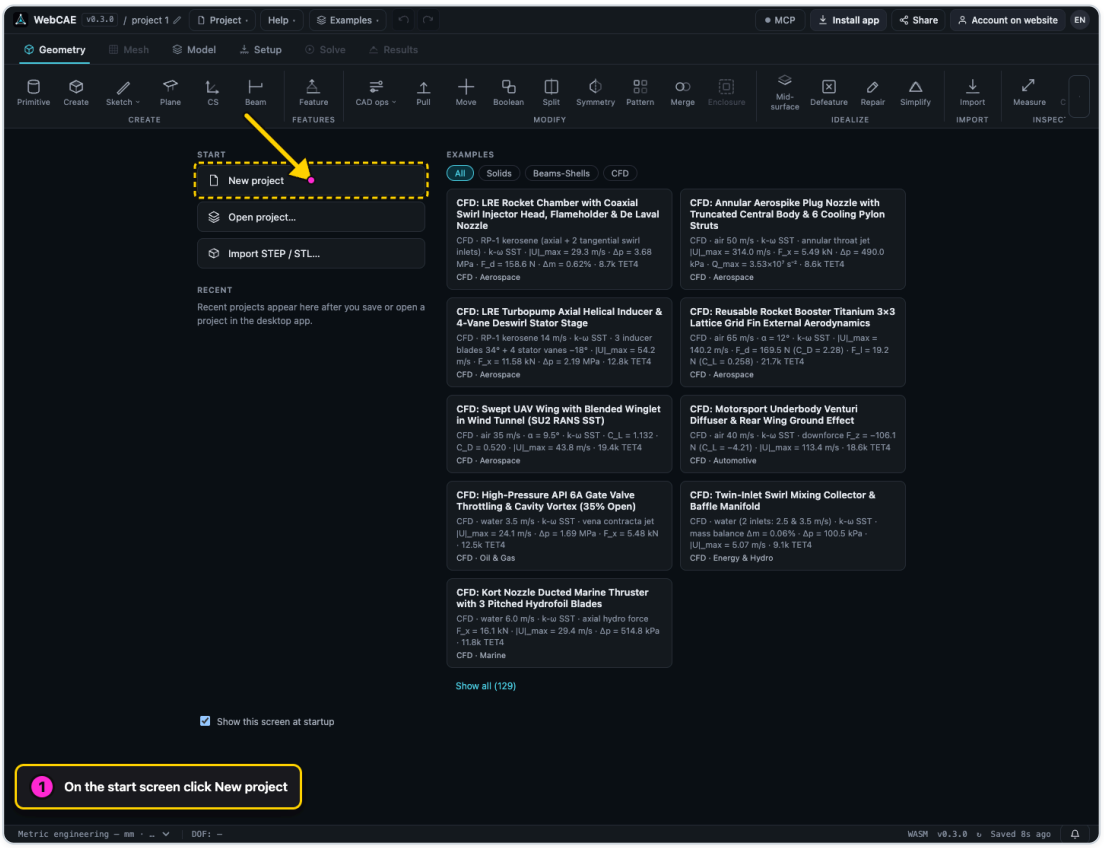
8–10 min | web (WASM) and desktop (native), shown in separate columns

Run: 2026-11-04-32ffaed2 / F10

STEP 1 Start a new project

Open WebCAE. The centre of the window shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and the gallery of **EXAMPLES**). Click **New project** (the arrow on the frame): the screen closes and the empty model opens. If you already see the empty model and no Start screen, skip this step.

WHY The Start screen covers the viewport and the tree until you choose what to open.



STEP 2 Build the cantilever with the mouse (lessons F01 and F09)

Do four things; the frame shows the result in the Setup tree.

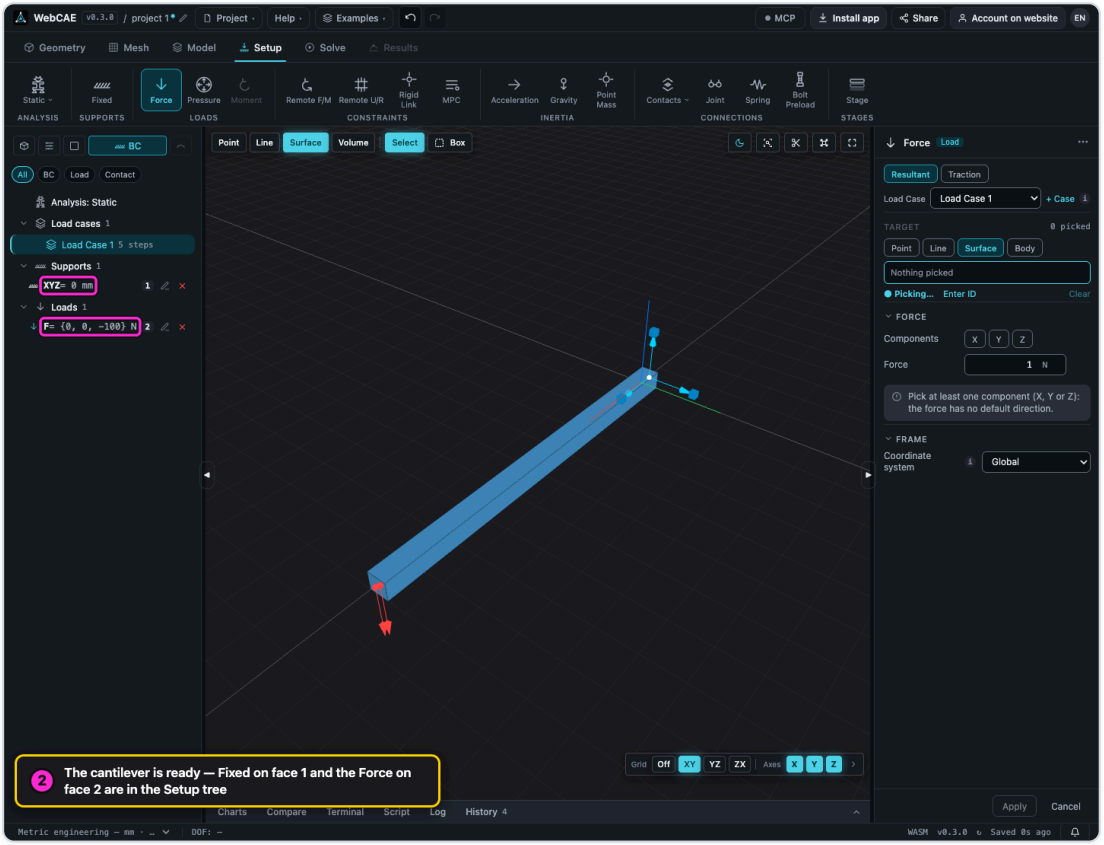
1. **Geometry → Primitive**, **Box**: **dx** = `200`, **dy** = `10`, **dz** = `10` mm, **Apply**, **OK**.

2. **Model → Material**: **E** = `200000` MPa, **ν** = `0.3`, **ρ** = `7850` kg/m³, **Apply**.

3. **Setup → Fixed**, **Enter ID**, type `1`, **Enter**, **Apply** (the end at x = 0).

4. **Setup → Force**, **Enter ID**, type `2`, **Enter**, press the **Z** chip, type `−100` into **Force**, **Apply** (the end at x = 200 mm).

WHY The next steps turn exactly this model into text and back; the Setup tree must list a support (**XYZ= 0 mm**) and a force (**F = …**).



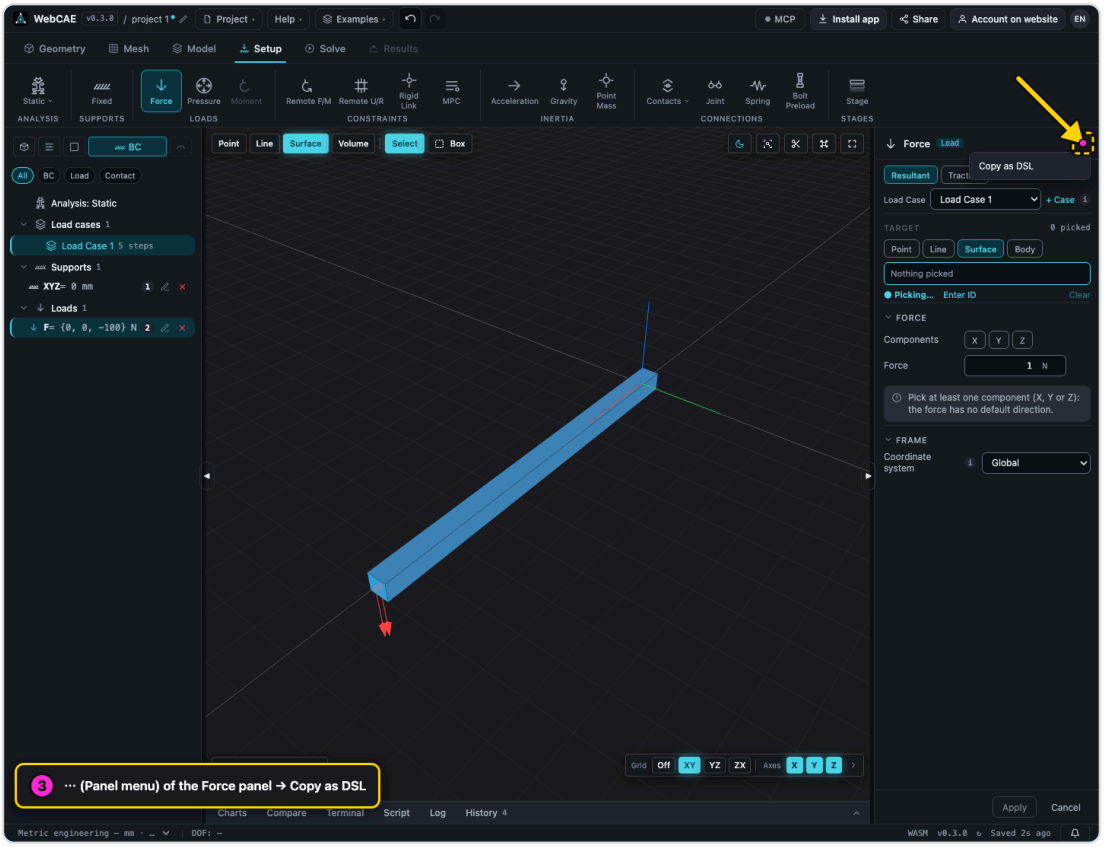
STEP 3

Copy a panel as DSL

In the Setup tree click the **Force** entry (**F = ...**), then in its panel click the ··· (**Panel menu**) in the header (framed) and choose **Copy as DSL** (the arrow). The panel itself shows no confirmation; the message **DSL copied to the clipboard (N line(s))** goes to the notification bell at the bottom right. The clipboard now holds the lines of the panel:

```
fix surface number 1
load surface number 2 force=-100 component=z case="Load Case 1"
```

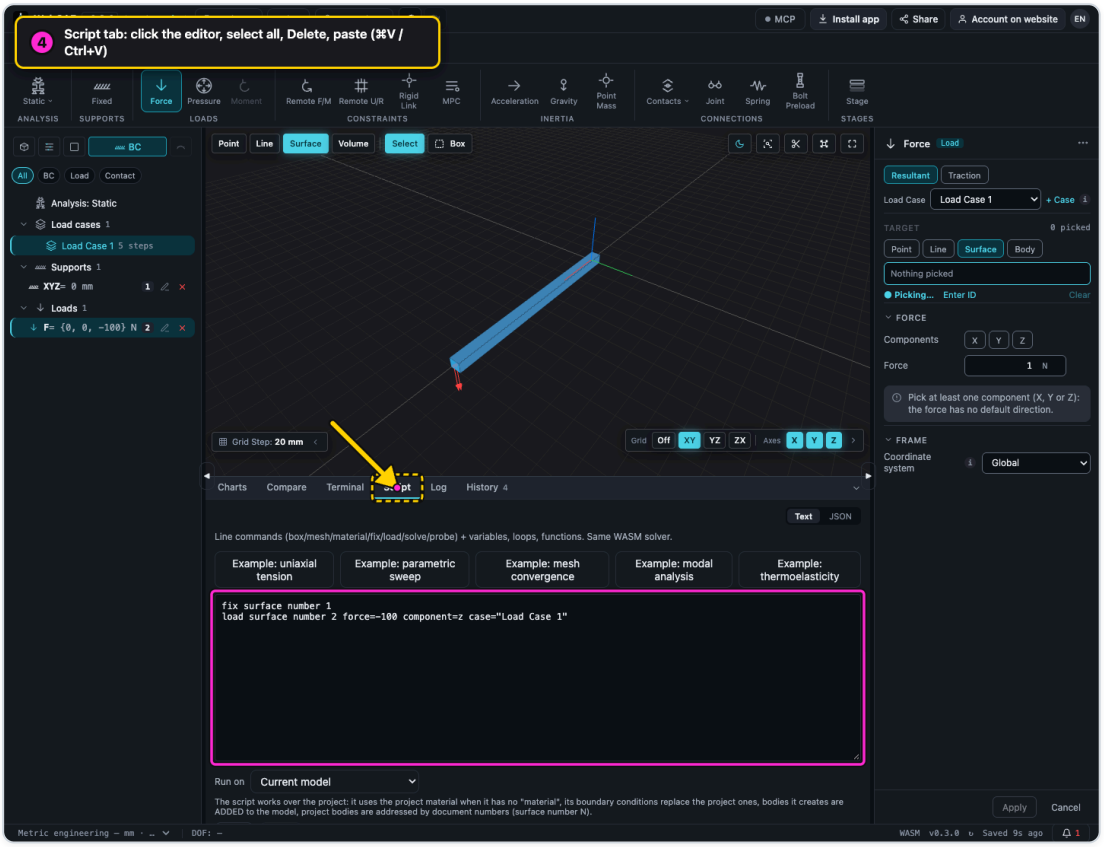
WHY Every panel can write itself as script lines, and the lines name the faces by the same numbers as the tree: `surface number 1`, `surface number 2`.



STEP 4 Paste into the Script tab

Open **Script** in the bottom dock (click the tab once — a second click on the active tab folds the dock again; drag the top edge of the dock up to make it taller). **The editor already contains a sample script — remove it first:** click into the editor, select everything (⌘A on a Mac, Ctrl+A elsewhere), press **Delete**, then paste (⌘V / Ctrl+V). The two copied lines are in the editor (framed).

WHY What you set with the mouse and what the script says are the same model.



STEP 5

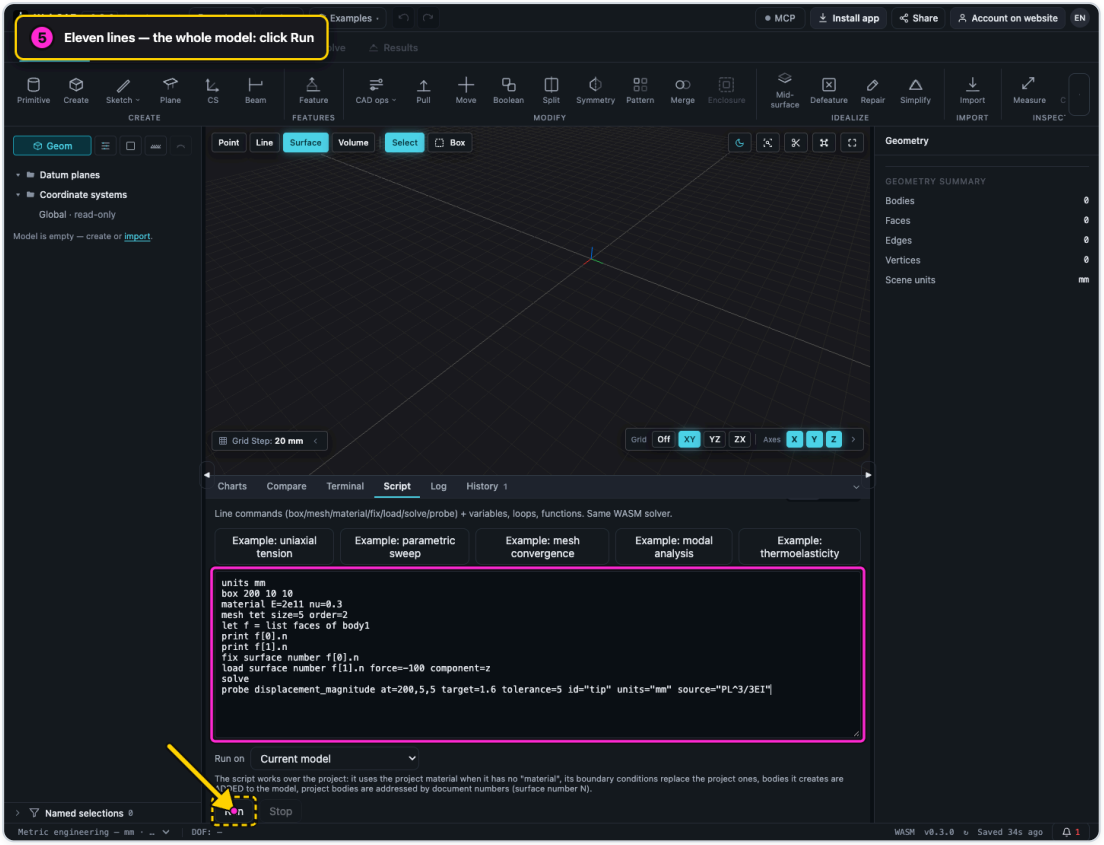
A whole model in eleven lines

Open the **Project** menu at the top, choose **New project**, and in the question *Save changes to "project 1"?* press **No** (the scripts below build their own geometry; a project with other bodies would confuse `body1`). In the **Script** tab click into the editor, select all, **Delete**, and paste the script (copy the block below; typing it by hand also works) — eleven lines:

```
units mm
box 200 10 10
material E=2e11 nu=0.3
mesh tet size=5 order=2
let f = list faces of body1
print f[0].n
print f[1].n
fix surface number f[0].n
load surface number f[1].n force=-100 component=z
solve
probe displacement_magnitude at=200,5,5 target=1.6 tolerance=5 id="tip" units="mm" source="
```

The frame shows the editor with this text (framed); click **Run** (the arrow, at the bottom edge of the dock; on a 900 px high window the lower part of the dock is below the visible area — scroll the dock with the mouse wheel or drag its top edge higher, until **Run** is visible).

WHY Geometry, material, mesh, support, load, solve and a check against the formula, in text. `print f[0].n` and `print f[1].n` write the numbers of the first two faces of the body (the clamp and the load face).

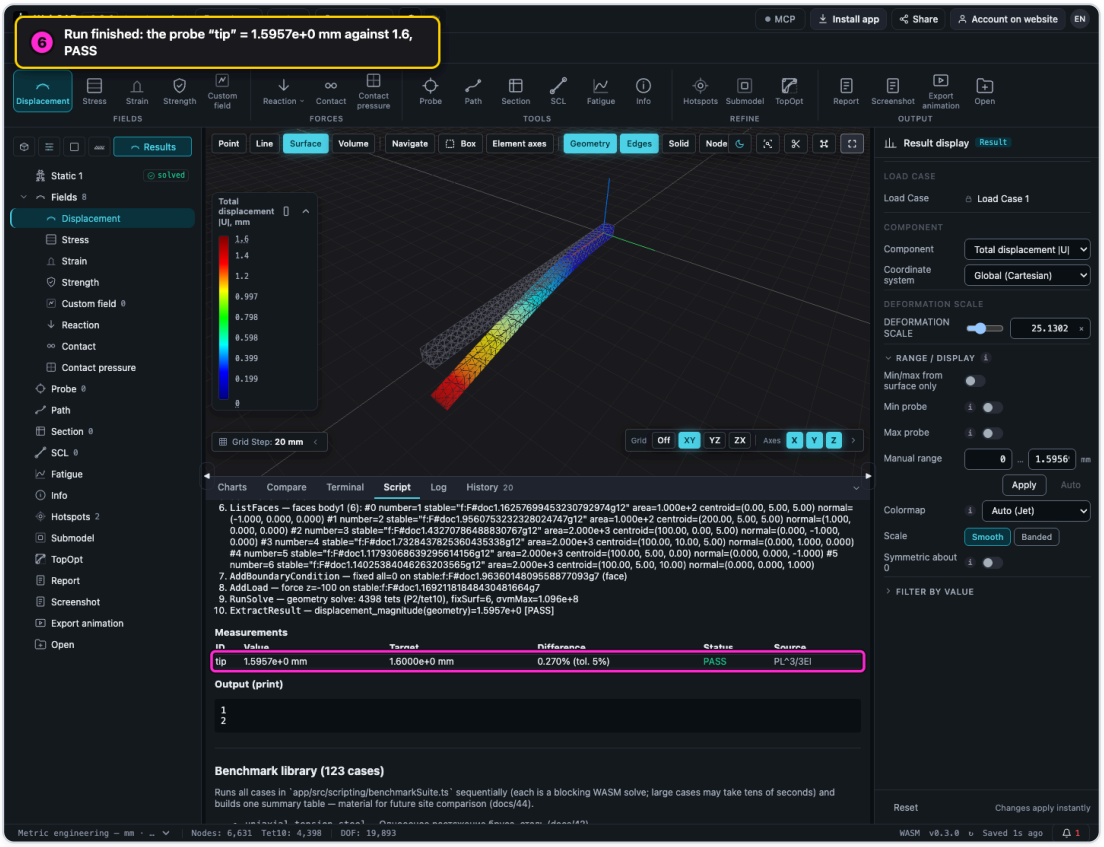

5 Eleven lines — the whole model: click Run
Examples
MCP
Install app
Share
Account on website
EN
Results
Primitive
Create
Sketch
Plane
CS
Beam
Feature
CAD ops
Pull
Move
Boolean
Split
Symmetry
Pattern
Merge
Enclosure
Mid-surface
Defeature
Repair
Simplify
Import
Measure
CREATE
FEATURES
MODIFY
IDEALIZE
IMPORT
INSPEC
Geom
Datum planes
Coordinate systems
Global · read-only
Model is empty — create or import.
Point
Line
Surface
Volume
Select
Box
Geometry
GEOMETRY SUMMARY
Bodies 0
Faces 0
Edges 0
Vertices 0
Scene units mm
Grid Step: 20 mm
Grid Off XY YZ ZX
Axes X Y Z
Charts
Compare
Terminal
Script
Log
History 1
Line commands (box/mesh/material/fix/load/solve/probe) + variables, loops, functions. Same WASM solver.
Example: uniaxial tension
Example: parametric sweep
Example: mesh convergence
Example: modal analysis
Example: thermoelasticity
units mm
box 200 10 10
material E=2e11 nu=0.3
mesh tet size=5 order=2
let f = list faces of body1
print f[0].n
print f[1].n
fix surface number f[0].n
load surface number f[1].n force=-100 component=z
solve
probe displacement_magnitude at=200,5,5 target=1.6 tolerance=5 id="tip" units="mm" source="PL^3/3EI"
Run on
Current model
The script works over the project: it uses the project material when it has no "material", its boundary conditions replace the project ones, bodies it creates are ADDED to the model, project bodies are addressed by document numbers (surface number N).
Run
Stop
Named selections 0
Metric engineering — mm · …
DOF: —
WASM v0.3.0
Saved 34s ago

STEP 6

Read the probe

When the run is over, press **Fit all content to screen** (top right of the viewport) to see the model. The **Measurements** table in the dock (framed) has one row: the probe `tip` — value **1.5957e+0 mm** (that is 1.5957 mm; the table prints numbers in this exponent form), target 1.6000e+0 mm, difference 0.270% (tol. 5%), status **PASS**. Below it the **Output (print)** block shows 1 and 2 — the two `print` lines.

WHY A probe is a regression test: the run reports PASS or FAIL against the formula.



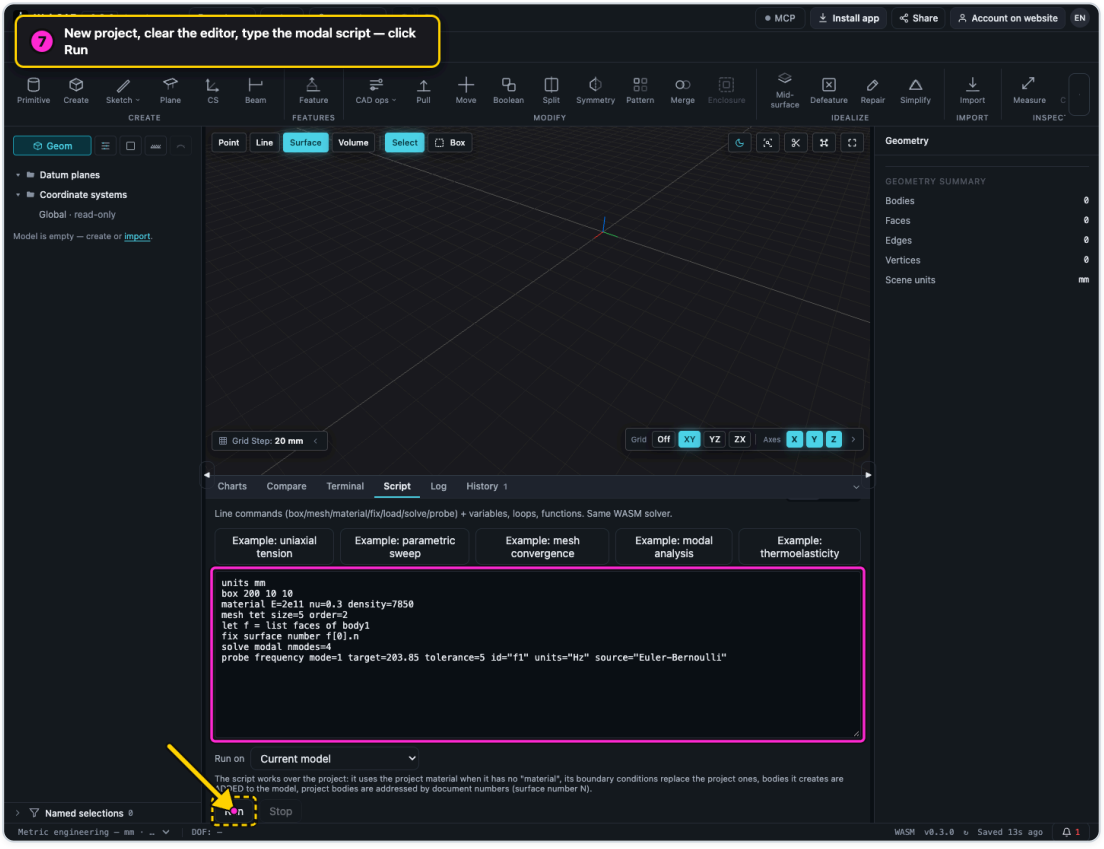
STEP 7 The same bar, modal analysis

Project → New project (answer **No** to the save question), click into the editor, select all, **Delete**, paste:

```
units mm
box 200 10 10
material E=2e11 nu=0.3 density=7850
mesh tet size=5 order=2
let f = list faces of body1
fix surface number f[0].n
solve modal nmodes=4
probe frequency mode=1 target=203.85 tolerance=5 id="f1" units="Hz" source="Euler-Bernoulli
```

and click **Run** (the arrow).

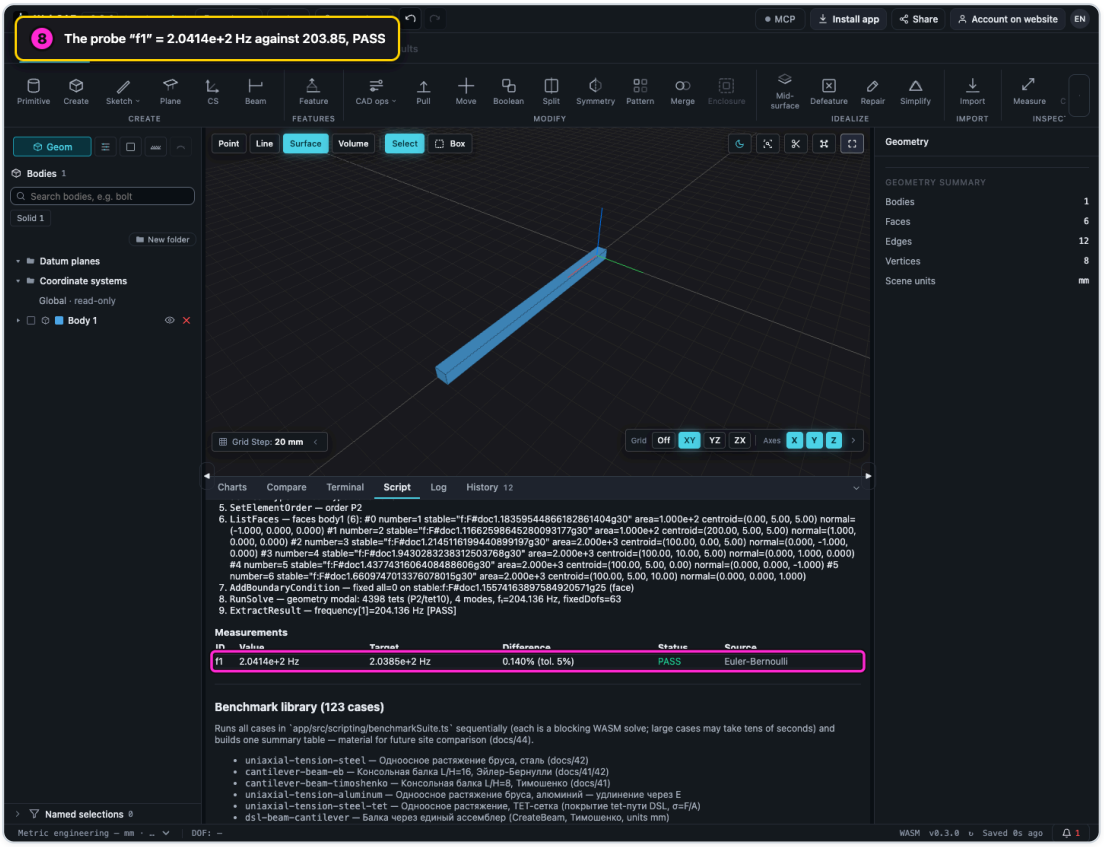
WHY Only the `solve` line and the probe changed; `density` was added to the material.



STEP 8 Read the first frequency

The **Measurements** table (framed): the probe `f1` — **2.0414e+2 Hz** (204.14 Hz) against the Euler–Bernoulli 2.0385e+2 Hz, difference 0.140% (tol. 5%), status **PASS**.

> **WHY** The same number as in the modal lesson, from eight lines.



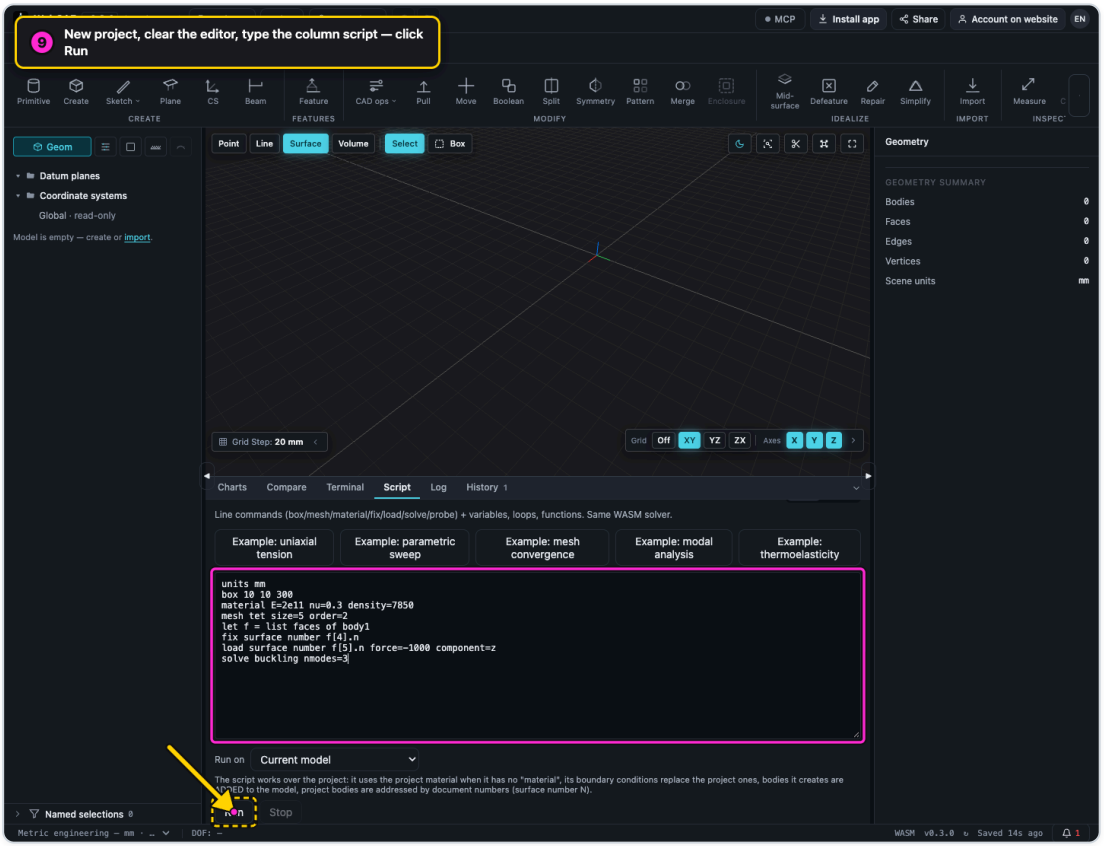
STEP 9 A slender column, buckling

Project → New project (answer **No** to the save question), click into the editor, select all, **Delete**, paste:

```
units mm
box 10 10 300
material E=2e11 nu=0.3 density=7850
mesh tet size=5 order=2
let f = list faces of body1
fix surface number f[4].n
load surface number f[5].n force=-1000 component=z
solve buckling nmodes=3
```

and click **Run** (the arrow).

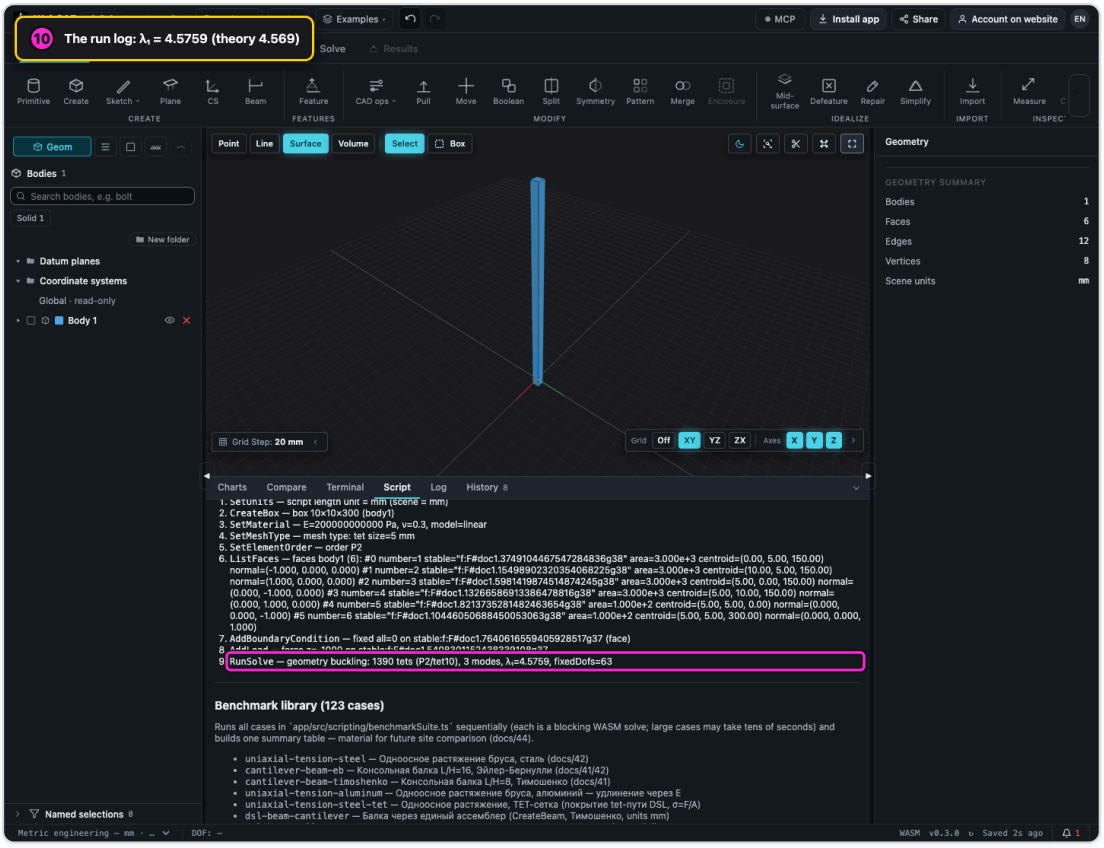
WHY A 10 × 10 × 300 mm column, clamped at one end and loaded with a reference force of 1000 N at the other.



STEP 10 Read the buckling factor

The run log (the framed line) says: **RunSolve — geometry buckling: 1390 tets (P2/tet10), 3 modes, λ_1=4.5759, fixedDofs=63**. The critical load is the factor times the reference force: $P_{cr} = \lambda_1 \times 1000\ \mathrm{N}$.

WHY Euler: $P_{cr} = \frac{\pi^2 EI}{4L^2} = 4569\ \mathrm{N}$, so $\lambda = 4.569$.



VERIFICATION BENCHMARK

Check against analytical formulas

Analysis	Case	Formula / source	Analytical	WebCAE web (browser)	WebCAE desktop (native)	Error web / native	Where you see it
Static deflection	200×10×10 mm cantilever, 100 N	$\delta = \frac{PL^3}{3EI}$	1.600 mm	1.5957 mm	1.5958 mm	-0.27 % / -0.26 %	Step 6 (Measurements)
Modal frequency	cantilever, first bending mode	$f_1 = \frac{1.875^2}{2\pi L^2}\sqrt{\frac{EI}{\rho A}}$	203.85 Hz	204.14 Hz	204.13 Hz	+0.14 % / 0.14 %	Step 8 (Measurements)
Buckling factor	10×10×300 mm column, $P_{ref} = 1000$ N	$\lambda = \frac{\pi^2 EI}{4L^2 P_{ref}}$	4.569	4.5759	4.5759	+0.15 % / 0.15 %	Step 10 (run log)

TROUBLESHOOTING & NOTES

Common mistakes

- **Leaving the sample script in the editor:** the editor starts with a sample (in Russian) — select all and Delete before pasting or typing, otherwise its lines run too.
- **Using MPa for Young's modulus in the DSL:** the material form takes MPa (`200000`), the DSL takes pascals (`E=2e11`). `E=200000` in a script means 200 kPa and gives deflections about 10^6 times too large.
- **Addressing faces by direction:** faces are referred to by number (`surface number f[0].n` or `surface number 1`), never by words such as "bottom".
- **Running a script in a project that already has bodies:** start with **Project → New** so that `body1` and the face numbers are what the script expects.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#11 — Sketch on a face and extrude (F11)** to draw a profile and build a part from it.

WebCAE 101

LESSON 11 · F11

Sketch on Face, Project 3D and Extrude

Goal Create a sketch directly on a face of a part, project the face boundary into the sketch, inset it with Offset, and extrude the inset loop as a boss joined to the plate.

Outcome One body: a $100 \times 100 \times 10$ mm plate with a centred $60 \times 60 \times 15$ mm boss, total volume $V = 154\,000\ \mathrm{mm}^3$. WebCAE Measure shows 154 000 mm^3.

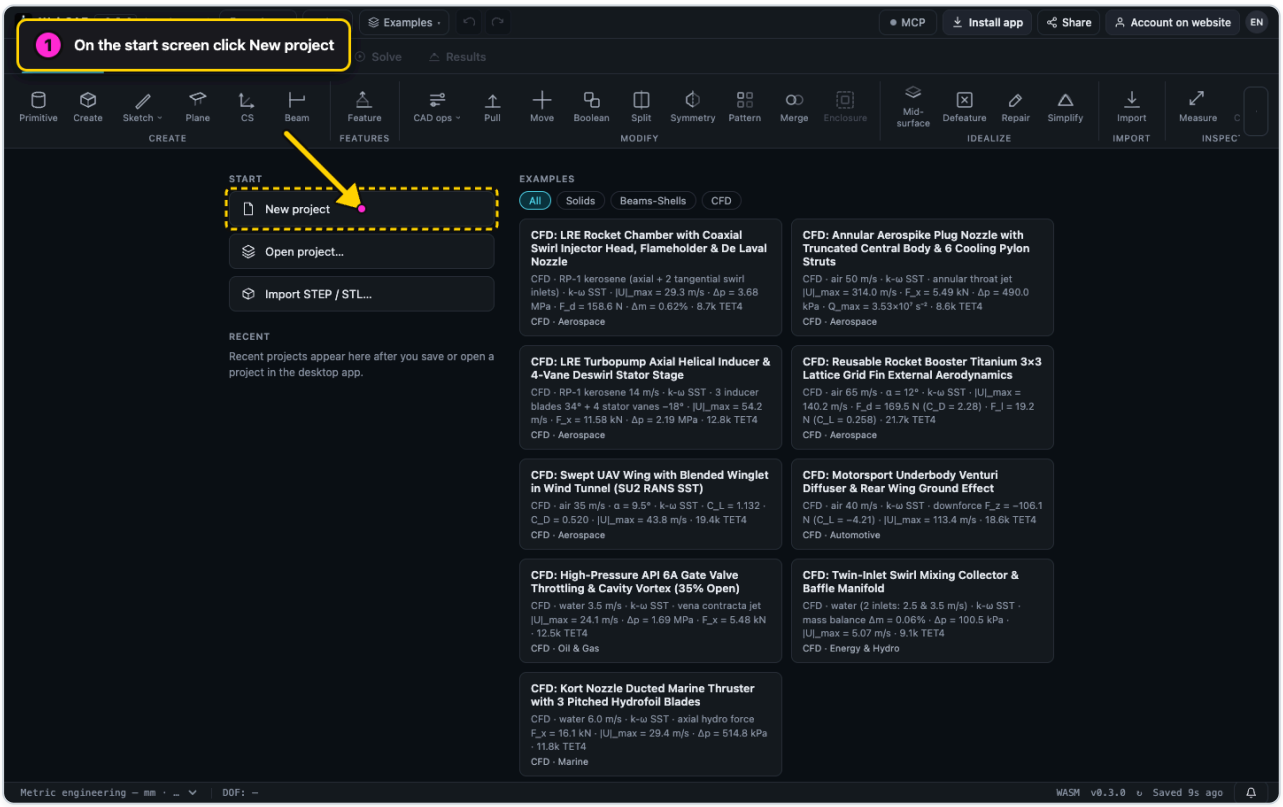
5–6 min | WebCAE web (browser), geometry engine OpenCASCADE in WASM

Run: 2026-11-04-32ffaed2 / F11

STEP 1 Start a new project

Open WebCAE. The centre of the window shows the **Start** screen (**New project**, **Open project...**, **Import STEP / STL...** and the gallery of **EXAMPLES**). Click **New project** (the arrow on the frame): the screen closes and the empty model opens. If you already see the empty model and no Start screen, skip this step.

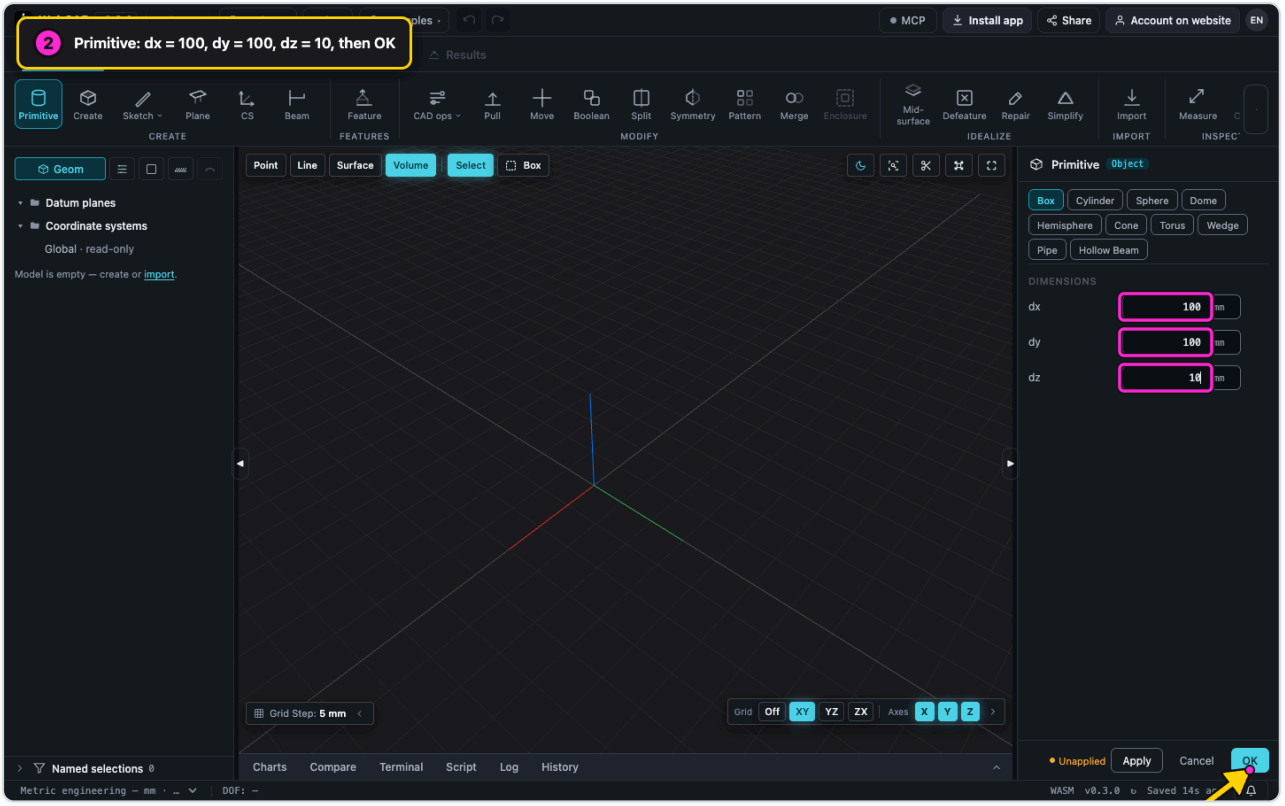
WHY The Start screen covers the viewport and the tree until you choose what to open; this lesson builds its plate from scratch.



STEP 2

Create the plate

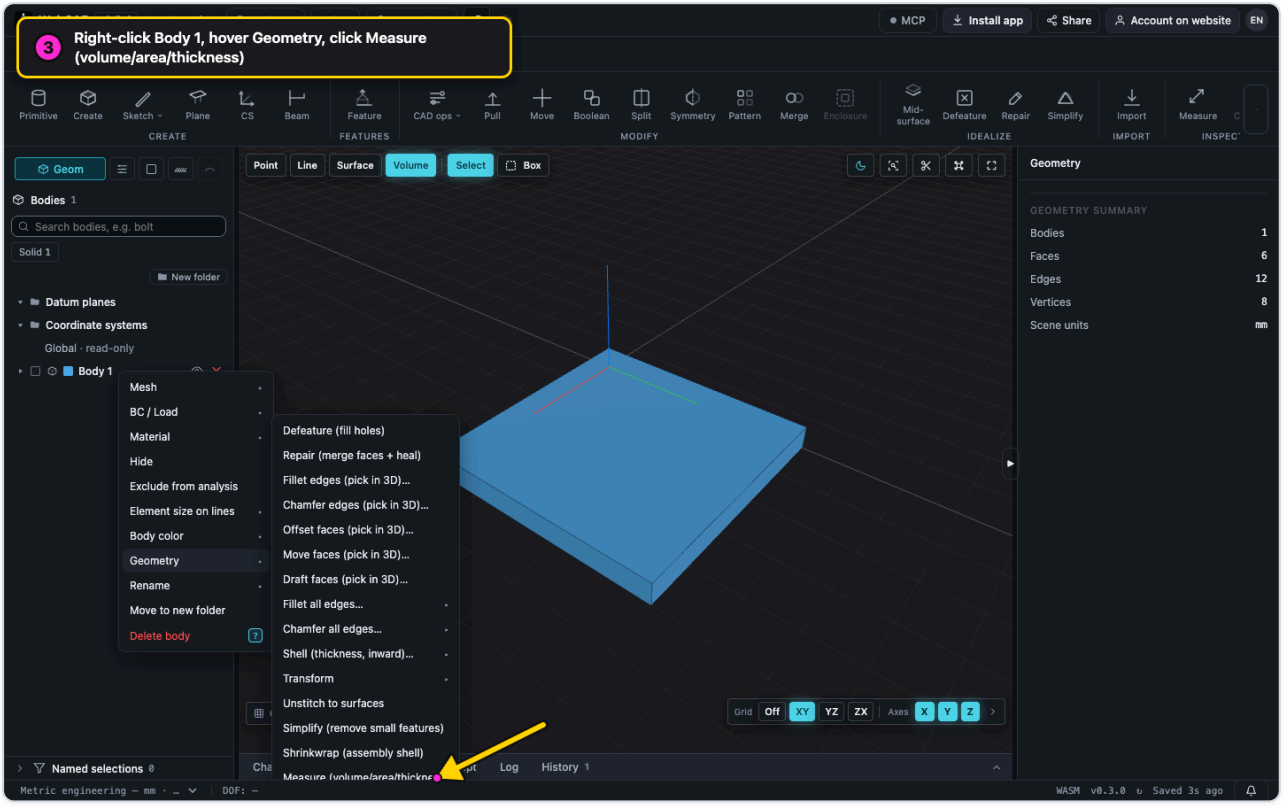
Open the **Geometry** tab and click **Primitive** in the ribbon. The default shape is a box. Type `100` in **dx**, `100` in **dy** and `10` in **dz**, then click **OK**. Body 1 appears in the tree.



STEP 3

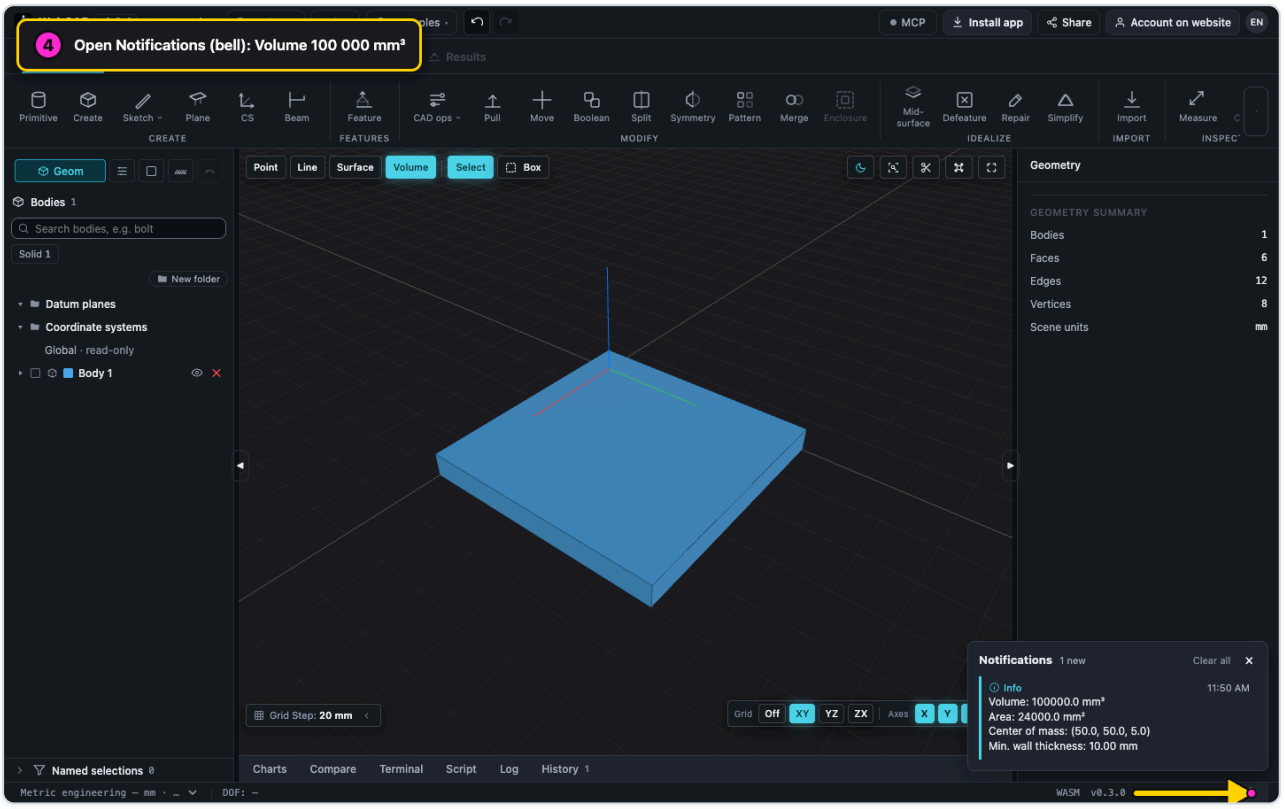
Open the Measure command of Body 1

Right-click **Body 1** in the tree, hover **Geometry** and click **Measure (volume/area/thickness)** at the bottom of the submenu.



STEP 4 Read the plate volume

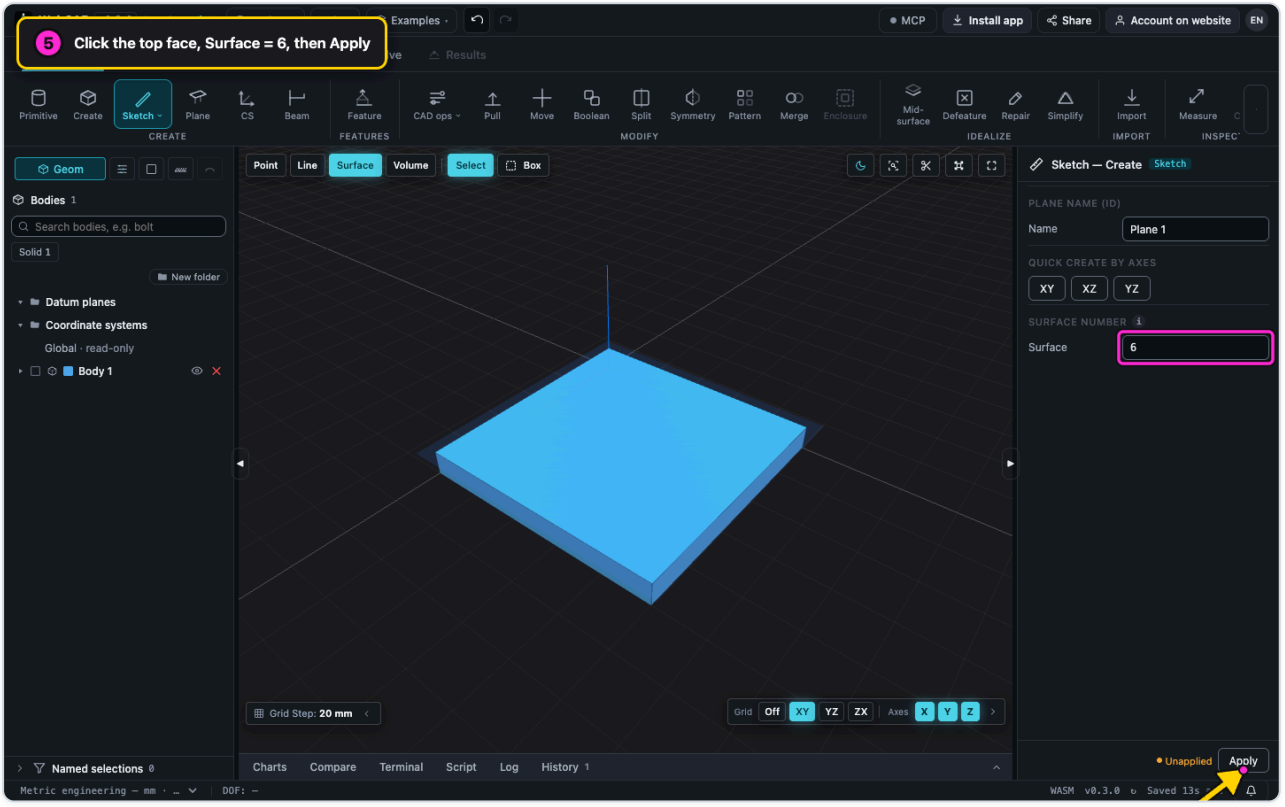
Click the bell at the bottom right to open **Notifications**. The newest entry lists **Volume**: 100 000 mm^3 (the plate: $100 \times 100 \times 10 = 100\,000$).



STEP 5

Create a sketch plane on the top face

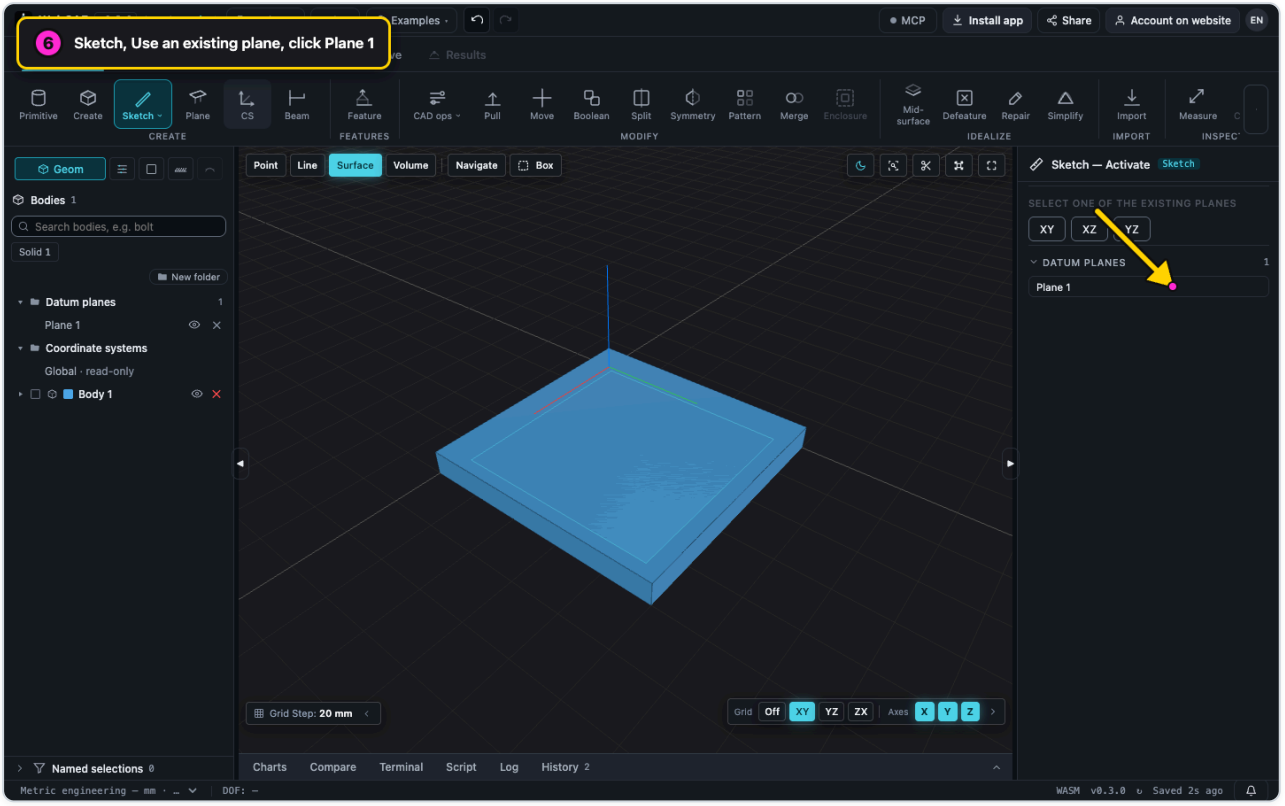
Click **Sketch** in the ribbon and choose **Create sketch** in the flyout. Click the top face of the plate in the viewport (the face at $Z = 10$): its number, **6**, appears in the **Surface** field (on a freshly opened page the plate's top face is **6**; the numbers of faces belong to the history of the document, so if yours differs, for example after reloading the page, use the number the field shows — it is the one you type in Step 7). Click **Apply**. A datum plane named **Plane 1** is created.



STEP 6

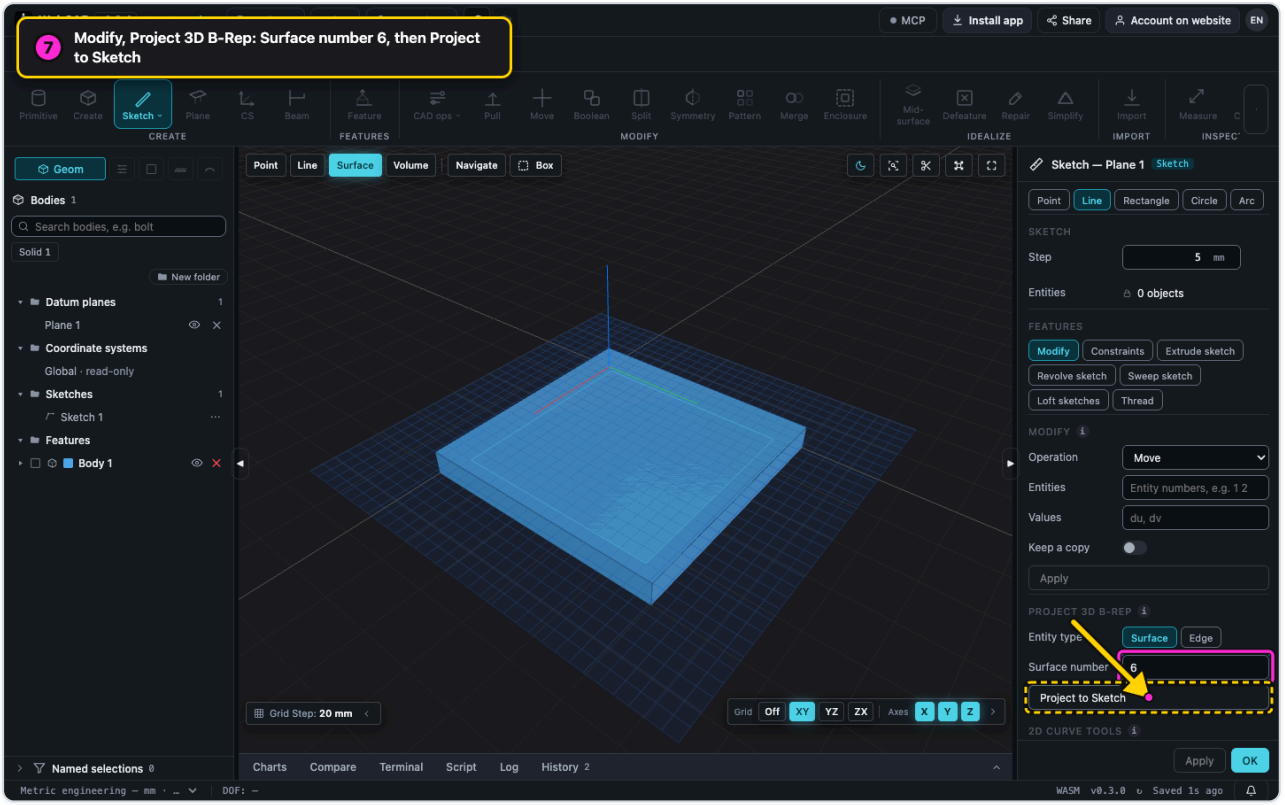
Open the sketch on Plane 1

Click **Sketch** again, choose **Use an existing plane** in the flyout and click **Plane 1** in the panel. The sketch session opens (panel title **Sketch — Plane 1**).



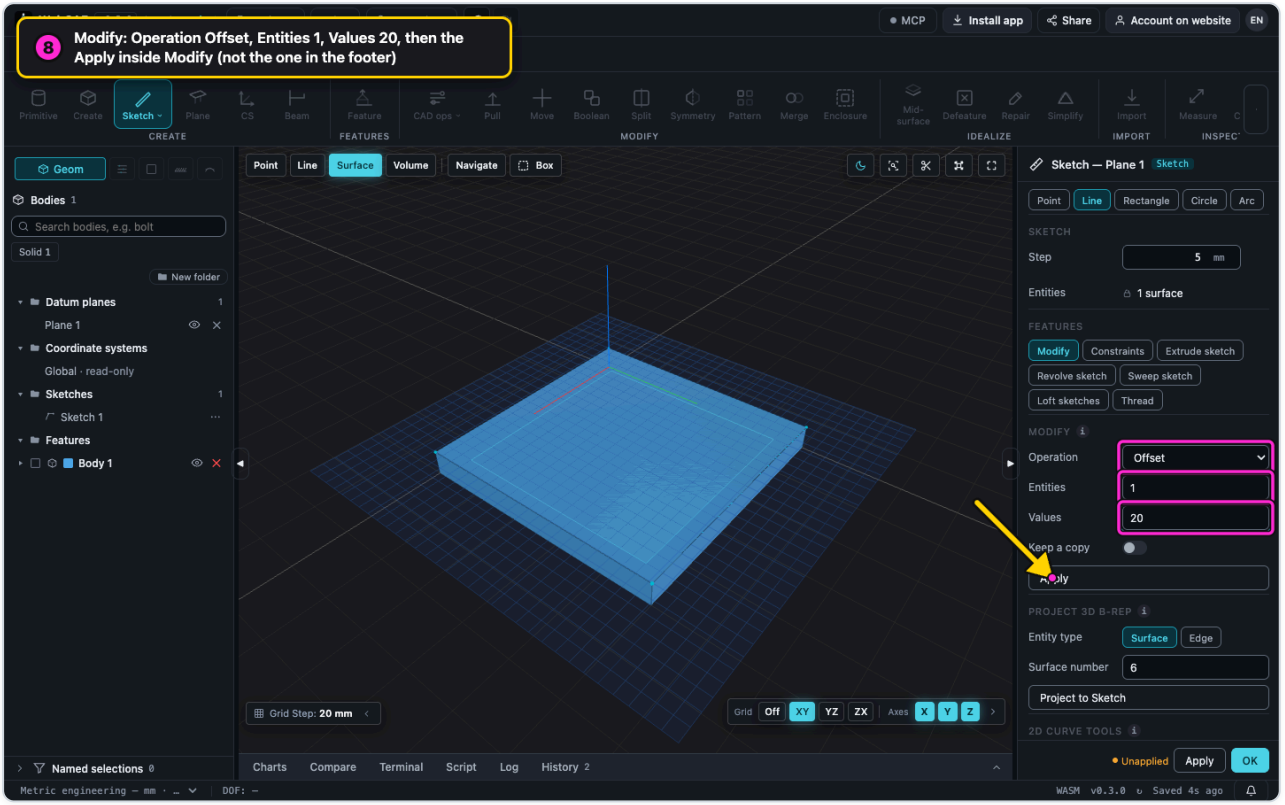
STEP 7 Project the top face into the sketch

In the sketch panel click **Modify**. In the section **Project 3D B-Rep** keep **Entity type** = **Surface**, type `6` in **Surface number** and click **Project to Sketch**. The 100×100 mm outline of the top face becomes sketch geometry.



STEP 8 Offset the outline inward by 20 mm

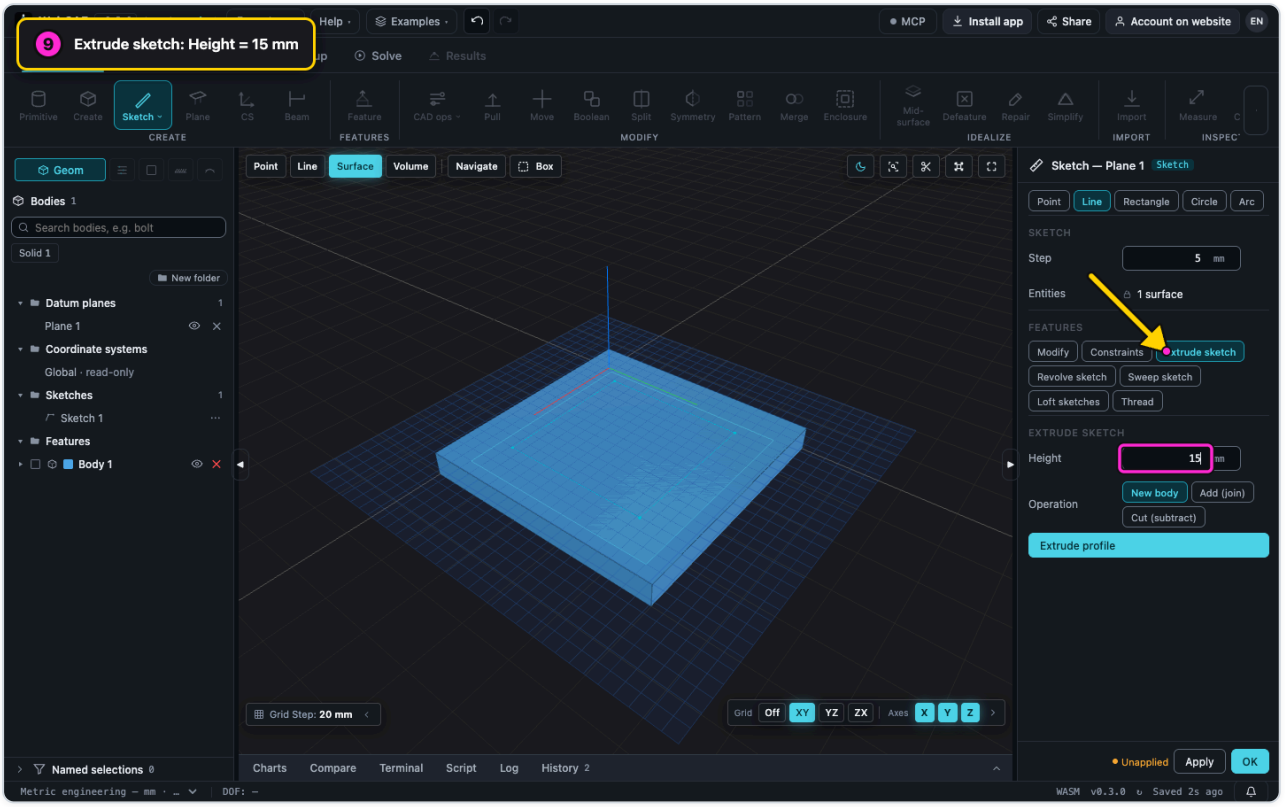
In the section **Modify** set **Operation** to **Offset**, type `1` in **Entities** (the projected outline), `20` in **Values**, leave **Keep a copy** off, and click the **Apply** button inside the Modify section (the one above **Project 3D B-Rep**, not the **Apply** in the panel footer). The outline shrinks to a 60×60 mm loop (area 3600 mm^2).



STEP 9

Open Extrude sketch and set the height

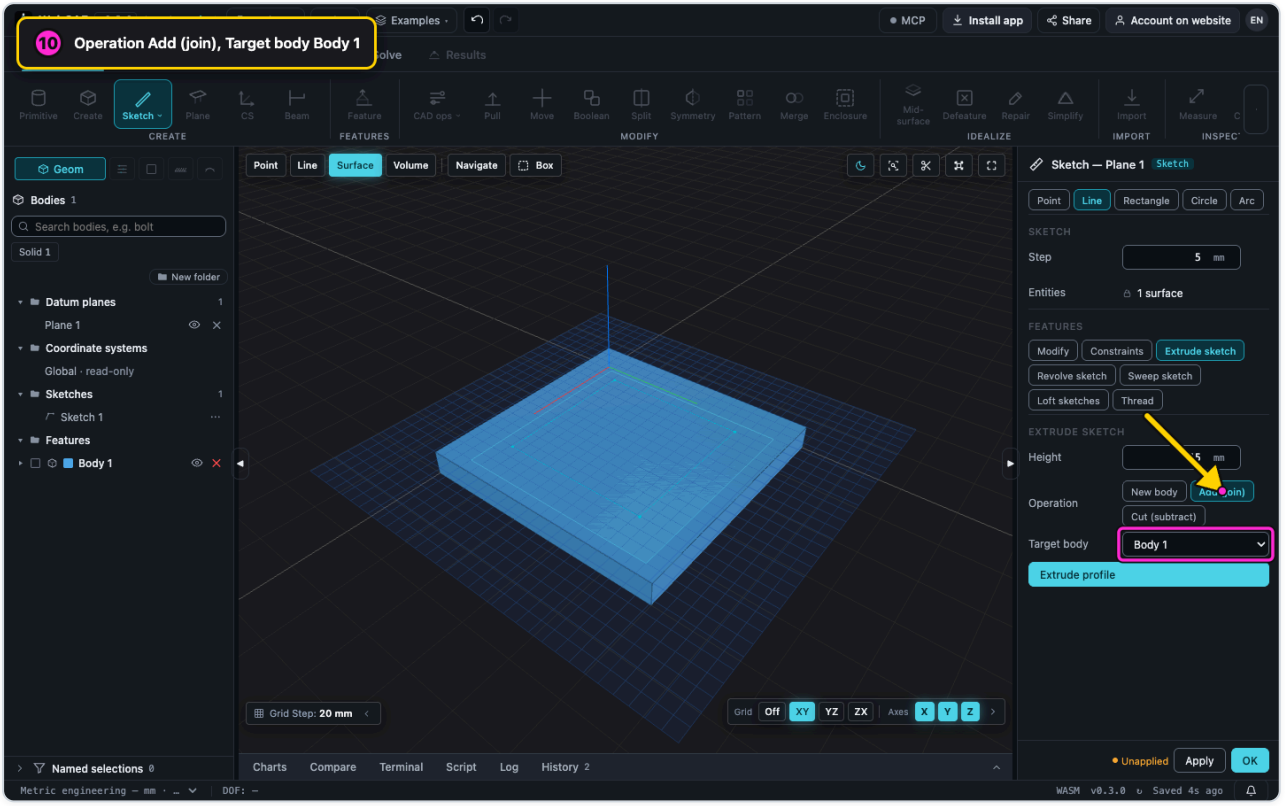
Click **Extrude sketch**. In the section **Extrude sketch** type `15` in **Height**.



STEP 10

Join the boss to Body 1

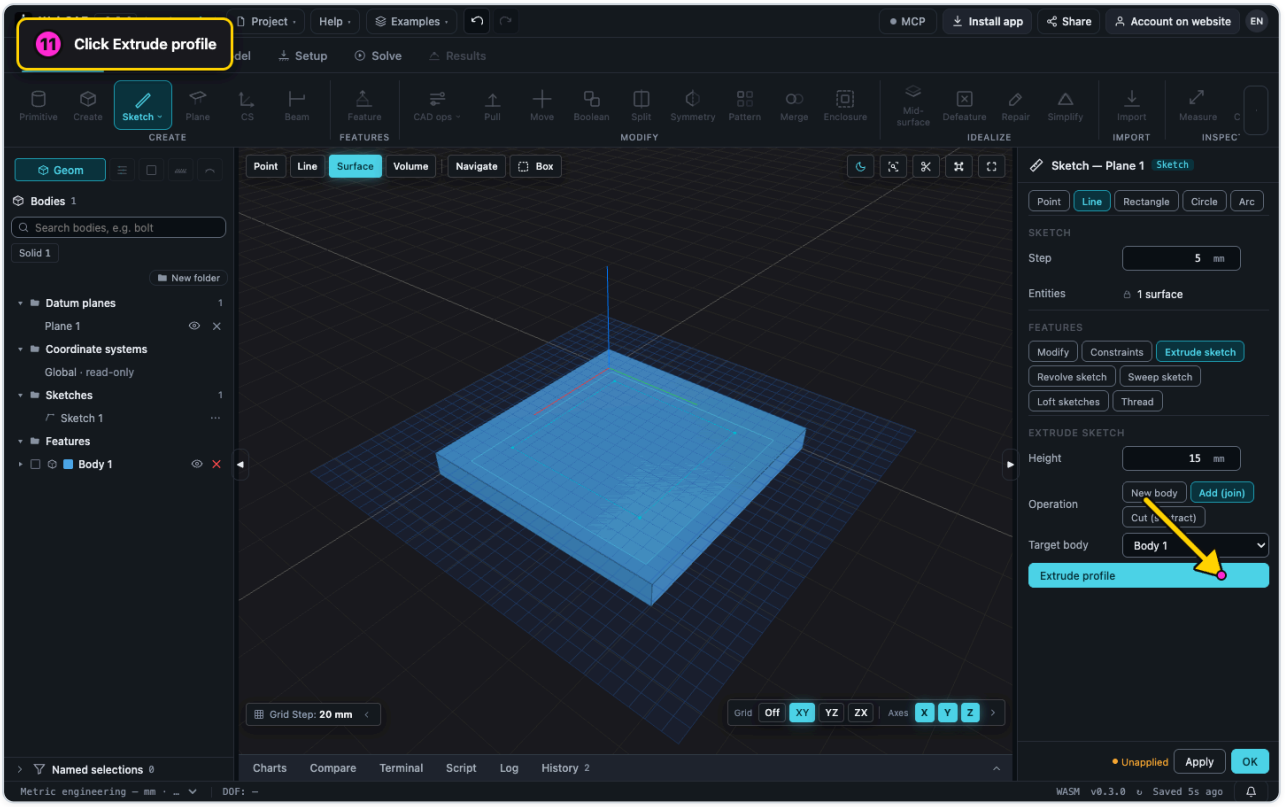
Click **Add (join)** in **Operation**, and choose **Body 1** in the **Target body** list (the default `— Auto (first body)` — also picks Body 1 here, but with several bodies choose explicitly).



STEP 11

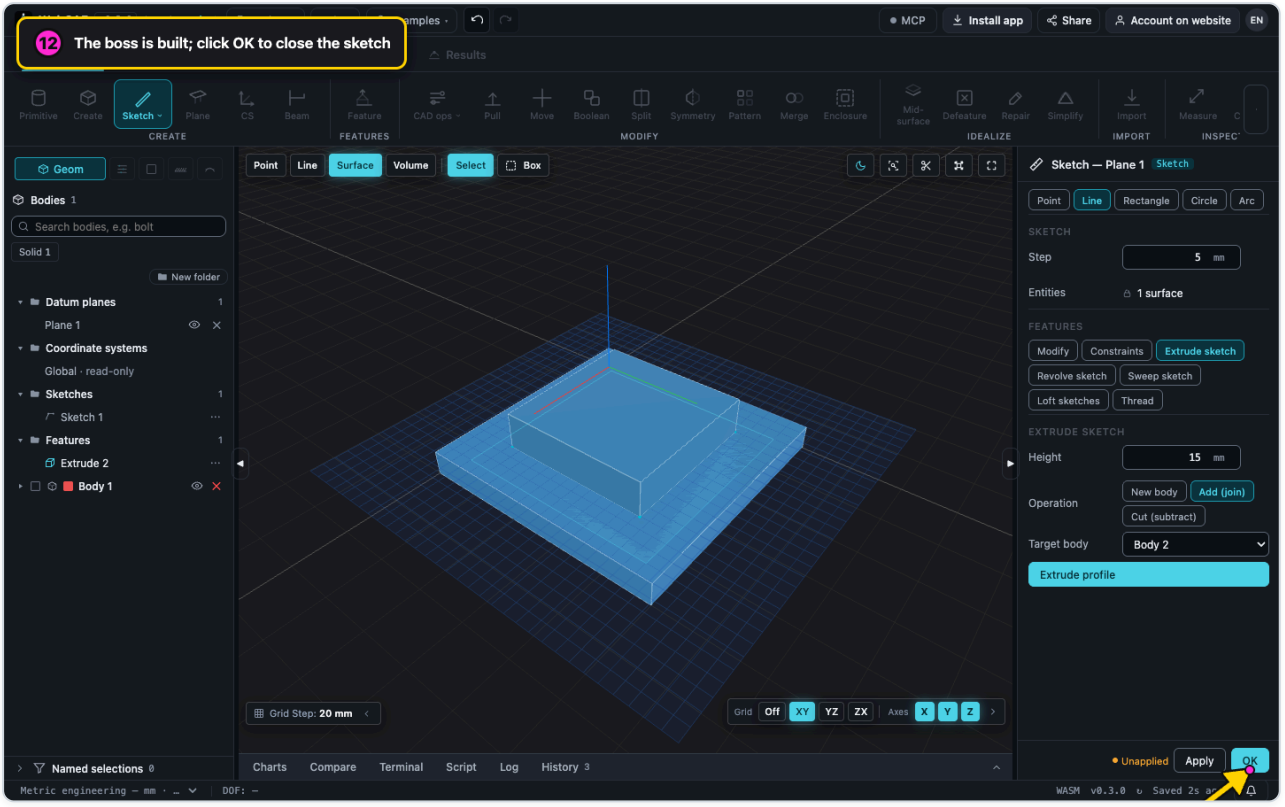
Extrude the profile

Click **Extrude profile**. The boss is built on the plate.



STEP 12 Close the sketch

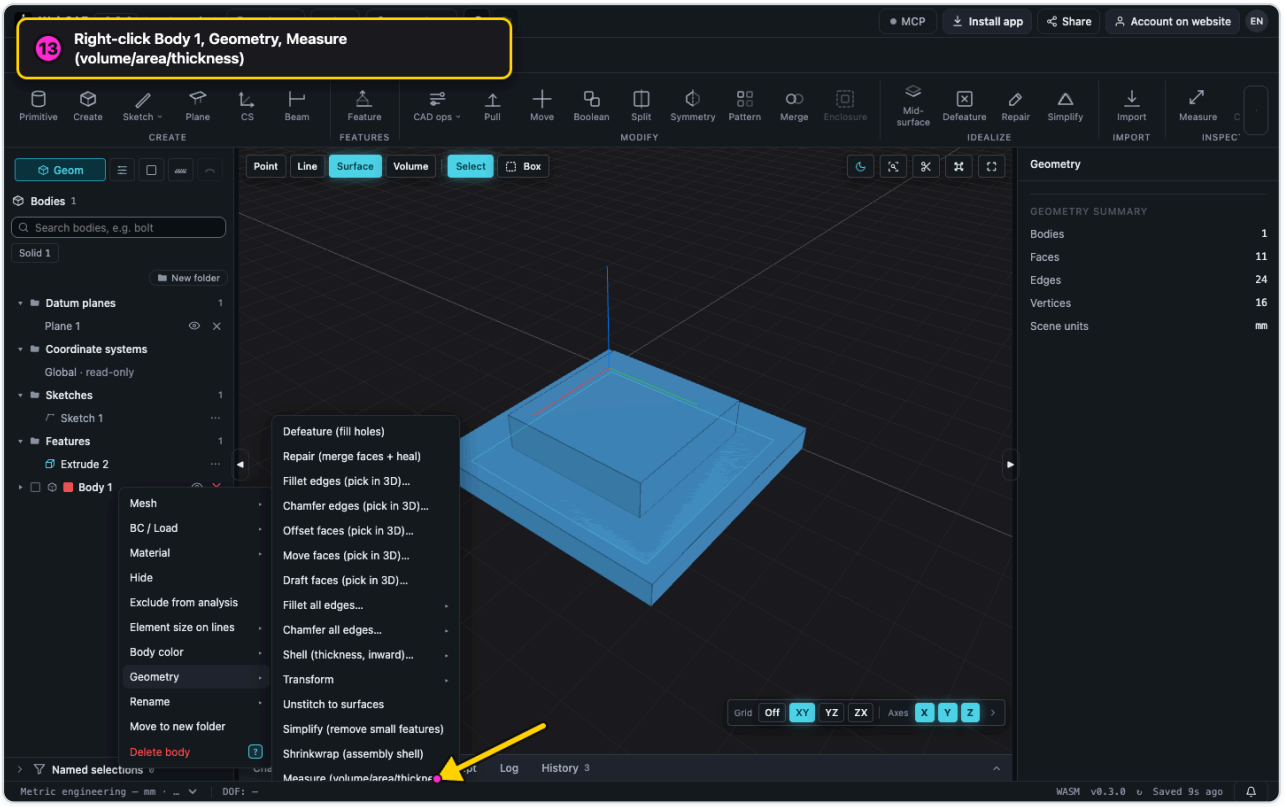
The viewport already shows the boss. The **Target body** list now reads **Body 2**: WebCAE numbers the rebuilt plate anew, while the tree still calls it **Body 1**; it is the same single body. Click **OK** in the panel footer to close the sketch session.



STEP 13

Measure the finished body

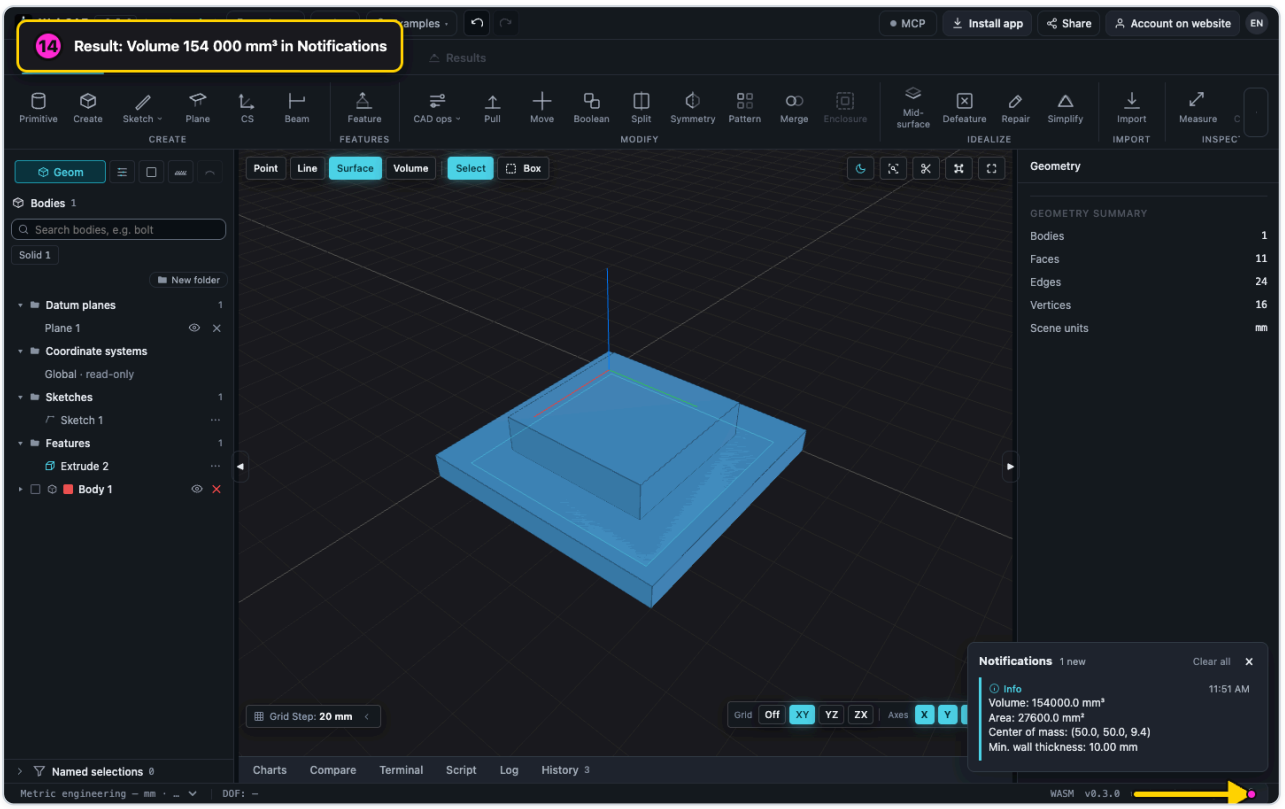
Right-click **Body 1**, hover **Geometry** and click **Measure (volume/area/thickness)** again.



STEP 14

Result: volume of the plate with the boss

Open **Notifications** (bell). **Volume**: 154 000 mm^3; the body is one piece ($100 \times 100 \times 25$ mm at the boss).



VERIFICATION BENCHMARK

Check against the geometry

Metric	Analytical	WebCAE web (browser)	WebCAE desktop (native)	Tolerance	Status
Plate volume	$100 \times 100 \times 10 = 100000$ mm^3	100 000 mm^3	— (geometry is OpenCASCADE in WASM, no native entry)	1 %	PASS
Volume with boss	$100000 + 60 \times 60 \times 15 = 154000$ mm^3	154 000 mm^3	—	2 %	PASS
Bodies after Add (join)	1	1	—	—	PASS

The added volume is $\Delta V = 60 \times 60 \times 15 = 54\,000\ \mathrm{mm}^3$; the numbers are read from **Measure (volume/area/thickness)** in Notifications, steps 3 and 13.

TROUBLESHOOTING & NOTES

Common mistakes

- **Using New body instead of Add (join):** the boss becomes a second body and the volume of Body 1 stays 100 000 mm^3.
- **Pressing the wrong Apply in step 7:** the Apply in the panel footer finishes the whole sketch, the Apply inside Modify performs the Offset.
- **Skipping Use an existing plane:** the sketch plane only exists after step 4; drawing starts after step 5.
- **Offset sign:** a positive value on the projected closed outline insets it.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#12 — Thin-Walled Housing, Direct Modeling & Measure 2.0 (F12)**.

WebCAE 101

LESSON 12 · F12

Thin-walled housing, direct modeling offset and measure

Goal Make an open thin-walled housing from a solid block with Shell / hollow, thicken one wall with Offset faces, and check the volumes and the wall thickness with the Measure commands.

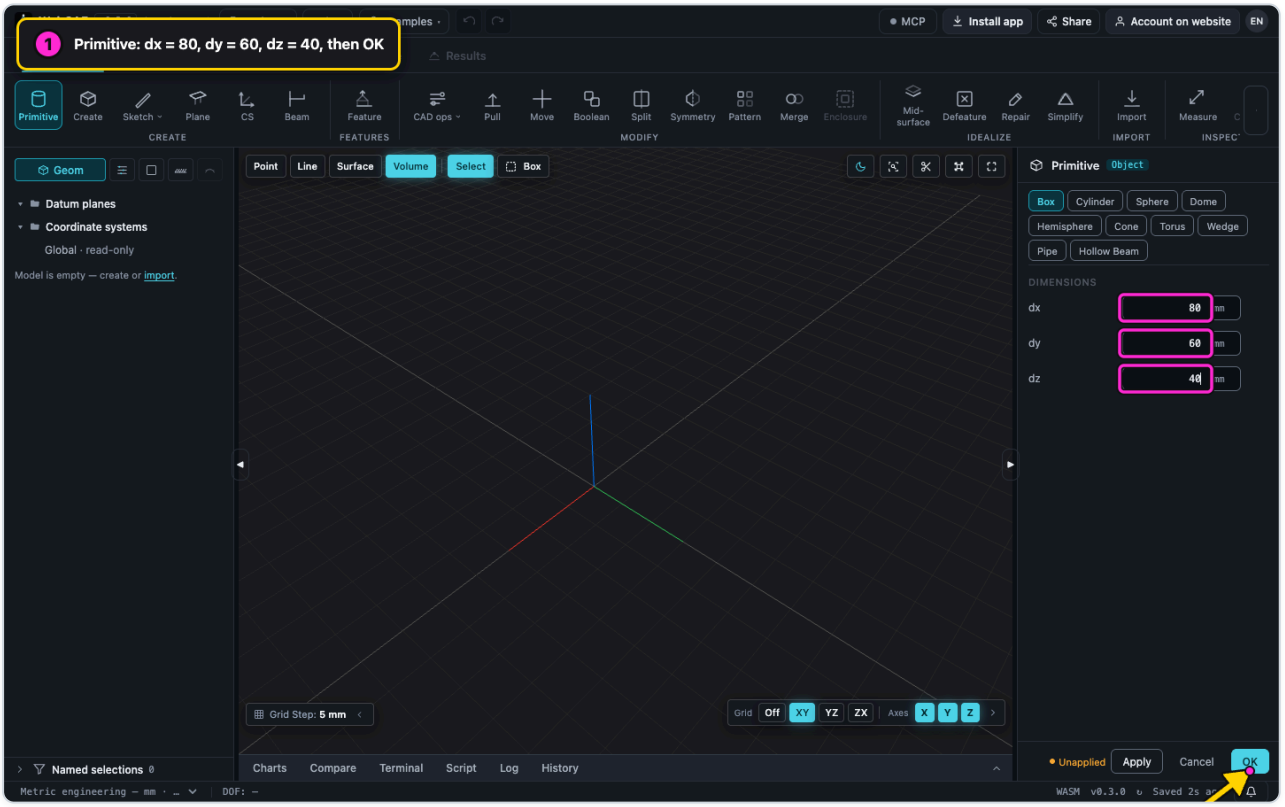
Outcome A housing with volume 57 216 mm^3, after the offset 62 016 mm^3, and a 6.000000 mm distance between the outer and the inner wall. WebCAE: 57 216, 62 016 mm^3 and 6.000000 mm.

5–6 min | WebCAE web (browser), geometry engine OpenCASCADE in WASM

Run: 2026-11-04-32ffaed2 / F12

STEP 1 Create the block

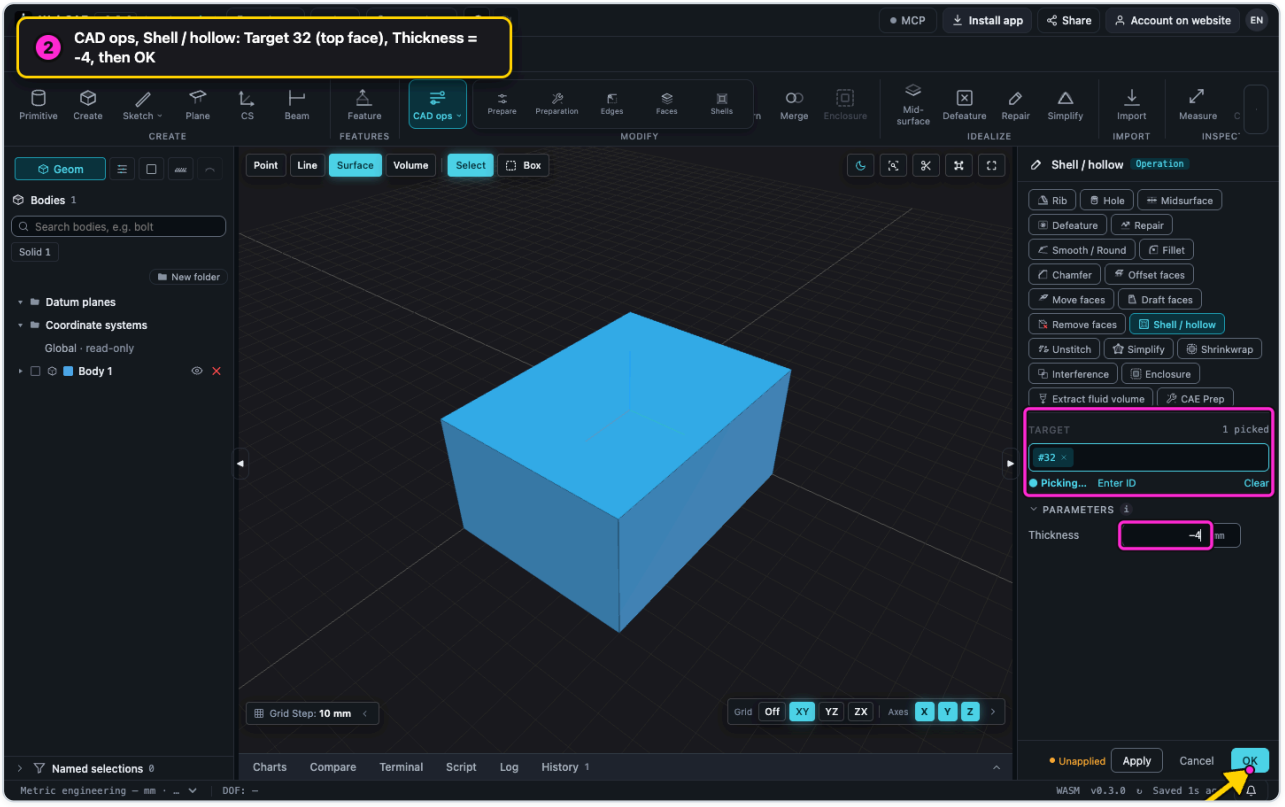
Open **Geometry**, click **Primitive** (shape **Box**), type `80` in **dx**, `60` in **dy**, `40` in **dz**, click **OK**. Body 1 appears in the tree.



STEP 2

Hollow the block, top face open

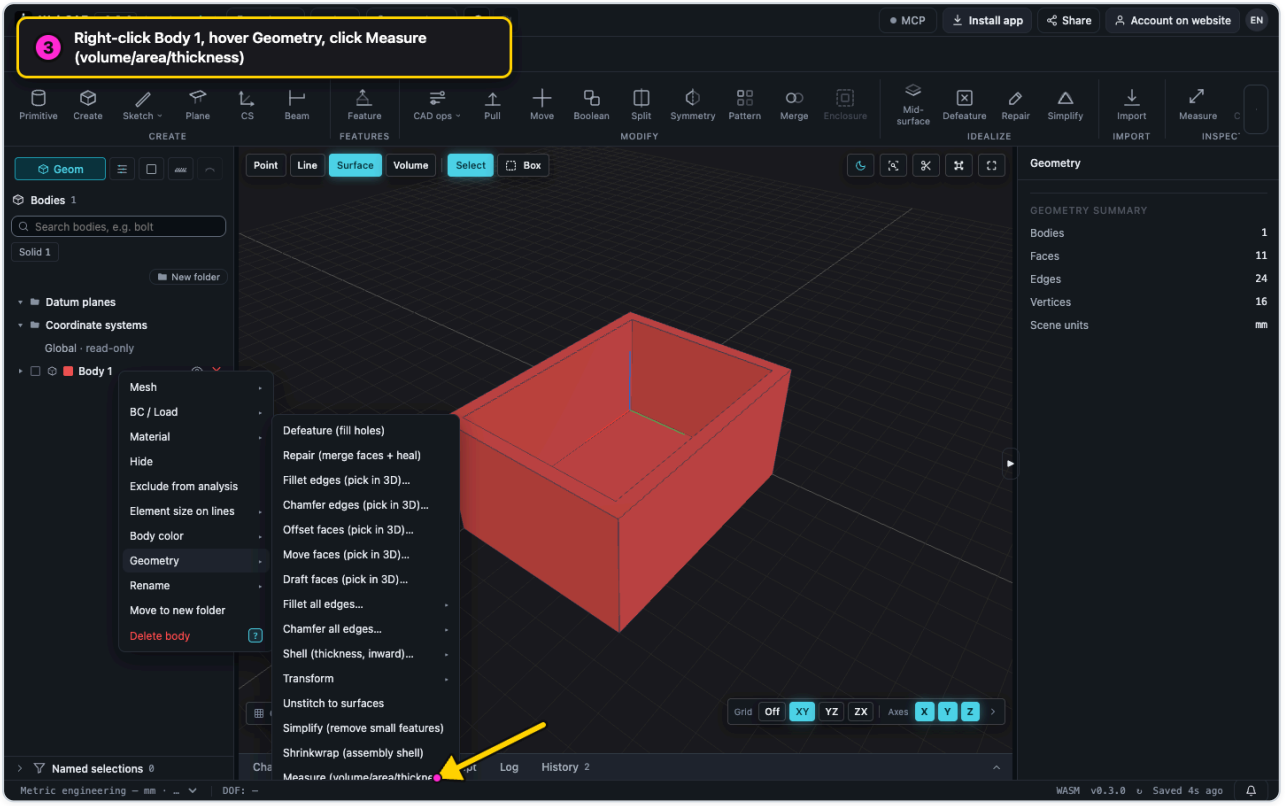
Click **CAD ops** in the ribbon and choose **Shell / hollow** in the panel. In **Target** click **Enter ID**, type `32` and press Enter: surface 32 is the top face of the block (Normal +Z, at $Z = 40$ mm, area 4 800 mm^2); it stays open. Type `−4` in **Thickness** (negative: the wall grows inward), then click **OK**.



STEP 3

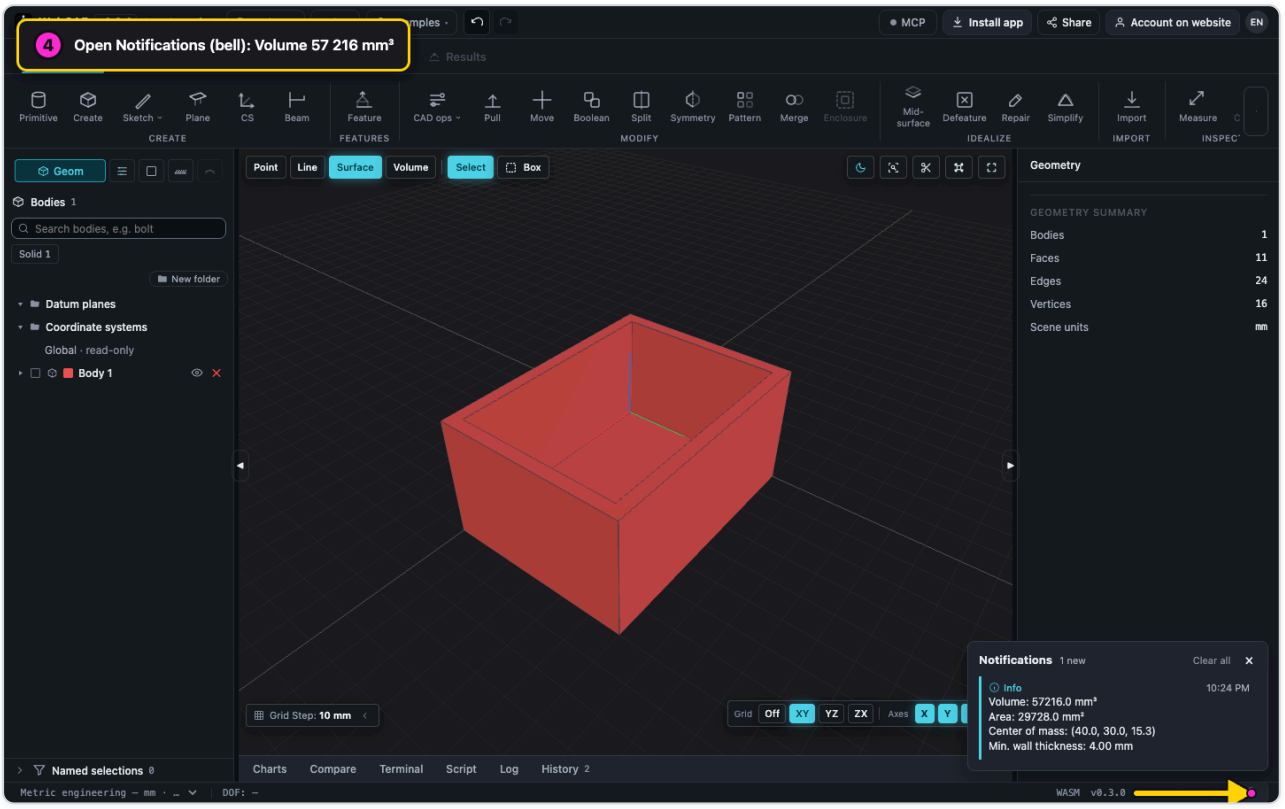
Open the Measure command of Body 1

Right-click **Body 1** in the tree, hover **Geometry** and click **Measure (volume/area/thickness)** at the bottom of the submenu.



STEP 4 Read the housing volume

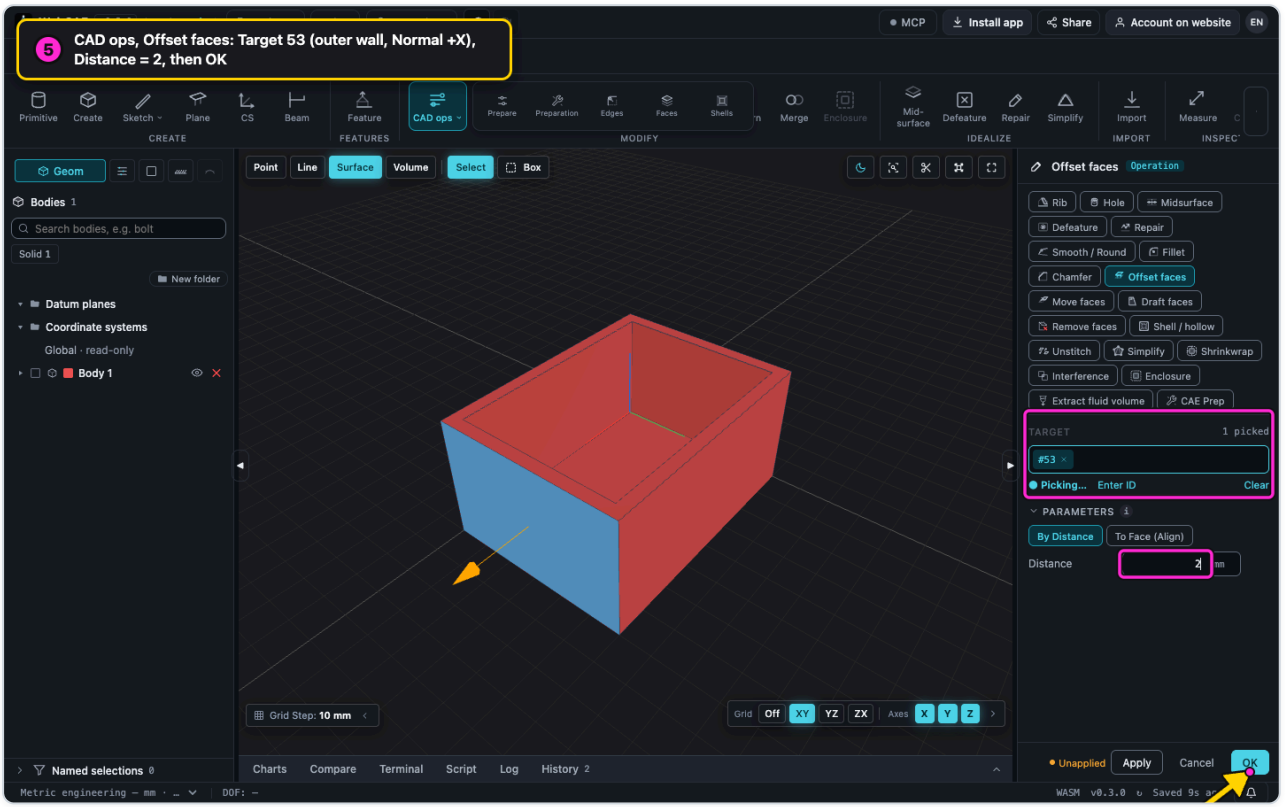
Click the bell at the bottom right to open **Notifications**. **Volume**: 57 216 mm^3. By hand: the block $80 \times 60 \times 40 = 192\,000$ minus the cavity $72 \times 52 \times 36 = 134\,784$ gives $57\,216$.



STEP 5

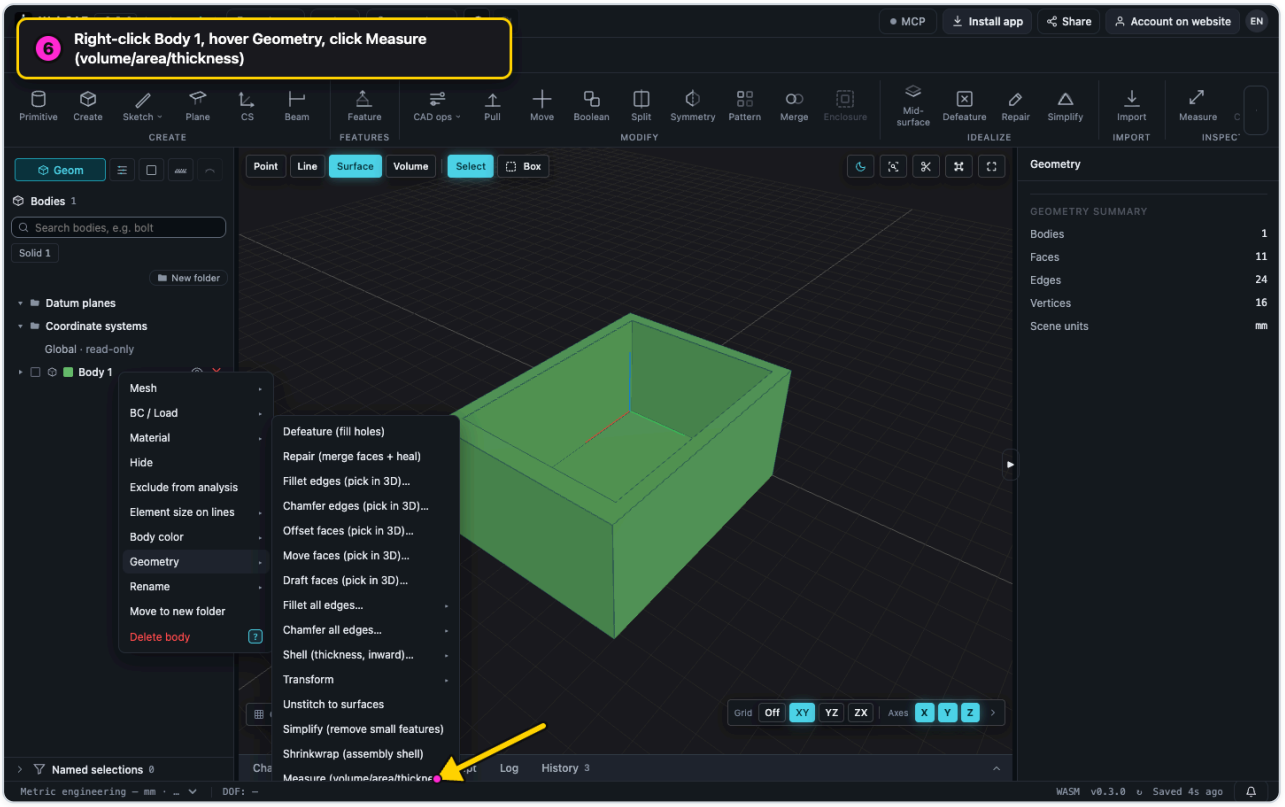
Thicken the right wall with Offset faces

Click **CAD ops** and choose **Offset faces**. In **Target** click **Enter ID**, type `53` and press Enter: surface 53 is the outer wall whose Normal points along +X (the wall at $X = 80$ mm, area 2 400 mm^2); it highlights in the viewport. Keep **By Distance**, type `2` in **Distance** and click **OK**. The wall moves outward by 2 mm: 4 mm becomes 6 mm.



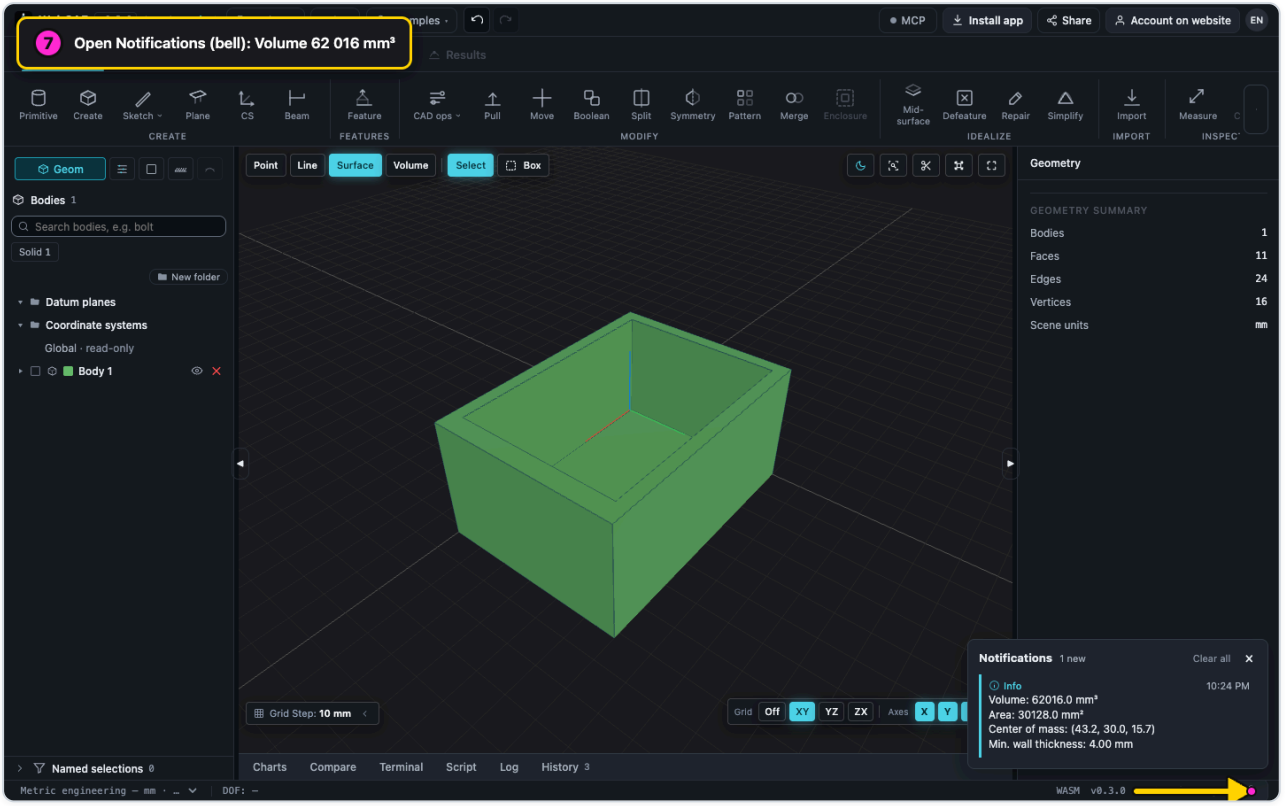
STEP 6 Open the Measure command of Body 1 again

Right-click **Body 1**, hover **Geometry**, click **Measure (volume/area/thickness)**.



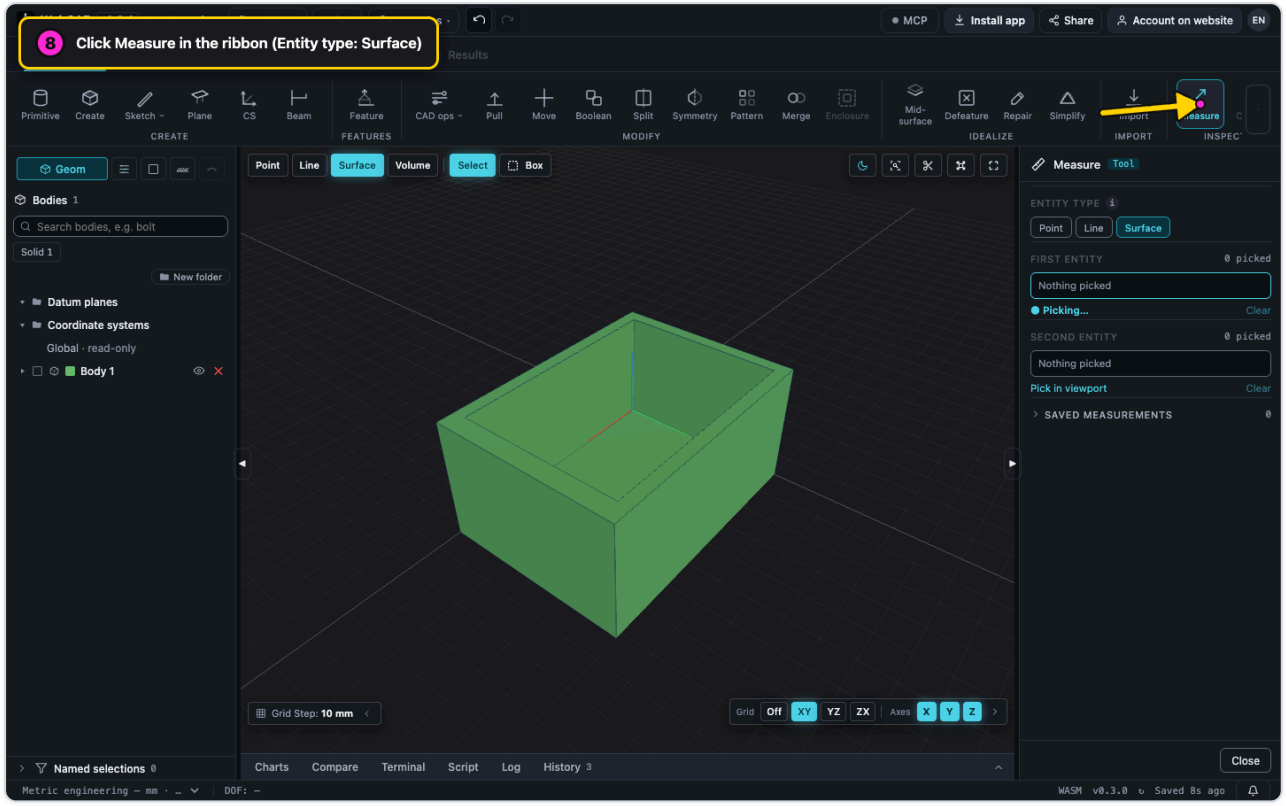
STEP 7 Read the volume after the offset

Open **Notifications** (bell): **Volume** 62 016 mm^3. By hand: $57\,216 + 60 \times 40 \times 2 = 62\,016$.



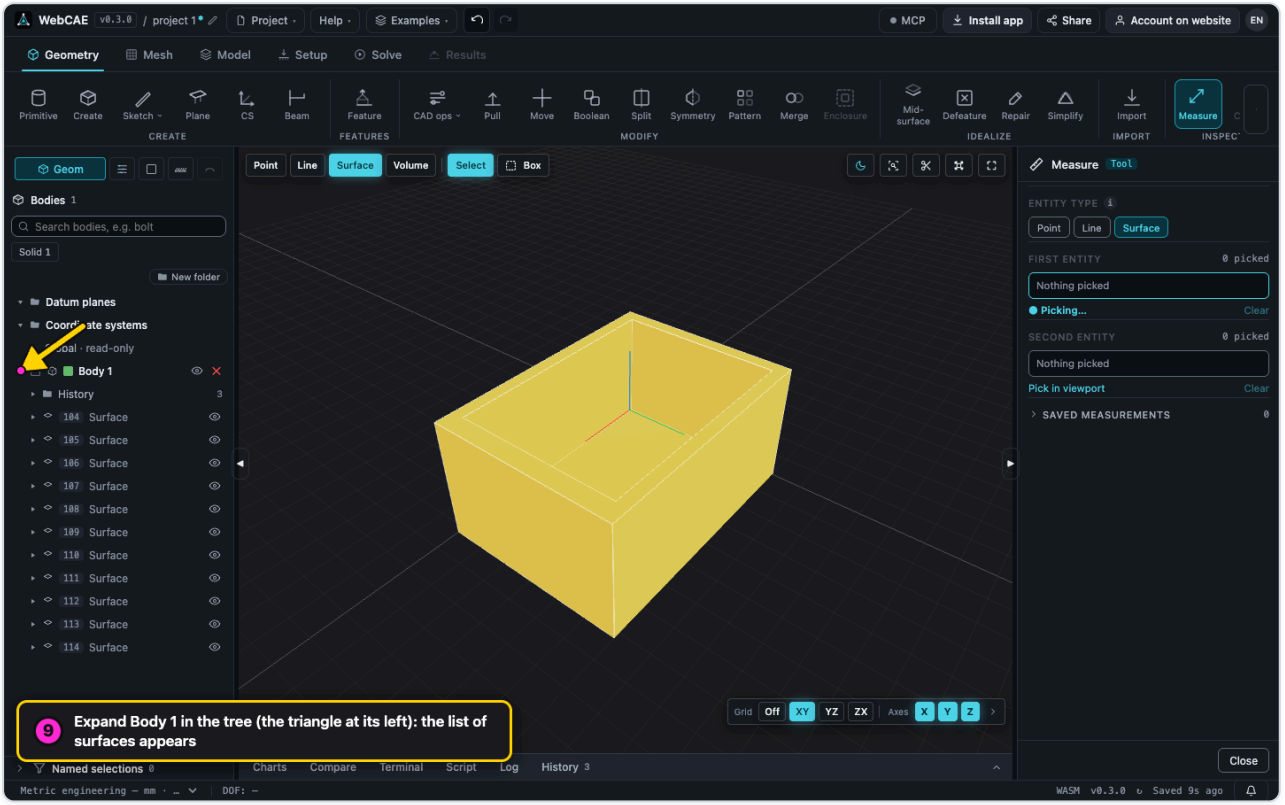
STEP 8 Open the Measure tool

Click **Measure** in the ribbon. The panel **Measure** opens with **Entity type** = **Surface**; both entity slots say **Nothing picked**.



STEP 9 Expand Body 1 in the tree

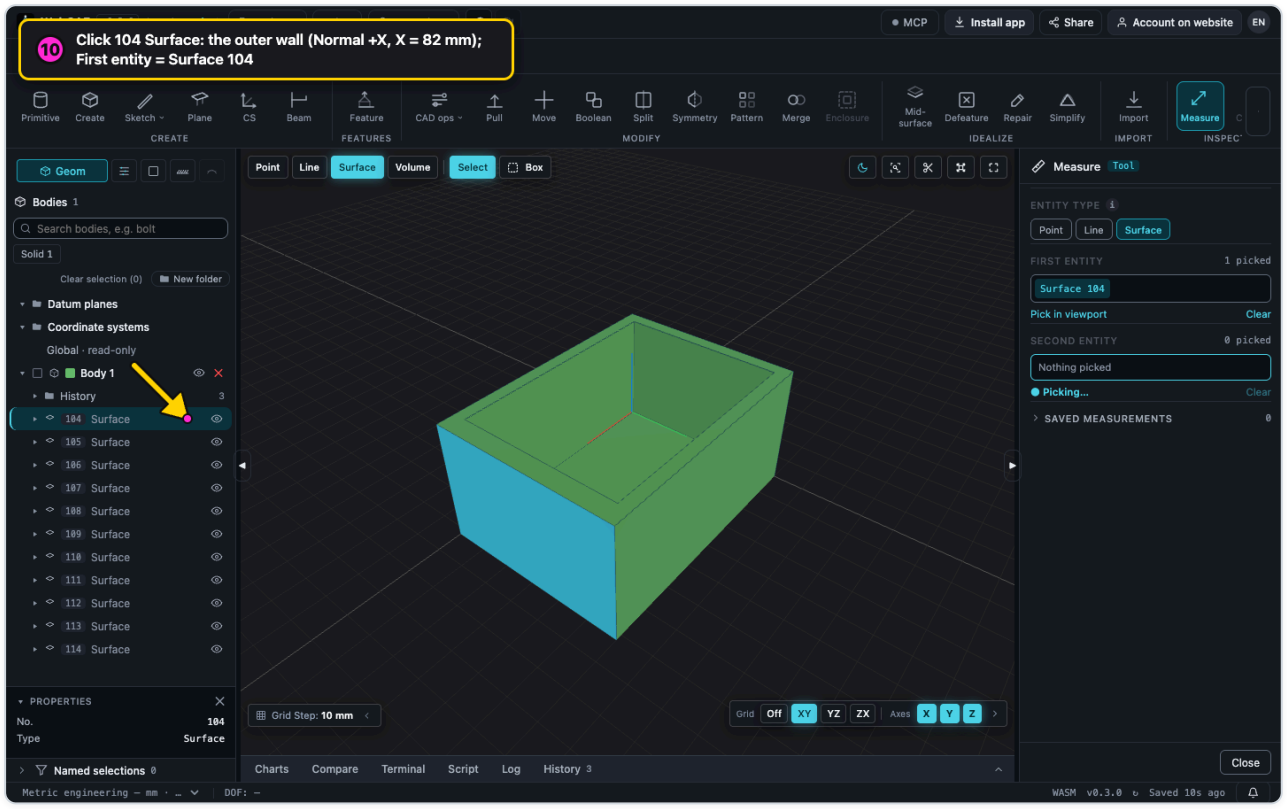
Click the triangle at the left of **Body 1**. The list of surfaces appears. Surface numbers changed after the two operations; the walls are now **104** and **110** (every operation renumbers surfaces, so always read the numbers in this list).



STEP 10

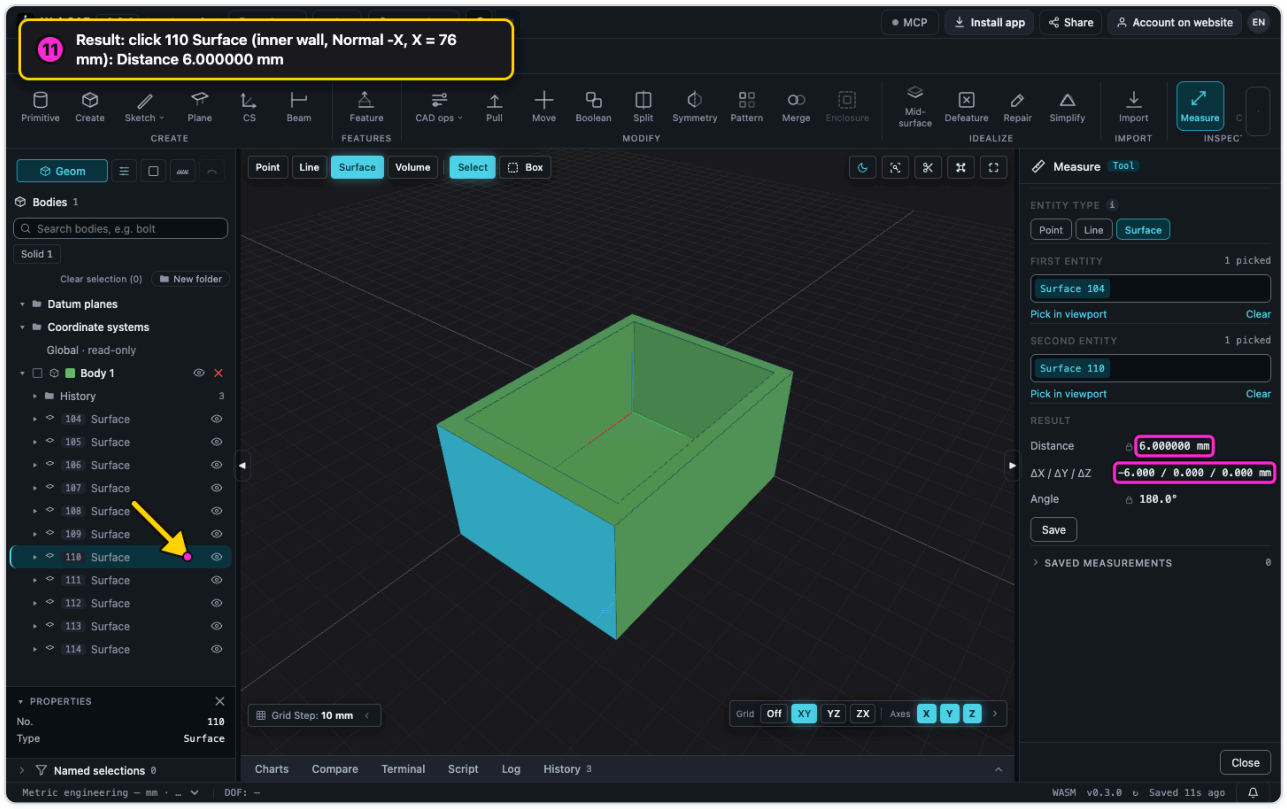
Pick the outer wall

Click **104 Surface**. This is the outer right wall after the offset (Normal +X, $X = 82$ mm); it highlights in the viewport and **First entity** reads **Surface 104**.



STEP 11 Result: pick the inner wall and read the distance

Click **110 Surface**: the inner face of the same wall (Normal –X, $X = 76$ mm). The panel shows **Distance** 6.000000 mm and **ΔX / ΔY / ΔZ** -6.000 / 0.000 / 0.000 mm; the faces are parallel (**Angle** 180.0°).



VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Difference
Housing volume (Shell −4, top open)	$192000 - 72 \times 52 \times 36 = 57216$ mm^3	57 216 mm^3	— (geometry is OpenCASCADE in WASM, no native entry)	0.00 %
Volume after Offset faces +2 mm	$57216 + 60 \times 40 \times 2 = 62016$ mm^3	62 016 mm^3	—	0.00 %
Wall thickness (surface 104 to 110)	$82 - 76 = 6$ mm	6.000000 mm	—	0.00 %

FORMULA

Cavity $V_c = (L - 2t)(W - 2t)(H - t) = 72 \times 52 \times 36 = 134\,784\ \mathrm{mm}^3$, material $V = 192\,000 - V_c = 57\,216\ \mathrm{mm}^3$. The wall offset adds $A \cdot d = 2400 \times 2 = 4800\ \mathrm{mm}^3$. The first number (192 000) is a hand calculation, it is not shown by WebCAE in this lesson.

TROUBLESHOOTING & NOTES

Common mistakes

- **Positive thickness in Shell / hollow:** the wall grows outward and the block stays solid-looking; inward walls need `−4`.
- **Wrong wall in Offset faces:** the target must be the outer wall (Normal +X); picking the inner face moves the cavity wall instead.
- **Old face numbers after an operation:** every operation renumbers surfaces; take numbers from the tree, as in step 8.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#13 — Hub & Pin Assembly with Circular Pattern (F13)**.

WebCAE 101

LESSON 13 · F13

Hub with a Bore and Four Pins (Circular Pattern)

Goal Model a cylindrical hub with a central bore, create one pin, move it to its place on the hub face and replicate it with a circular pattern.

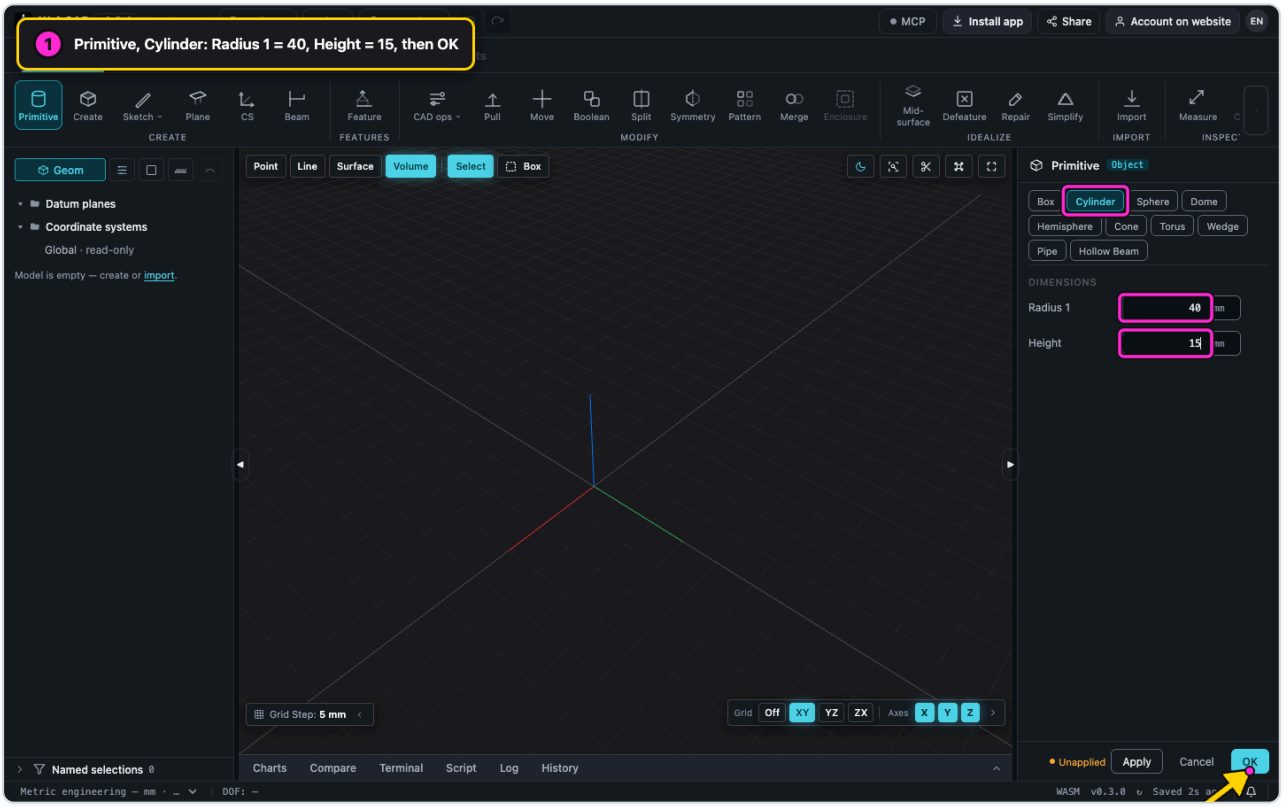
Outcome One hub body (volume 68612.4 mm^3) and four pins at a pitch radius of 26 mm, 90° apart, each 1256.6 mm^3. The pins stand on the top face of the hub: they touch it, they do not overlap it (5 bodies).

6–7 min | WebCAE web (browser), geometry engine OpenCASCADE in WASM

Run: 2026-11-04-32ffaed2 / F13

STEP 1 Create the hub blank

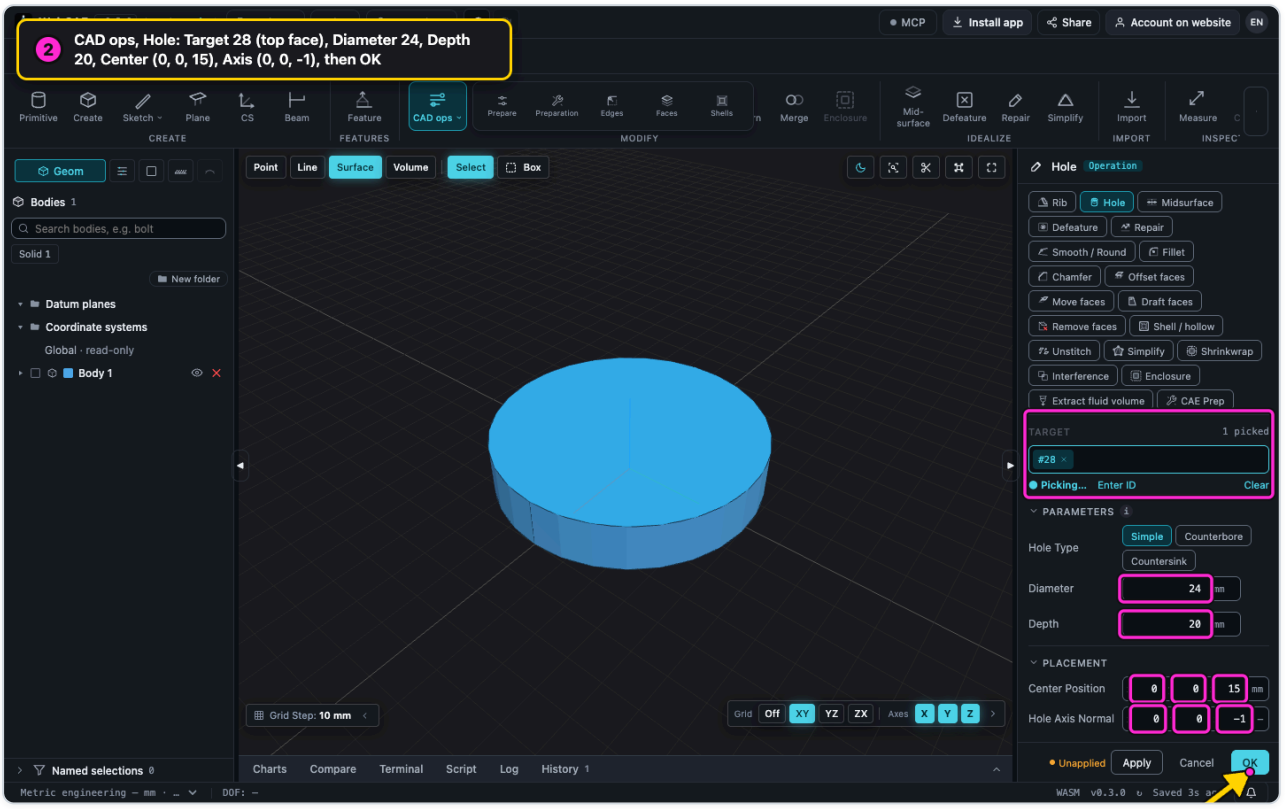
Open **Geometry**, click **Primitive**, choose **Cylinder**. Type `40` in **Radius 1** and `15` in **Height**, click **OK**. Body 1 appears.



STEP 2

Bore the central hole

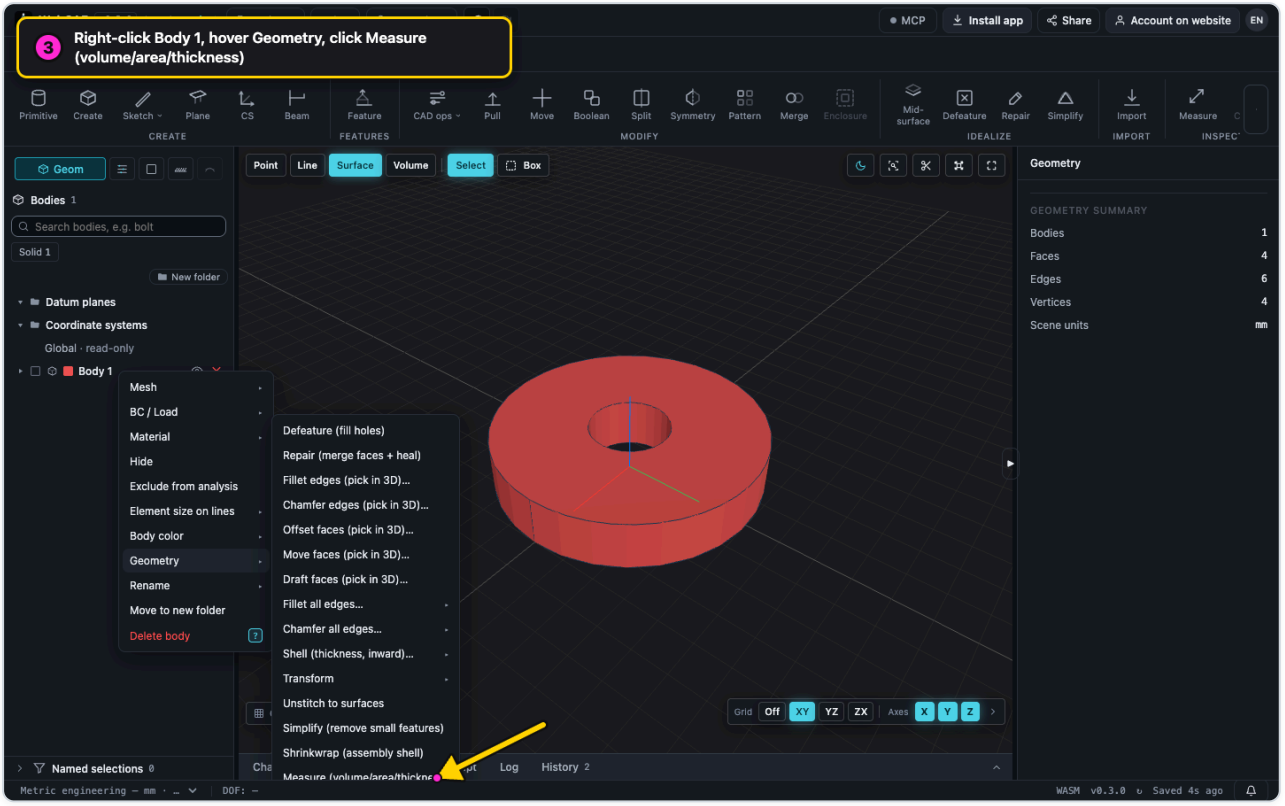
Click **CAD ops** in the ribbon and choose **Hole**. In **Target** click **Enter ID**, type `28` and press Enter: surface 28 is the top circular face of the hub (Normal +Z, at $Z = 15$ mm). Keep **Hole Type** = **Simple**. Type `24` in **Diameter** and `20` in **Depth** (deeper than the 15 mm hub: a through hole). **Center Position**: x `0`, y `0`, z `15`. **Hole Axis Normal**: x `0`, y `0`, z `-1` (the hole goes down from the top face). Click **OK**.



STEP 3

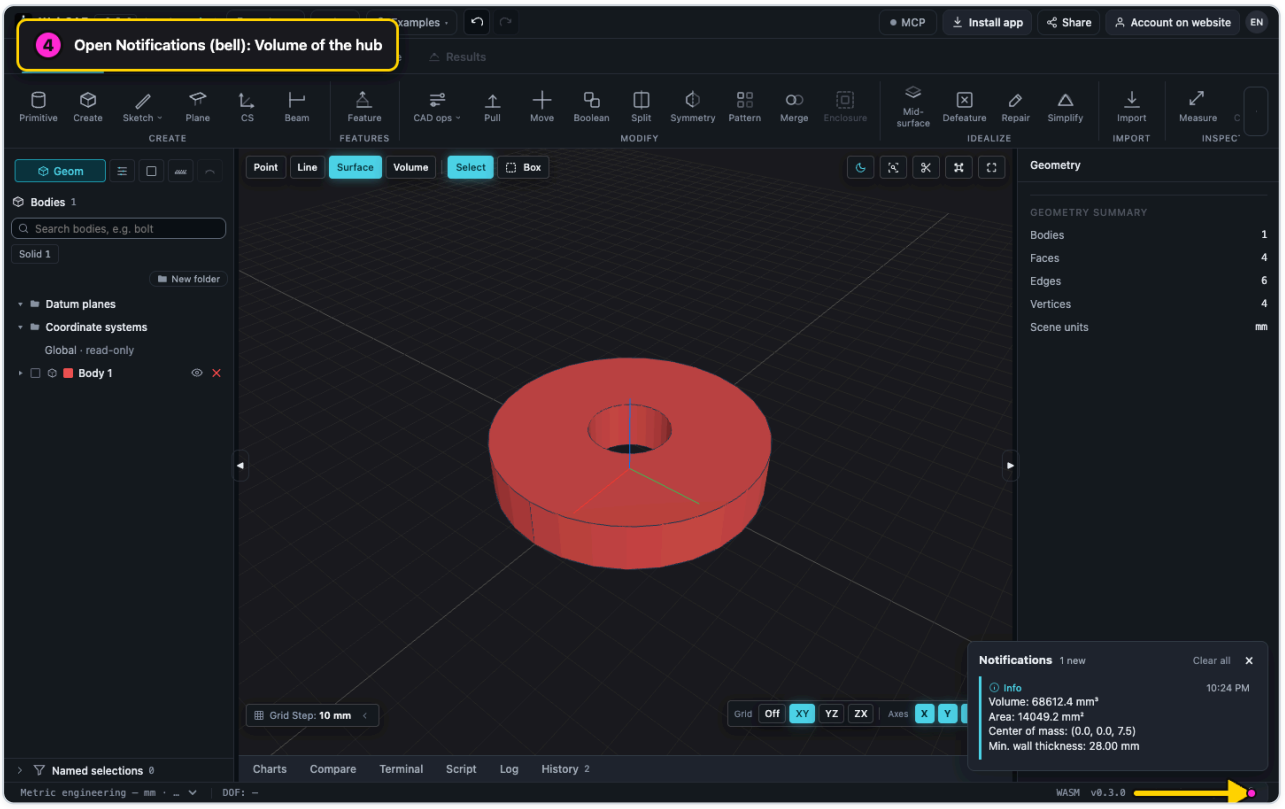
Open the Measure command of Body 1

Right-click **Body 1** in the tree, hover **Geometry** and click **Measure (volume/area/thickness)** at the bottom of the submenu.



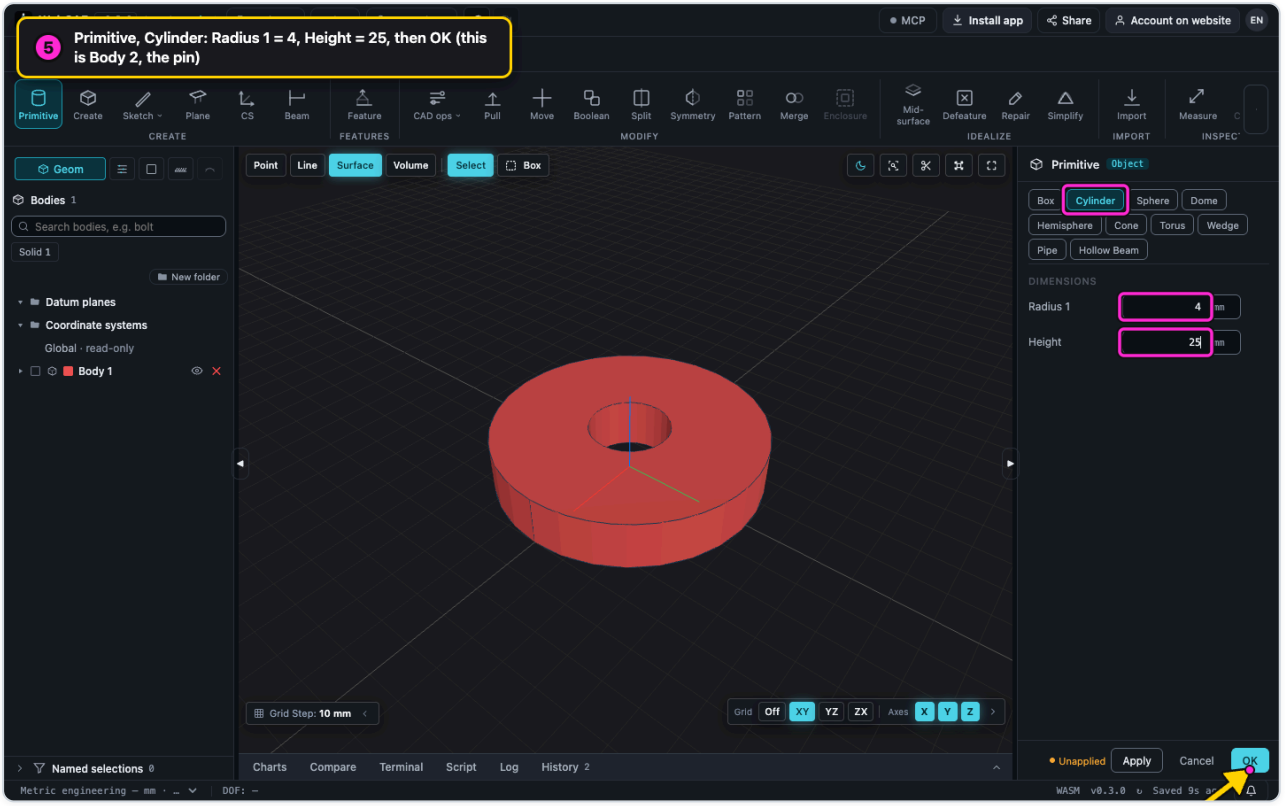
STEP 4 Read the hub volume

Click the bell at the bottom right to open **Notifications**. **Volume**: 68612.4 mm^3. Center of mass x, y = (0, 0) is not needed here; the volume is.



STEP 5 Create the pin

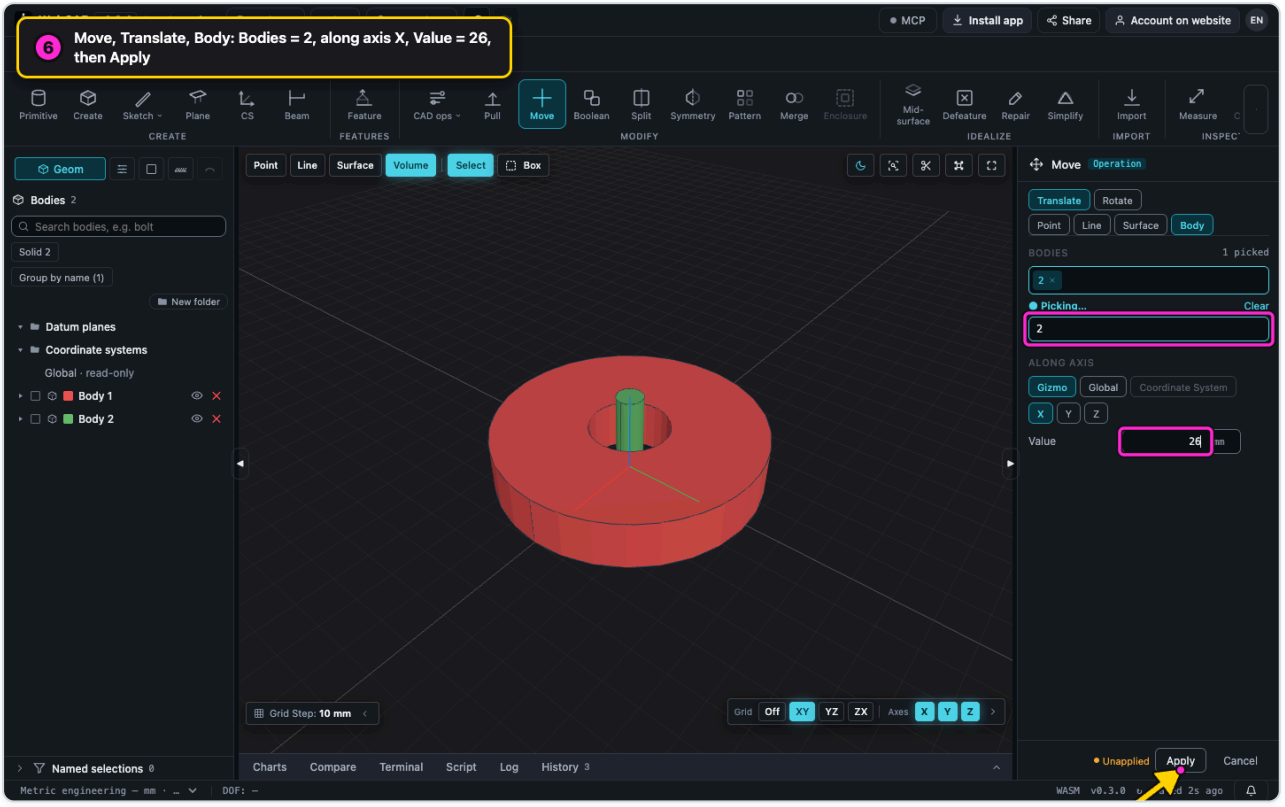
Click **Primitive**, choose **Cylinder**, type `4` in **Radius 1** and `25` in **Height**, click **OK**. This is **Body 2**; it stands at the origin, inside the bore, and will be moved.



STEP 6

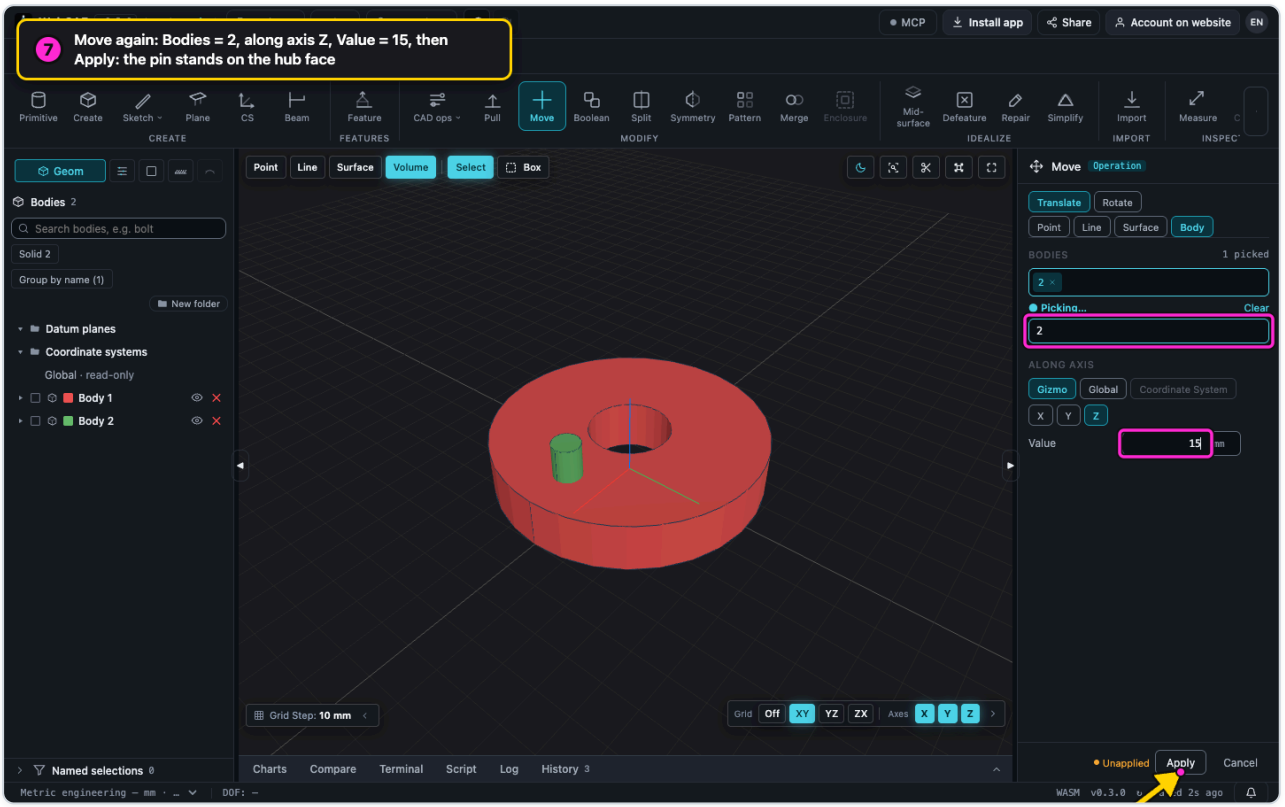
Move the pin out to the pitch circle

Click **Move** in the ribbon. In the panel keep **Translate**, click **Body** (the pick mode, not Point, Line or Surface), type `2` in the **Body #** box and press Enter, under **Along axis** keep **Gizmo** and click **X**, type `26` in **Value**, click **Apply**. The pin is now 26 mm from the hub axis.



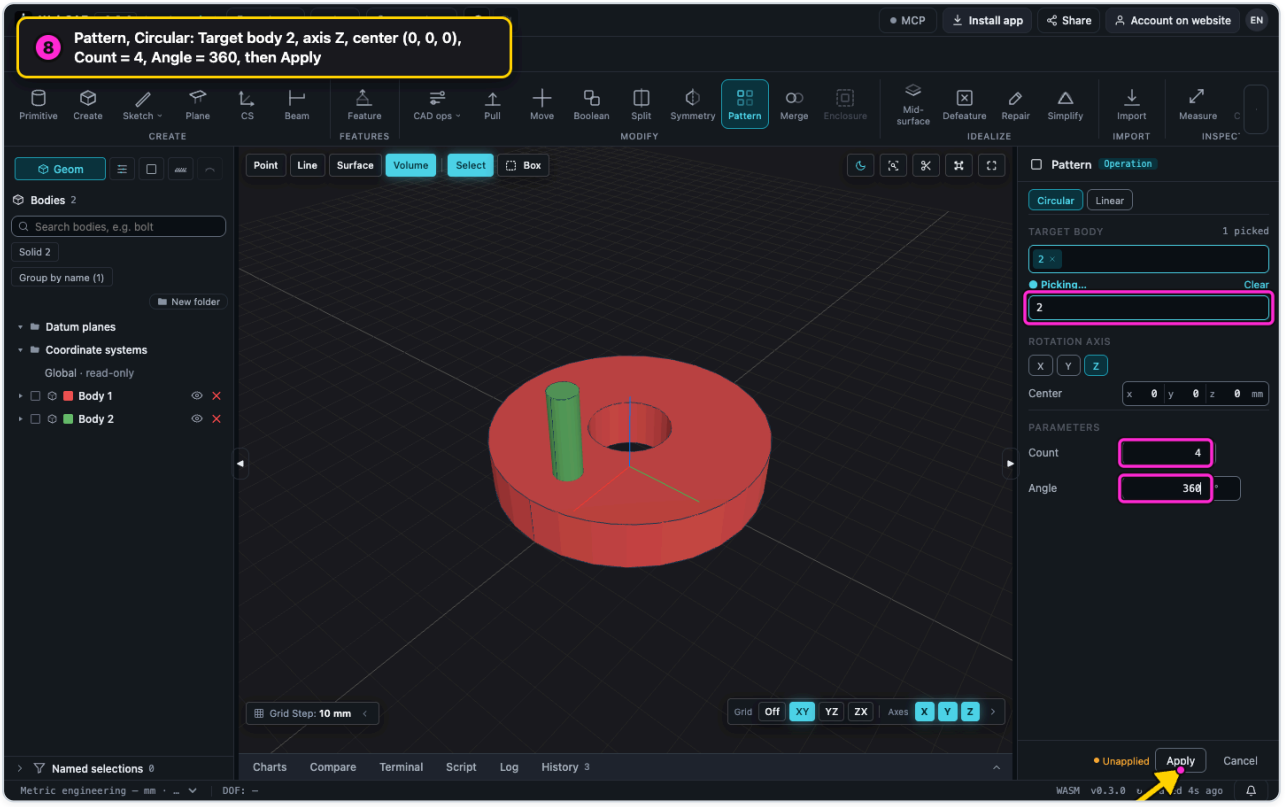
STEP 7 Lift the pin onto the hub face

In the same panel pick body `2` again, click **Z** and type `15` in **Value**, click **Apply**, then **Cancel** to close the panel. The pin now stands on the top face of the hub: it occupies Z from 15 to 40 mm and touches the hub face without entering the hub material. (The circle of the pin, radius 4 mm at 26 mm, lies on the ring between the bore, radius 12 mm, and the rim, radius 40 mm.)



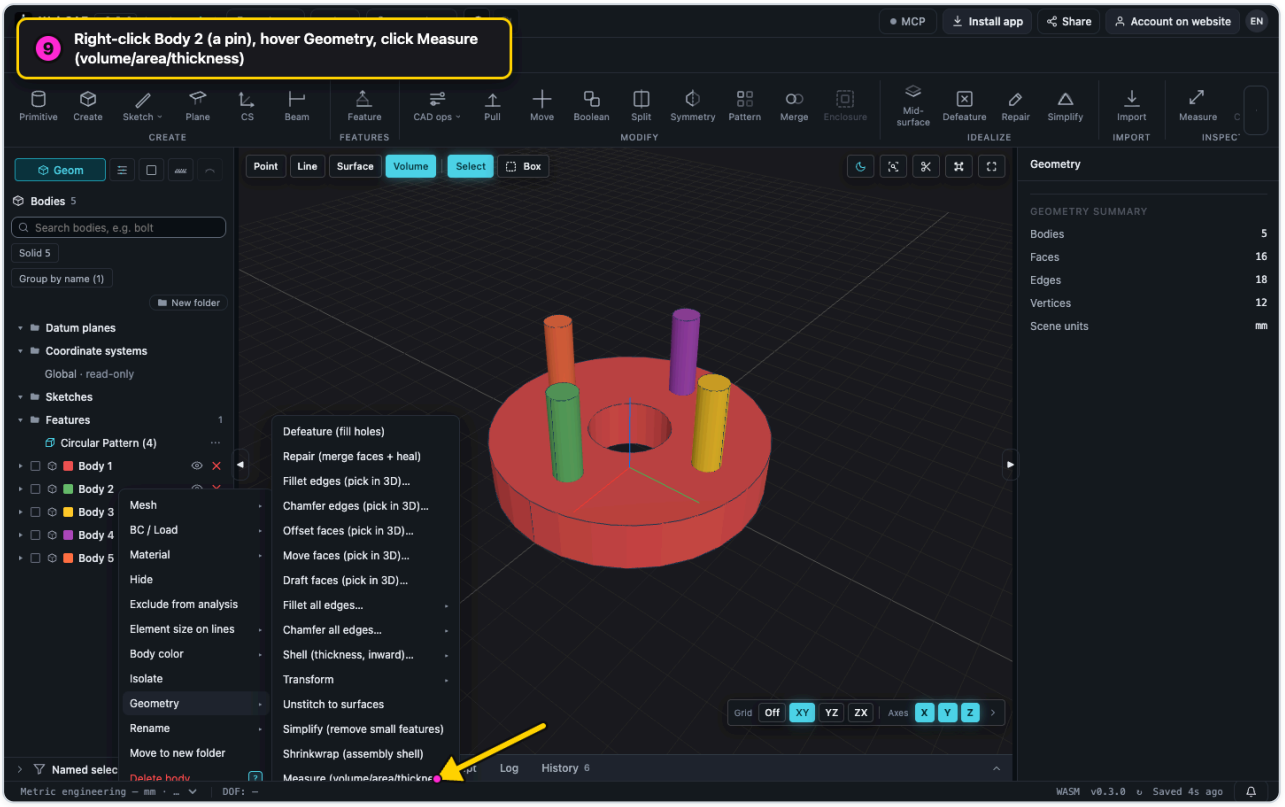
STEP 8 Replicate the pin around the hub axis

Click **Pattern** in the ribbon and choose **Circular**. Type `2` in **Target body** and press Enter, **Rotation axis**: **Z**, **Center** (0, 0, 0), **Count** `4`, **Angle** `360`. Click **Apply**, then **Cancel**. Four pins appear 90° apart; the tree shows five bodies.



STEP 9 Open the Measure command of a pin

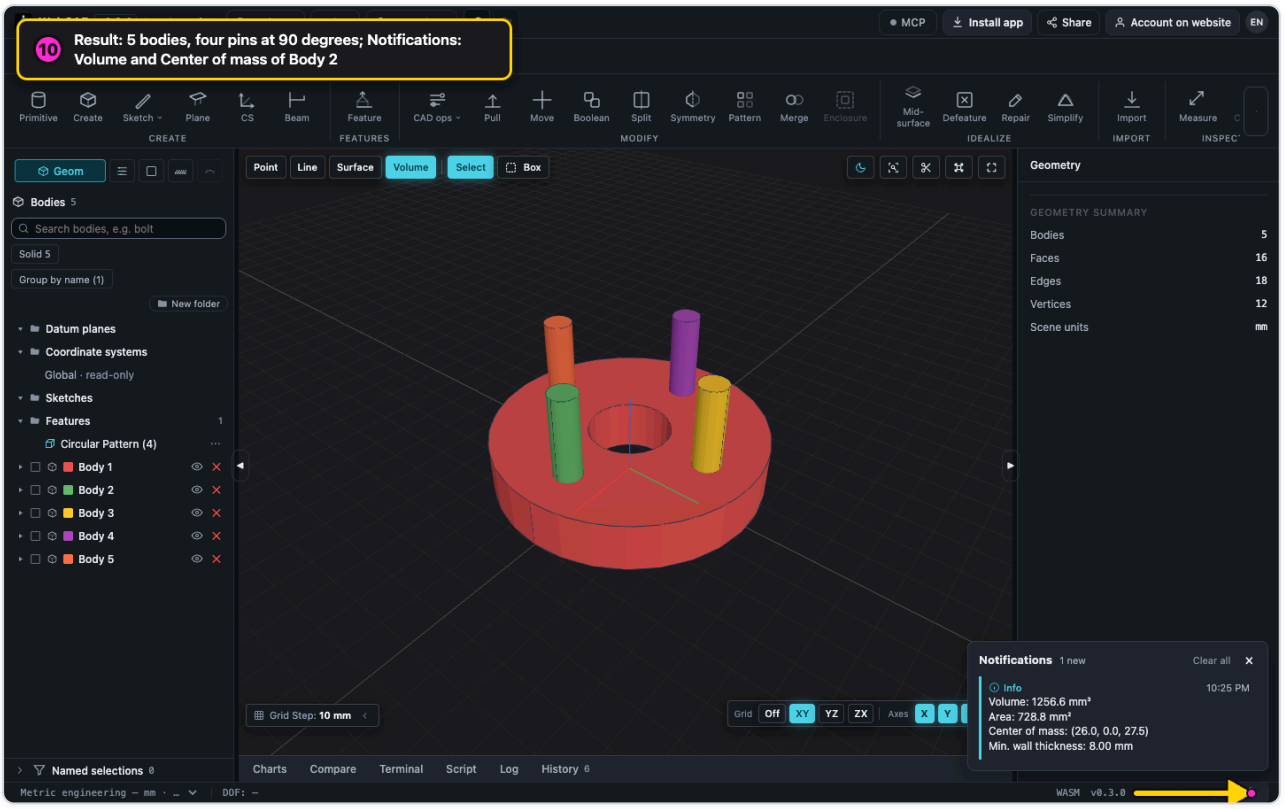
Right-click **Body 2**, hover **Geometry**, click **Measure (volume/area/thickness)**.



STEP 10

Result: five bodies, the pin volume and its position

The viewport shows the hub with four pins, the tree lists **Body 1** to **Body 5**. In **Notifications** (bell) read for **Body 2**: **Volume** 1256.6 mm^3 and **Center of mass** (26.0, 0.0, 27.5): $x = 26$ is the pitch radius, $z = 27.5 = 15 + 25/2$ shows the pin standing on the hub face.



VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Difference
Hub volume (R 40, bore Ø24, H 15)	$\pi(40^2 - 12^2) \times 15 = 68612.4$ mm^3	68612.4 mm^3	— (geometry is OpenCASCADE in WASM, no native entry)	0.00 %
Pin volume (R 4, H 25)	$\pi \times 4^2 \times 25 = 1256.6$ mm^3	1256.6 mm^3	—	-0.00 %
Pitch radius of the pins	26 mm	26.0, 26.0, 26.0, 26.0 mm	—	0.0 %
Angles of the pins	0, 90, 180, 270°	0, 90, 180, 270°	—	—
Pin height position	Z from 15 to 40 mm	15.0 to 40.0 mm (all four)	—	—
Bodies	1 hub + 4 pins = 5	5	—	—

FORMULA

$V_{hub} = \pi(R^2 - r^2)H = \pi(40^2 - 12^2) \times 15 = 68612.4\ \mathrm{mm}^3$, $V_{pin} = \pi \times 4^2 \times 25 = 1256.6\ \mathrm{mm}^3$. Measure reads the B-Rep volume of the body (the number in Notifications); the display mesh of the same bodies is faceted and gives 68089.7 and 1244.4 mm^3, about 1 % less. Volumes are listed per body and not summed: the pins touch the hub face but are not joined to it.

TROUBLESHOOTING & NOTES

Common mistakes

- **Move left in Surface mode:** in the Move panel click **Body**; in Surface mode the pick takes a face, not the pin, and nothing moves.
- **Skipping the lift (step 7):** a pin moved only along X stays at Z = 0 to 25 mm, inside the hub material.
- **Hole radius instead of diameter: Diameter** is 24 mm for a 12 mm radius.
- **Wrong axis in the pattern:** for a hub standing on the XY plane the rotation axis is **Z**.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#14 — Split Body and Half-Model Analysis (F14)**.

WebCAE 101

LESSON 14 · F14

Split Body and a Half-Model Cantilever

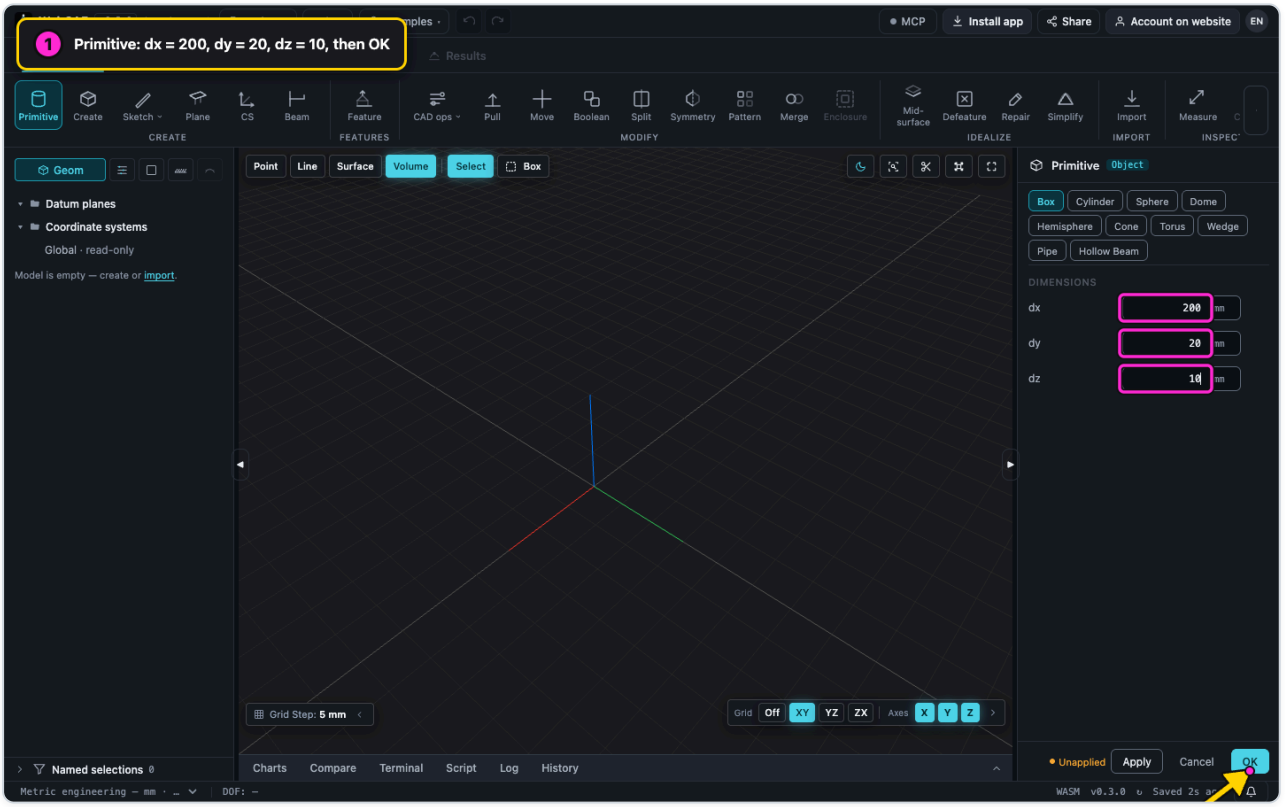
Goal Cut a bar in two with the Split command, keep one half, and check a cantilever solve against the beam formula.

Outcome A $100 \times 20 \times 10$ mm cantilever with a tip deflection $\delta = 0.0993$ mm; the beam formula gives 0.1000 mm (0.67 % apart).

8–10 min | WebCAE web (browser) | Run: 2026-11-04-32ffaed2 / F14

STEP 1 Create the bar

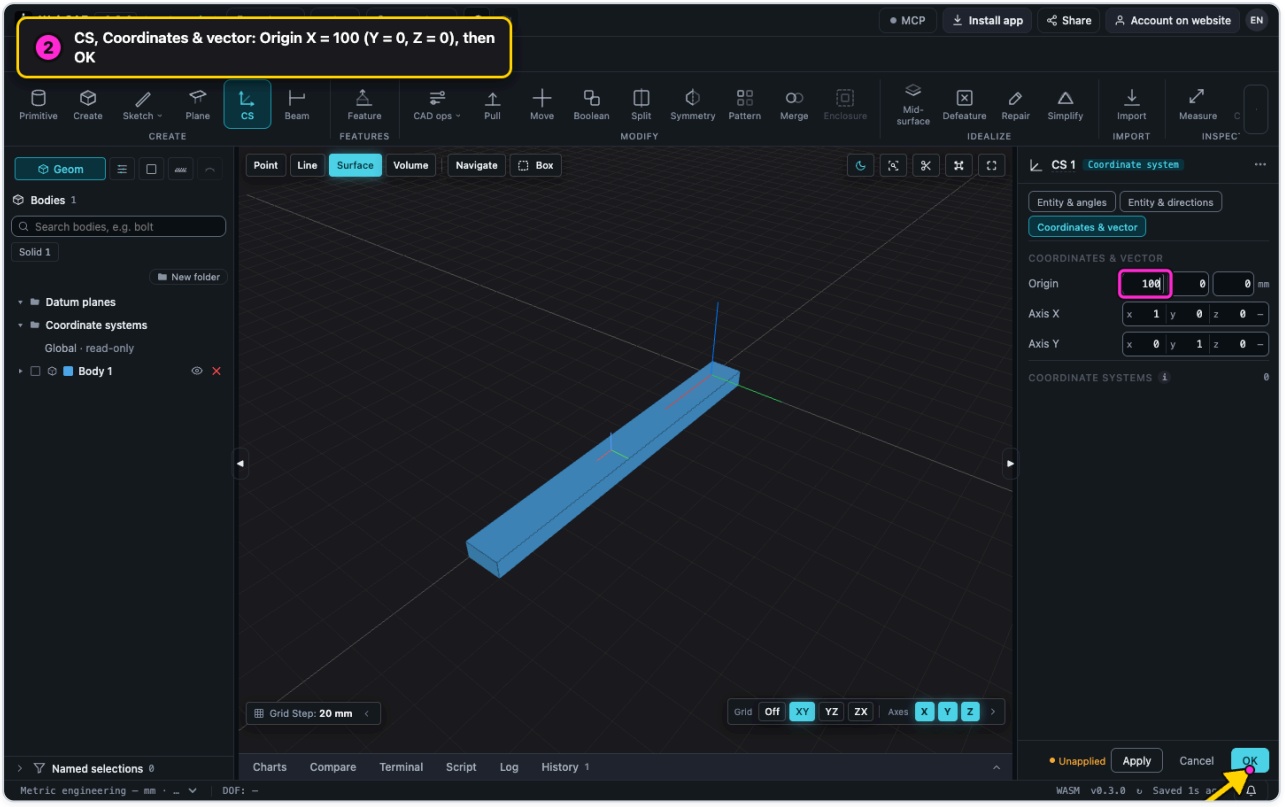
Open **Geometry**, click **Primitive** (shape **Box**), type `200` in **dx**, `20` in **dy**, `10` in **dz**, click **OK**. The bar lies along X from 0 to 200 mm.



STEP 2

Put a coordinate system at X = 100

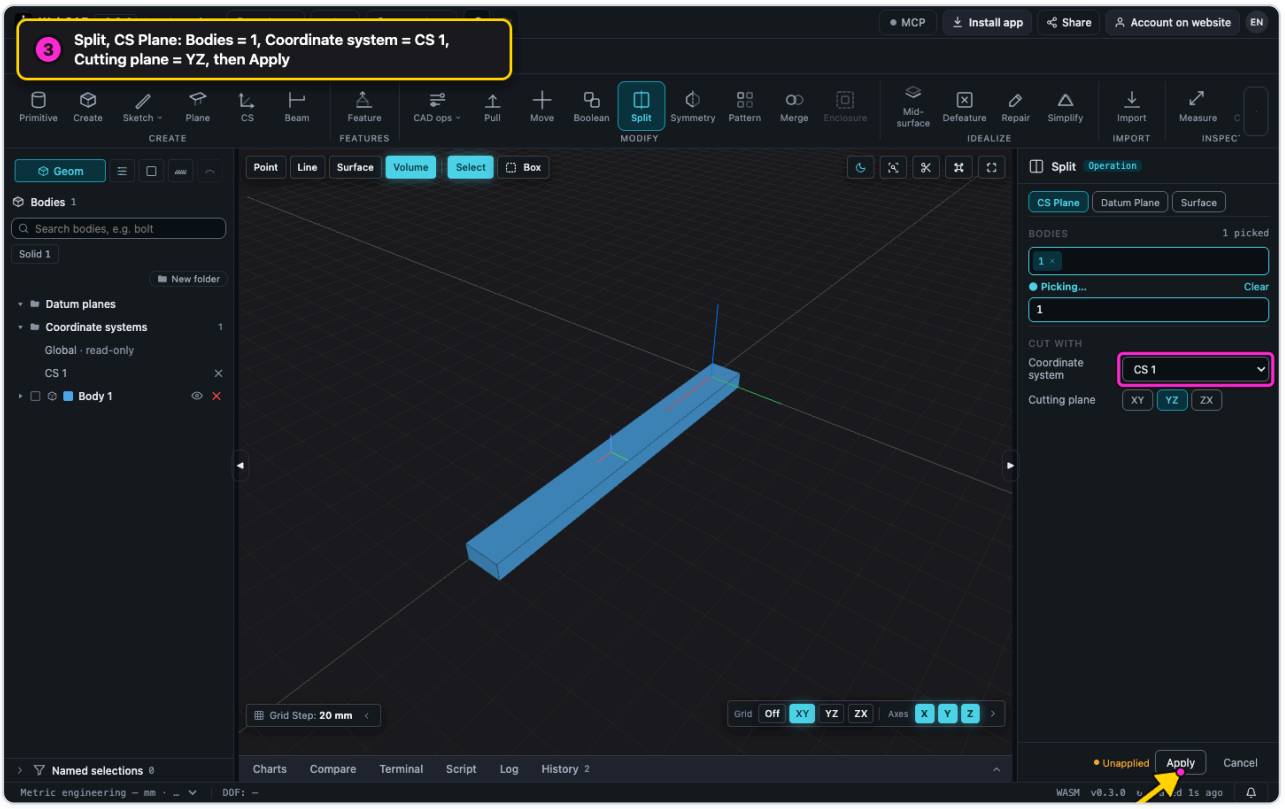
Split cuts along a plane of a coordinate system, so first make one at the cut. Click **CS** in the ribbon, then **Coordinates & vector** in the panel. Under **Origin** type `100` in **X**, leave **Y** and **Z** at 0 and the axes at their defaults, click **OK**. The coordinate system **CS 1** appears in the tree under **Coordinate systems**.



STEP 3

Split the bar

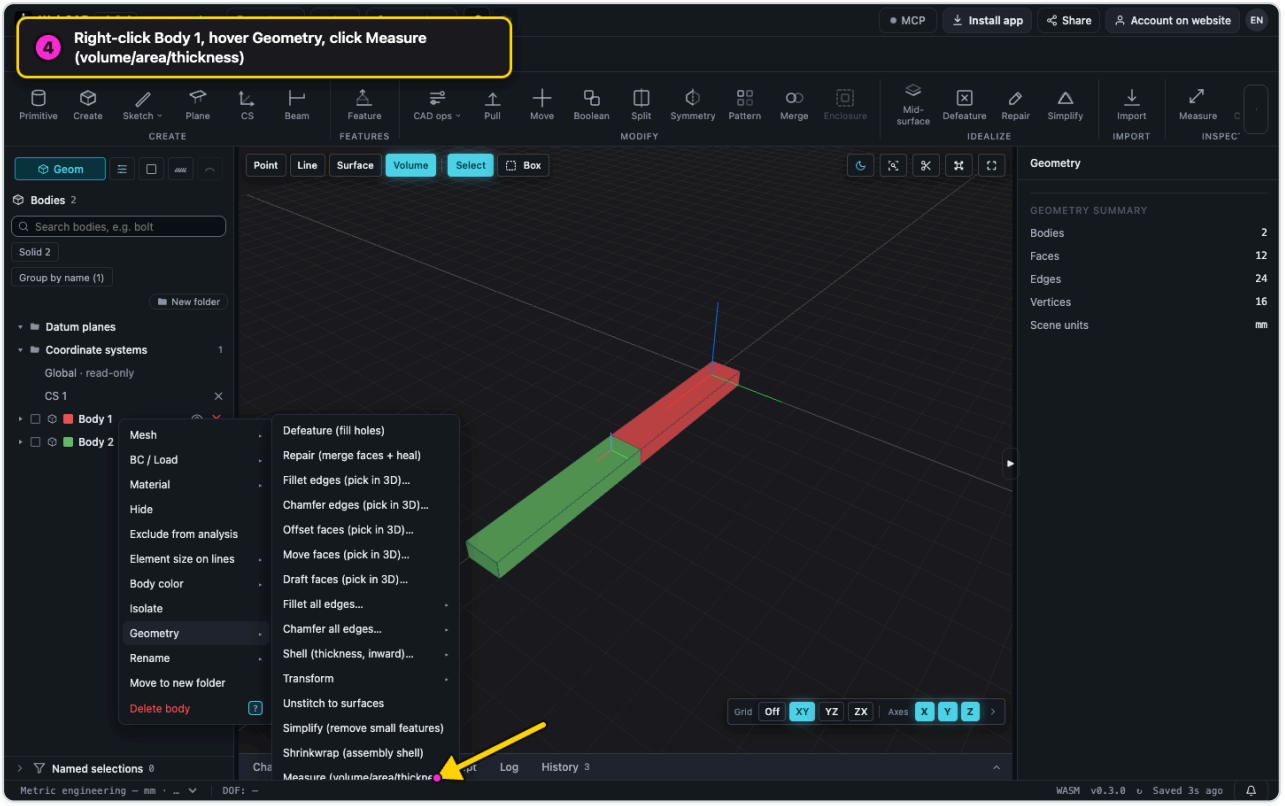
Click **Split** in the ribbon and keep **CS Plane** (the chip at the top of the panel). **Bodies** already holds the bar (1 picked). Under **Cut with** choose **CS 1** in **Coordinate system** and click **YZ** in **Cutting plane** (the plane whose normal is X). Click **Apply**. The bar splits at $X = 100$ mm into two bodies; the tree shows **Body 1** and **Body 2**.



STEP 4

Open the Measure command of Body 1

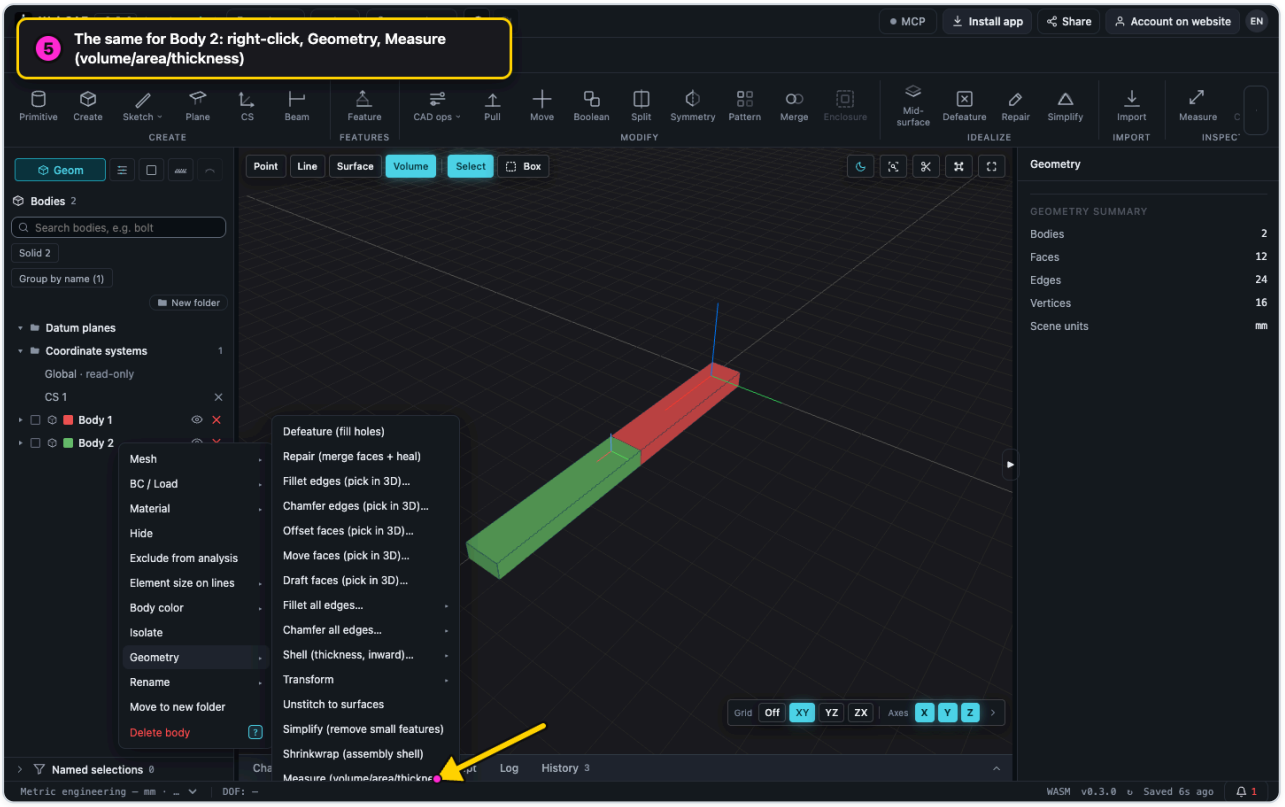
Right-click **Body 1** in the tree, hover **Geometry** and click **Measure (volume/area/thickness)** at the bottom of the submenu.



STEP 5

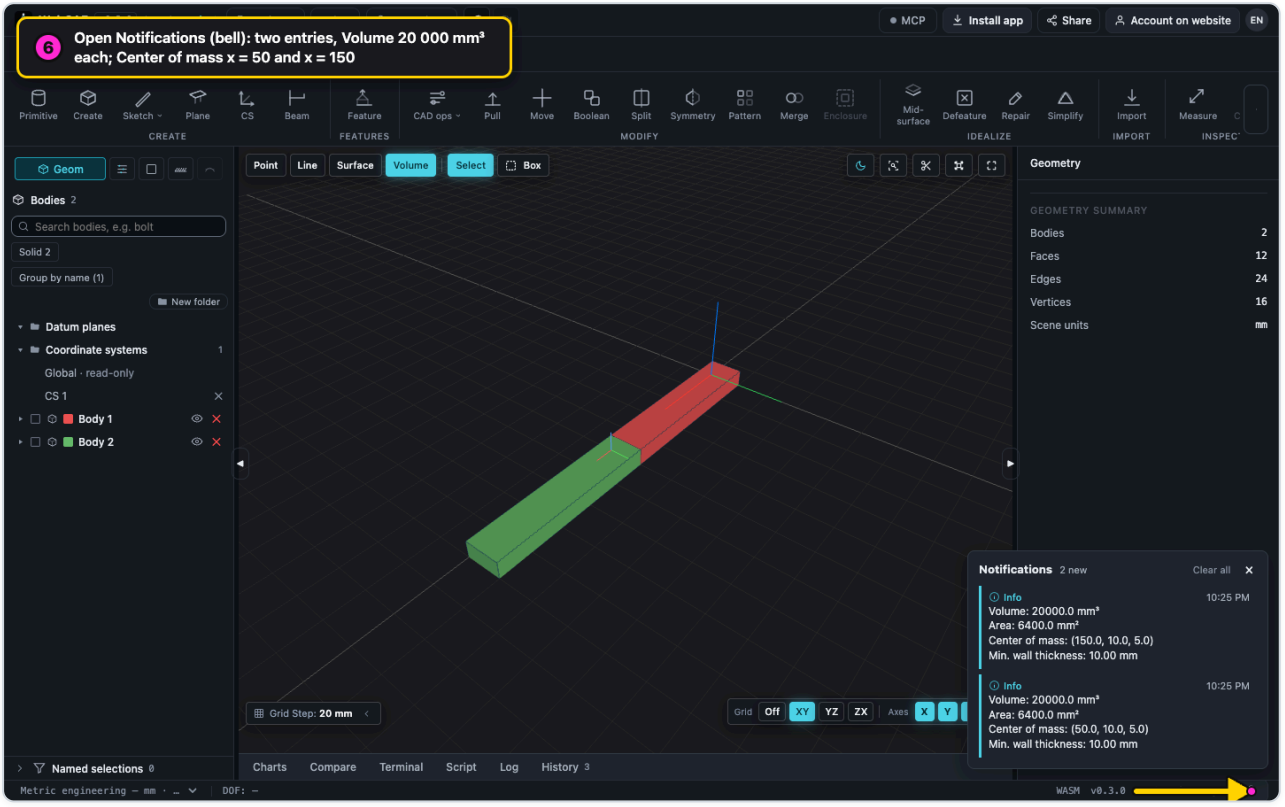
Measure Body 2 the same way

Right-click **Body 2**, hover **Geometry**, click **Measure (volume/area/thickness)**.



STEP 6 Read both halves

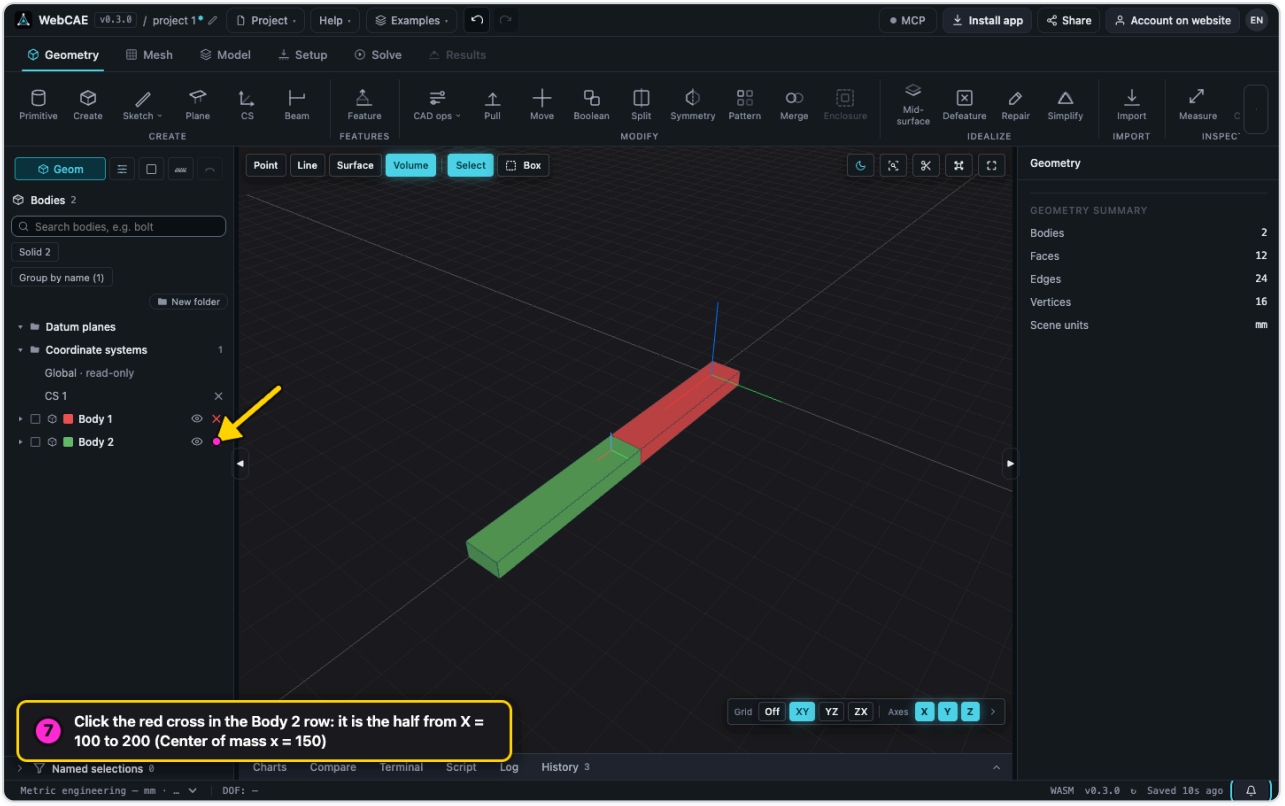
Click the bell at the bottom right to open **Notifications**. Both entries show **Volume** 20000 mm^3 (half of 40 000). **Center of mass** tells the halves apart: x = 150 for the entry measured last (Body 2) and x = 50 for the first (Body 1) — Body 2 is the half from X = 100 to 200, Body 1 the half from X = 0 to 100.



STEP 7

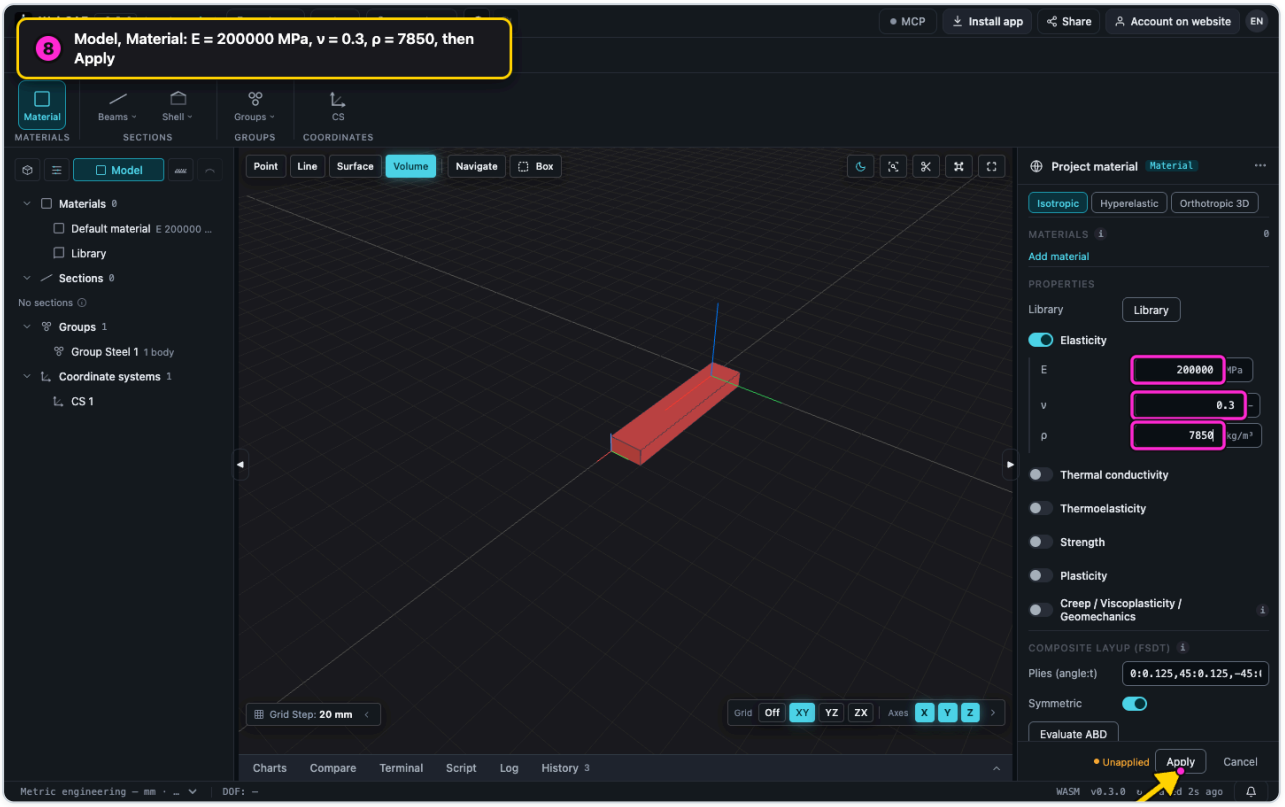
Delete the far half

Click the red cross at the right of the **Body 2** row (the half with Center of mass x = 150). One body remains: the half from X = 0 to 100.



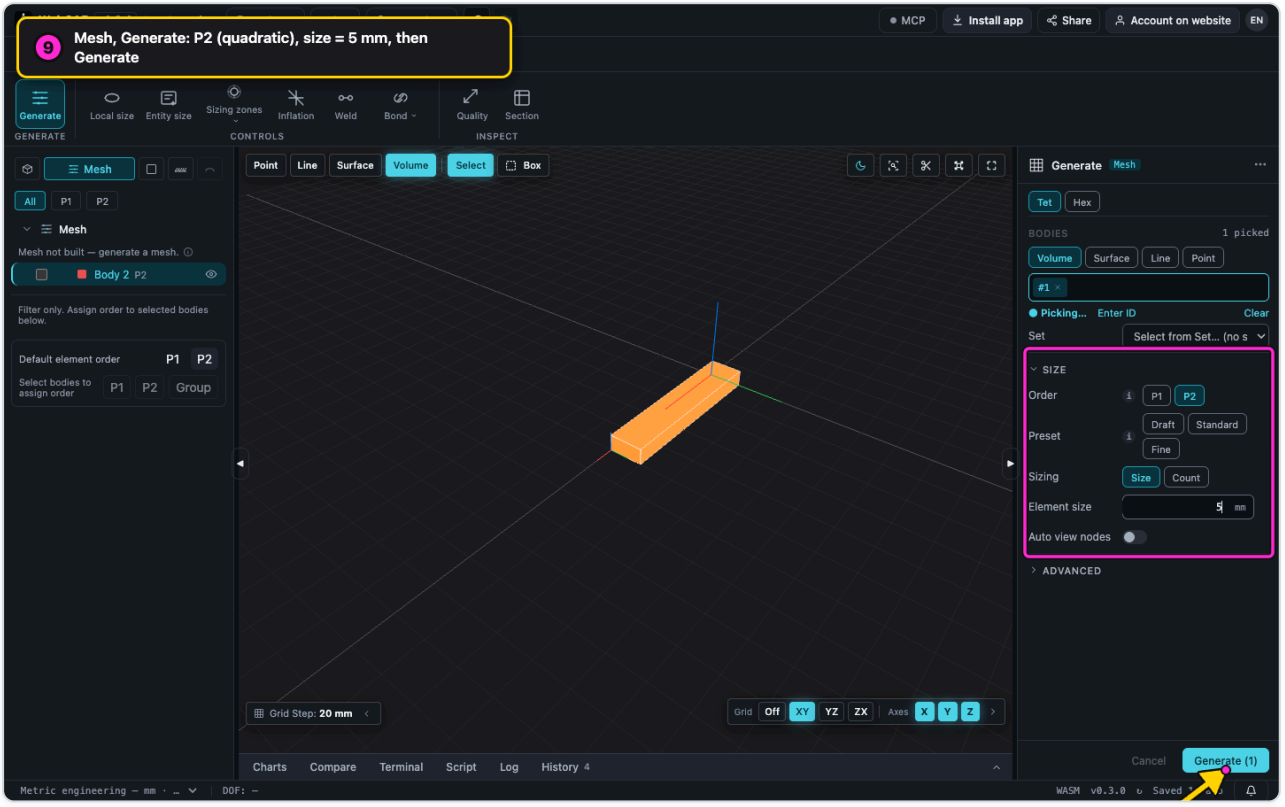
STEP 8 Assign the material

Open the **Model** tab and click **Material**. Type `200000` in **E**, `0.3` in **ν** and `7850` in **ρ**, click **Apply**.



STEP 9 Generate a quadratic mesh

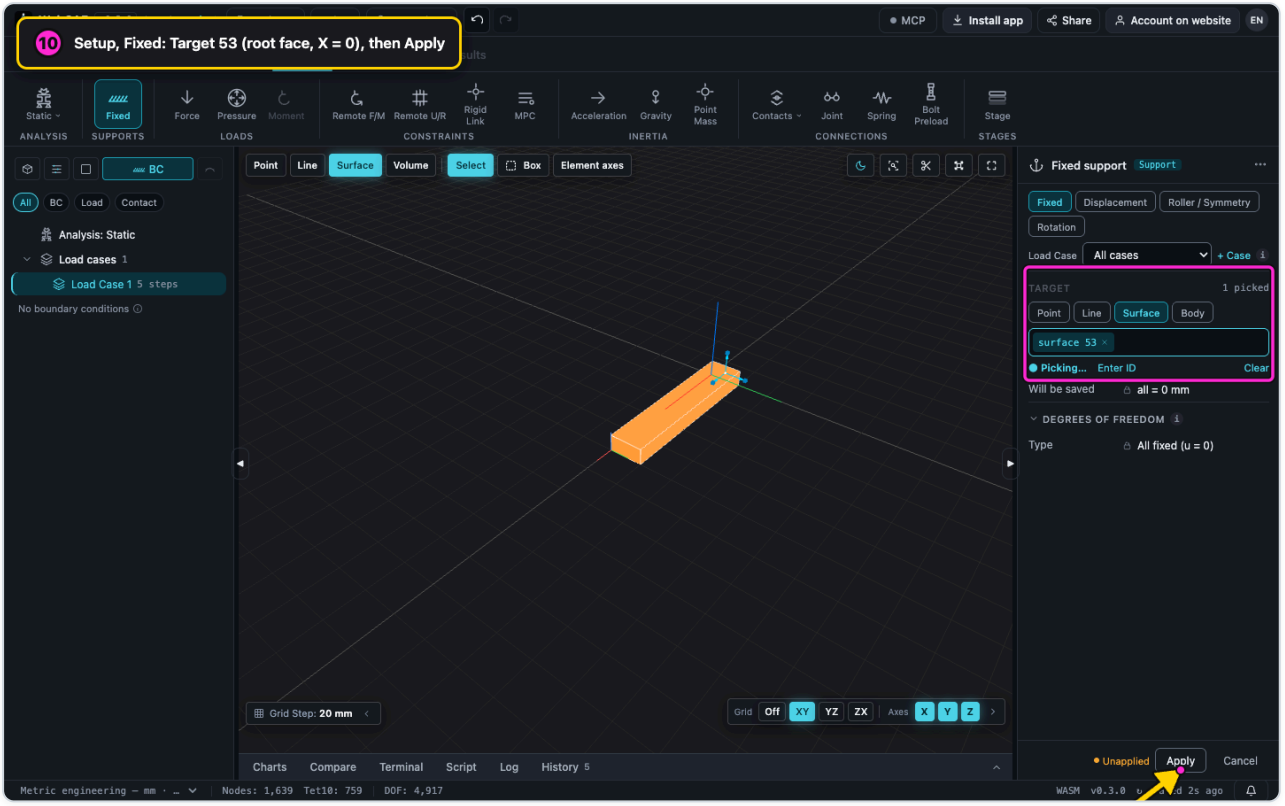
Open the **Mesh** tab, click **Generate**, choose **P2** (quadratic tetrahedra), type `5` mm in the size field and click **Generate**. WebCAE builds 759 Tet10 elements; the status bar at the bottom of the window shows **Tet10: 759** from now on.



STEP 10

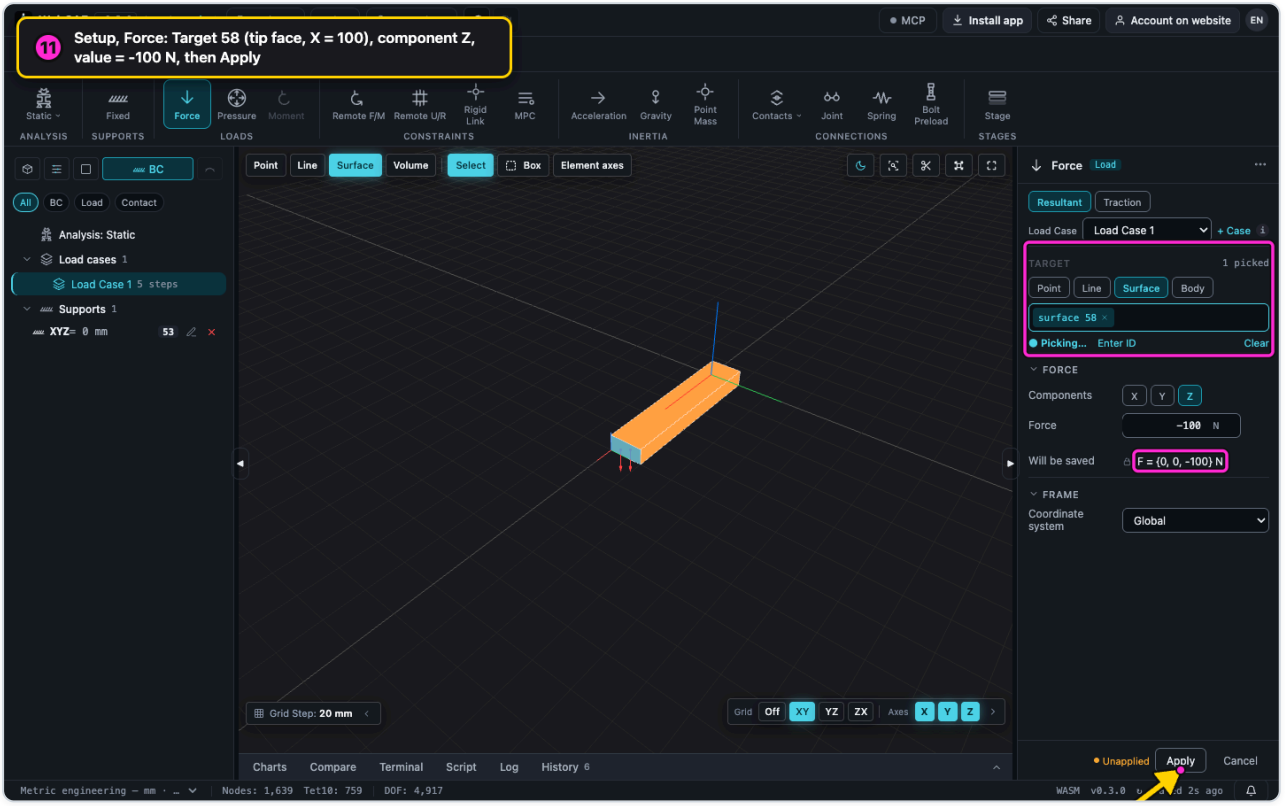
Clamp the root

Open the **Setup** tab, click **Fixed**, click **Enter ID**, type `53` (the end face at X = 0), press Enter and click **Apply**.



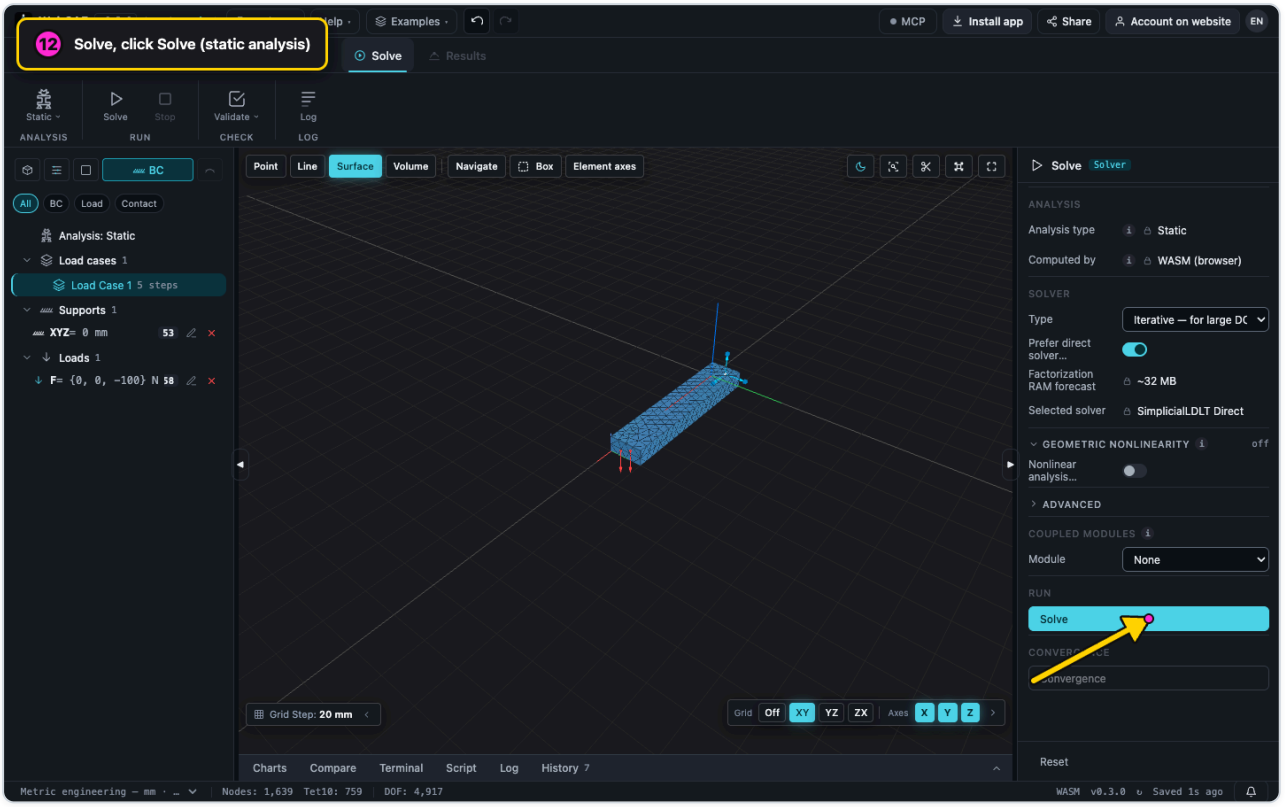
STEP 11 Load the tip

Click **Force**, **Enter ID**, type `58` (the cut face at X = 100), press Enter, click the **Z** component, type `-100` (N) and click **Apply**.



STEP 12 Solve

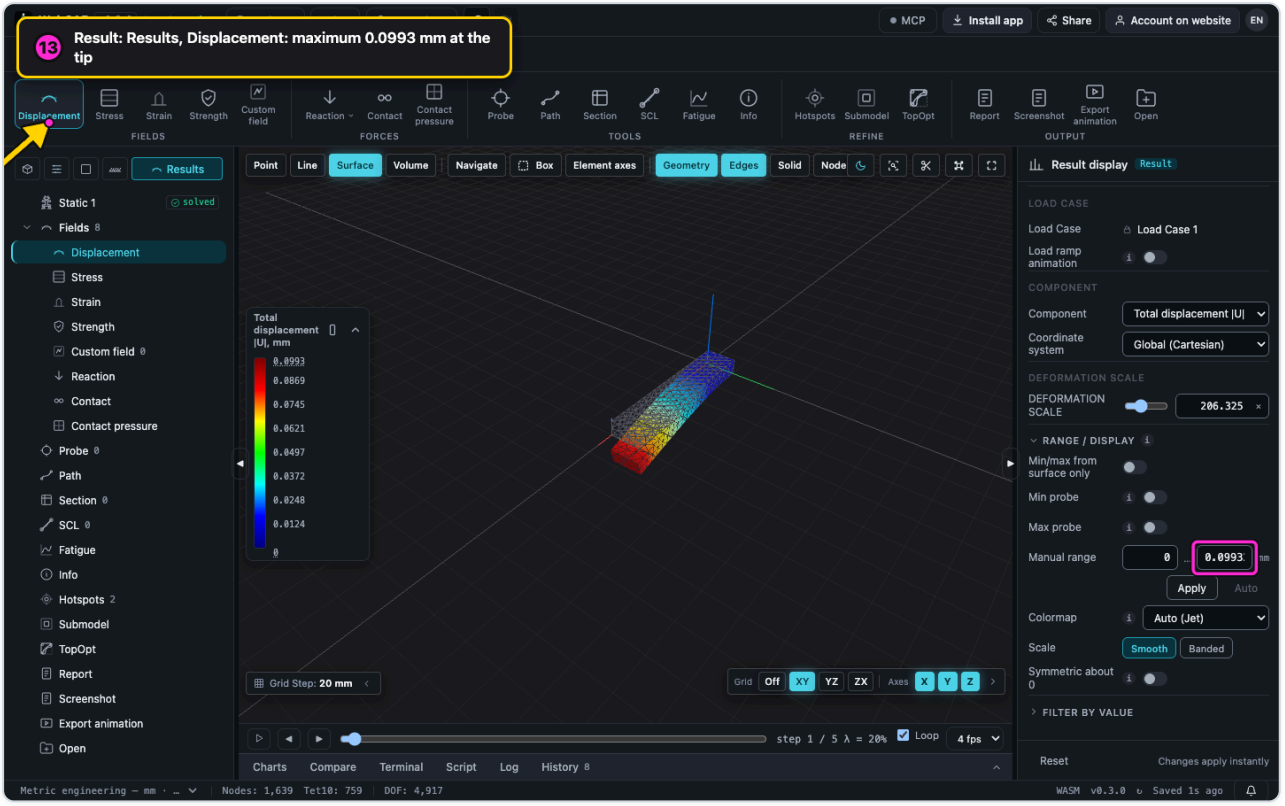
Open the **Solve** tab and click **Solve**.



STEP 13

Result: tip deflection

Open the **Results** tab and click **Displacement**. The upper end of the range (**manual-range-max** field) reads 0.0993 mm: the largest displacement, at the free end.



VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Difference (web)
Volume of each half	$100 \times 20 \times 10 = 20000$ mm^3	20000 and 20000 mm^3	— (geometry is OpenCASCADE in WASM, no native entry)	0.00 %
Quadratic tetrahedra (Tet10), P2, 5 mm	— (no by-hand value)	759 (status bar, steps 10 to 13)	671 (different mesher: the desktop run uses the native mesher)	—
Tip deflection δ	$\frac{PL^3}{3EI} = 0.1000$ mm	0.0993 mm	0.0993 mm (same problem set as a script and solved by the native solver, see docs/2026-11-05-tutorials-101-wave1-native-runs-findings.md)	-0.67 %

FORMULA

$I = \frac{bh^3}{12} = \frac{20 \times 10^3}{12} = 1666.67\ \mathrm{mm}^4$, $\delta = \frac{PL^3}{3EI} = \frac{100 \times 100^3}{3 \times 200000 \times 1666.67} = 0.1000$ mm. The slenderness is $L/h = 10$, so shear deformation adds $\frac{PL}{\kappa GA} = 0.0008$ mm (Timoshenko: 0.1008 mm). WebCAE is 0.7 % below the Euler–Bernoulli figure; the clamped root also blocks the Poisson contraction of the wide section, so a 3D model is slightly stiffer than either beam theory.

TROUBLESHOOTING & NOTES

Common mistakes

- **Cutting without a coordinate system at X = 100:** with **Global** in **Coordinate system** the plane YZ passes through X = 0 and does not cut the bar.
- **Deleting the wrong half:** the clamped end must be the half with Center of mass x = 50; check it in step 6.
- **Splitting is not symmetry:** cutting a body in two only divides geometry; a symmetry boundary condition on the cut face would be a separate step.
- **Force axis:** click **Z** before typing –100, otherwise the load acts along another axis.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#15 — CAD Defeaturing and Midsurface Extraction (F15)**.

WebCAE 101

LESSON 15 · F15

Defeaturing and Midsurface

Goal Make a small hole in a thin plate, remove it again with Remove faces (by radius), and extract the mid-surface sheet of the plate.

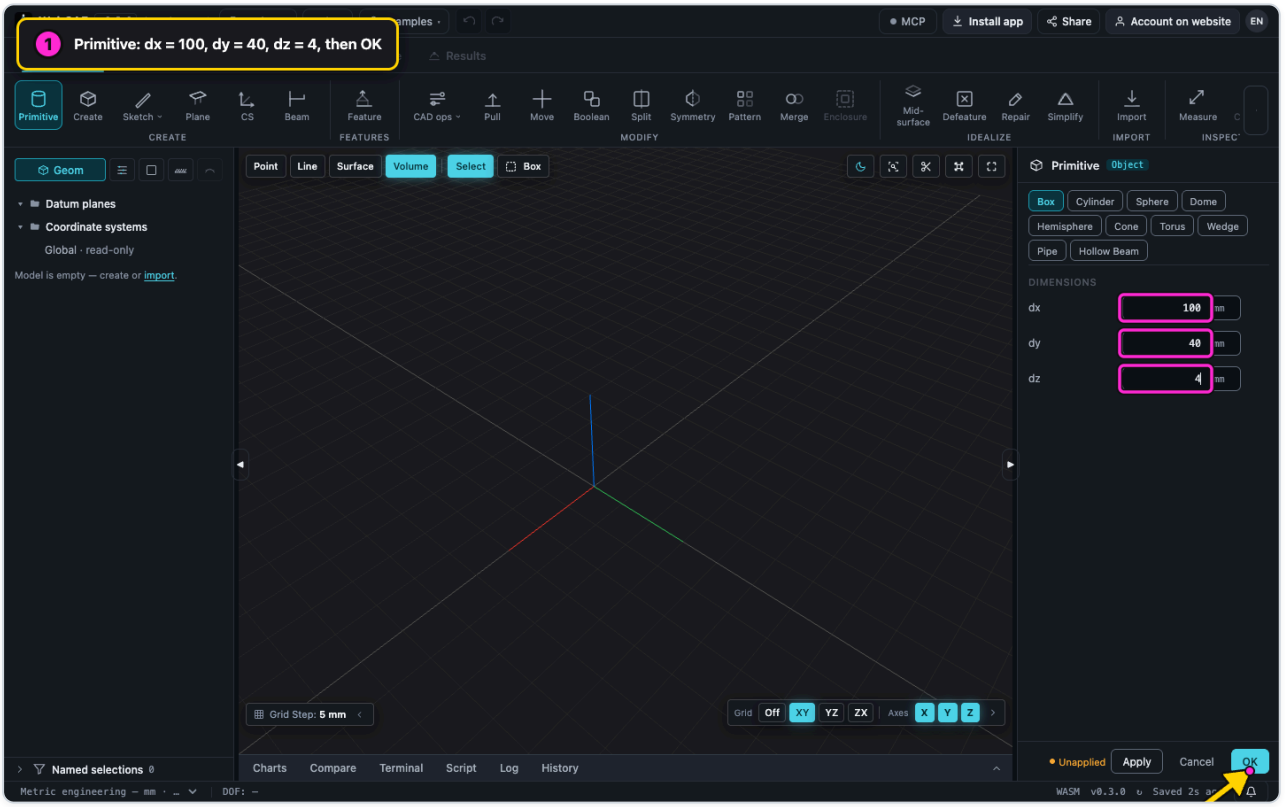
Outcome The plate returns to 16 000 mm^3 with 6 faces; the mid-surface sheet has an area of 4000 mm^2 (WebCAE: 4000.0).

6–7 min | WebCAE web (browser), geometry engine OpenCASCADE in WASM

Run: 2026-11-04-32ffaed2 / F15

STEP 1 Create the plate

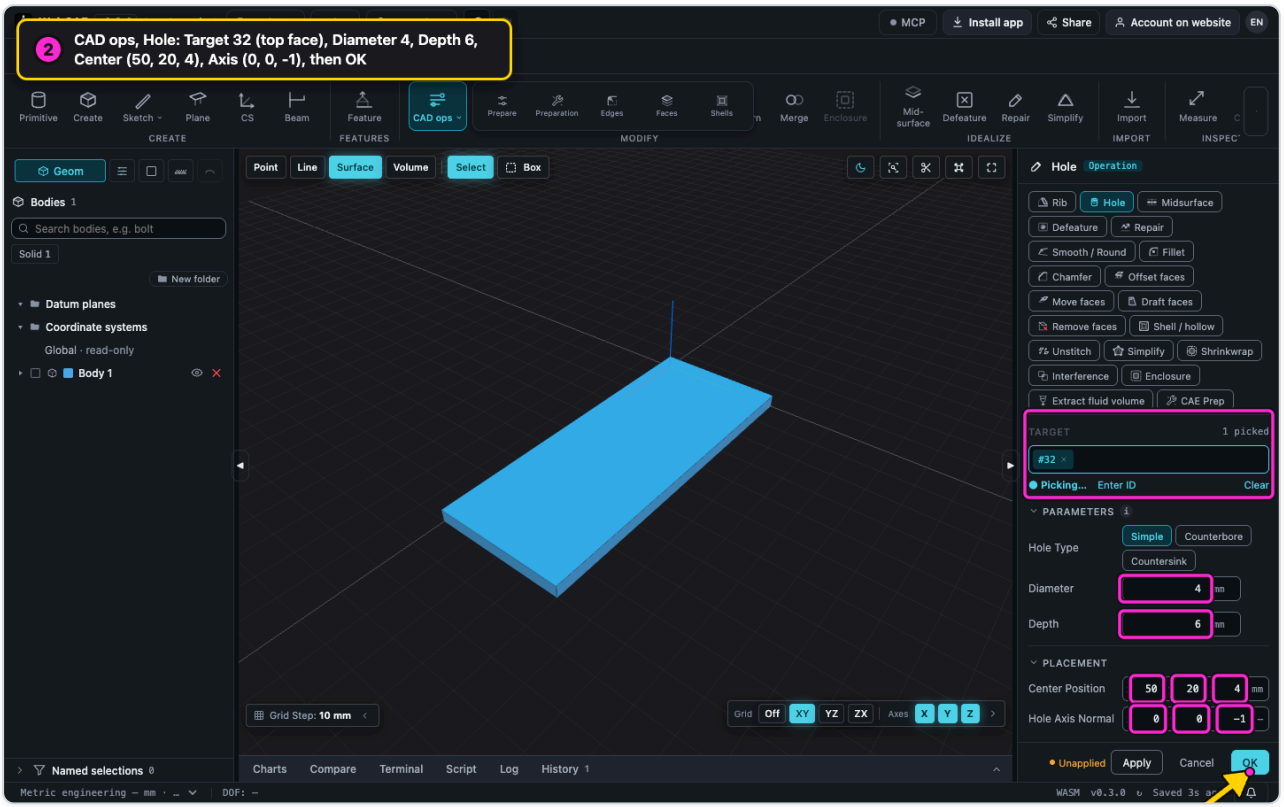
Open **Geometry**, click **Primitive** (shape **Box**), type `100` in **dx**, `40` in **dy**, `4` in **dz**, click **OK**. Body 1 appears.



STEP 2

Drill a Ø4 hole

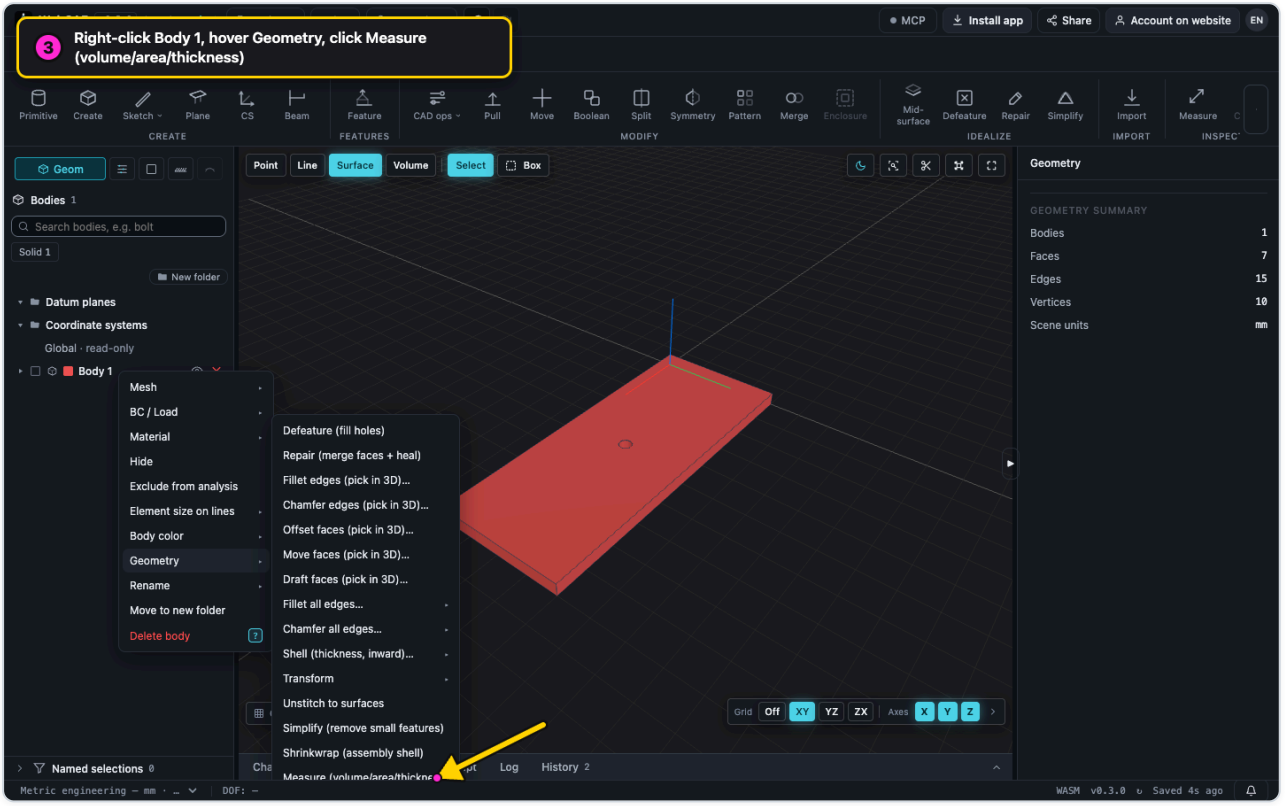
Click **CAD ops** in the ribbon and choose **Hole**. In **Target** click **Enter ID**, type `32` and press Enter: surface 32 is the top face of the plate (Normal +Z, at $Z = 4$ mm, area 4000 mm^2). Keep **Hole Type** = **Simple**. Type `4` in **Diameter** and `6` in **Depth** (deeper than the 4 mm plate: through). **Center Position**: x `50`, y `20`, z `4`. **Hole Axis Normal**: x `0`, y `0`, z `-1`. Click **OK**.



STEP 3

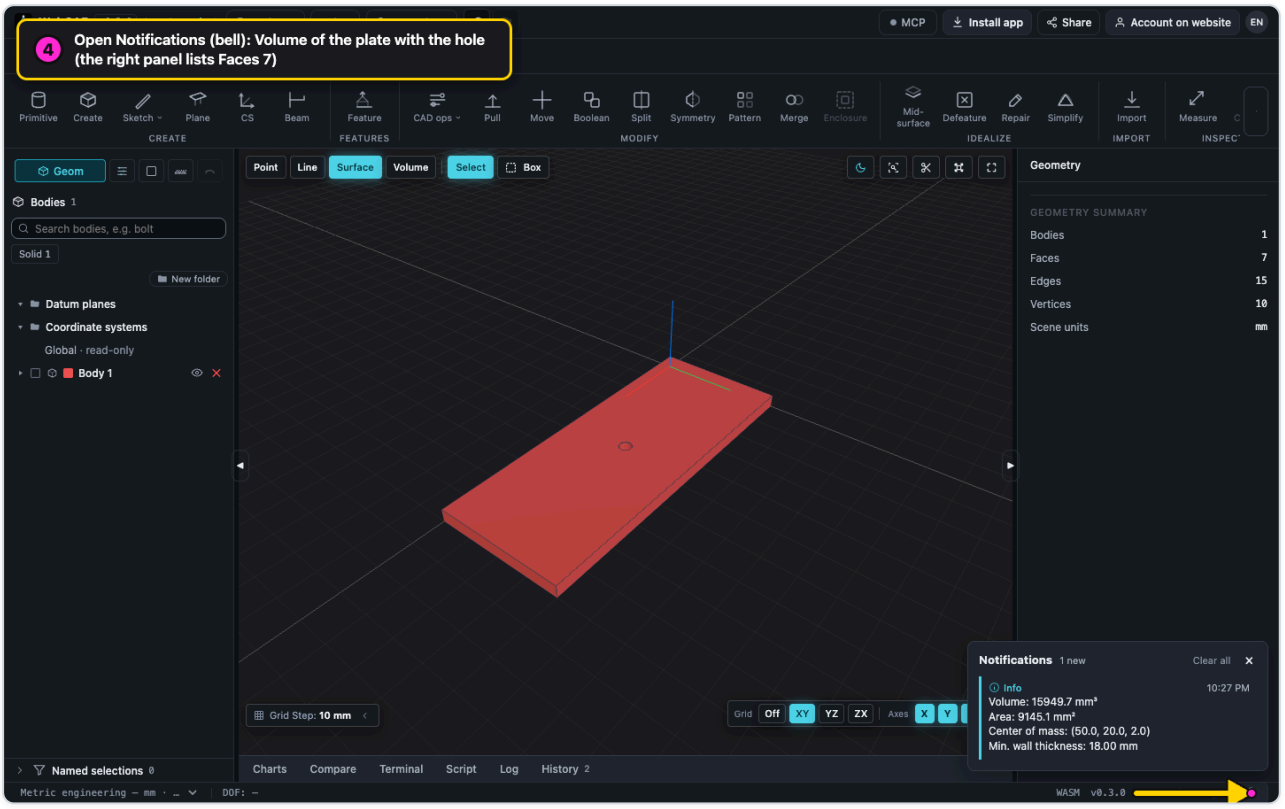
Open the Measure command of Body 1

Right-click **Body 1** in the tree, hover **Geometry** and click **Measure (volume/area/thickness)** at the bottom of the submenu.



STEP 4 Read the volume of the plate with the hole

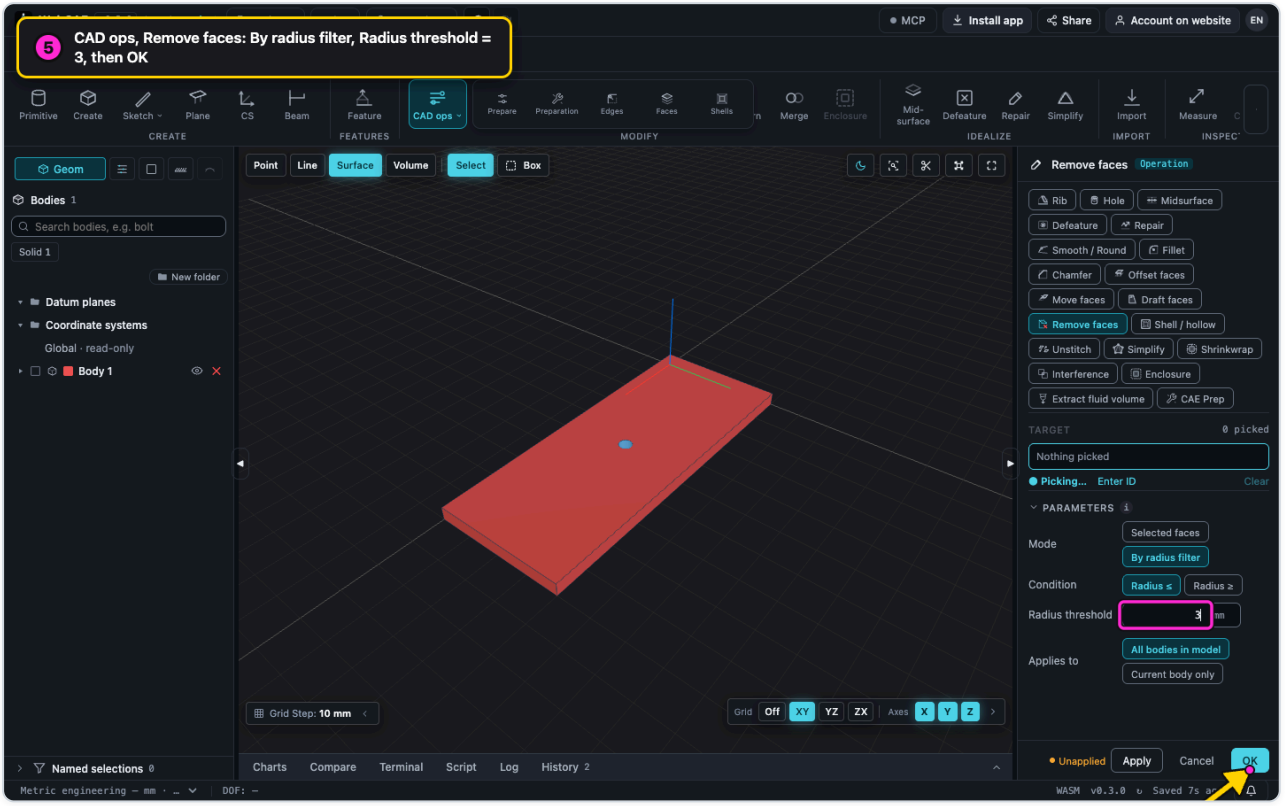
Click the bell at the bottom right to open **Notifications**. **Volume**: 15949.7 mm^3. The summary on the right (nothing selected) lists **Faces** 7: six plate faces and the wall of the hole.



STEP 5

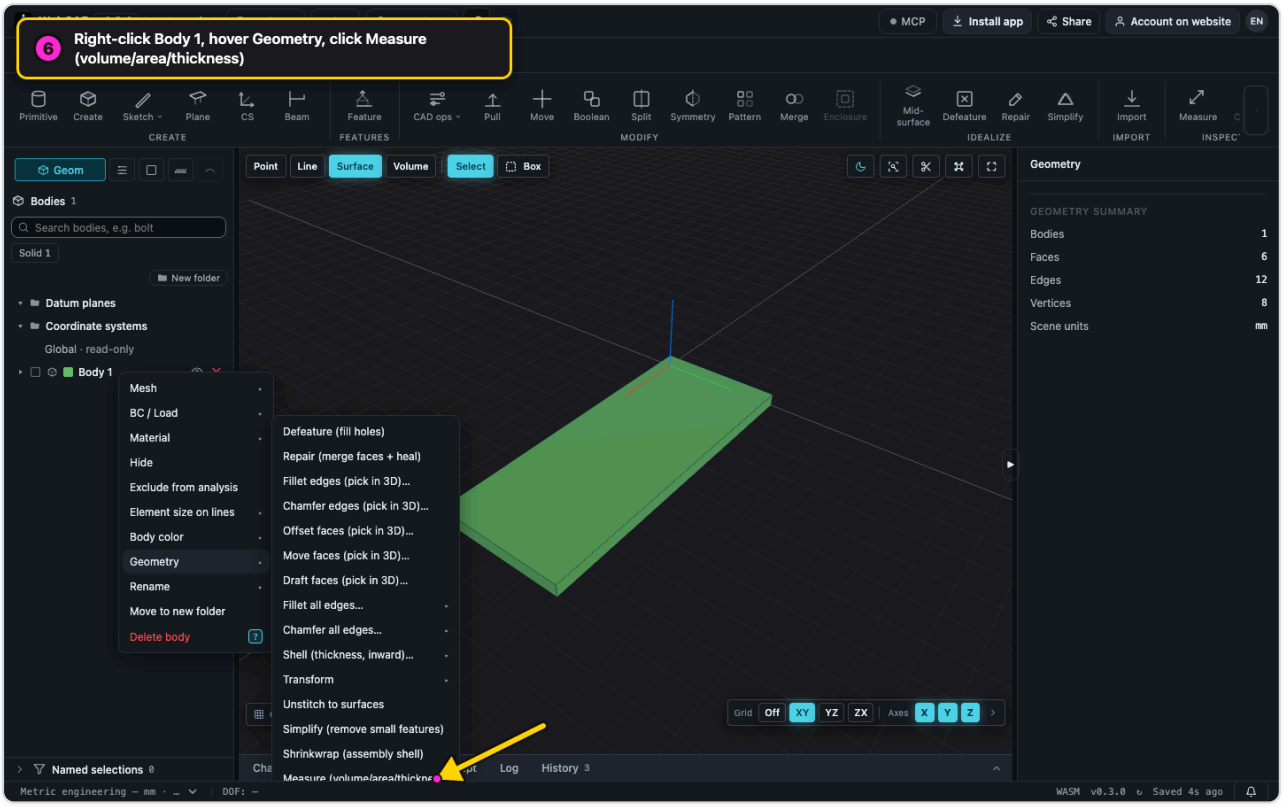
Remove the hole by radius

Click **CAD ops** and choose **Remove faces**. Switch the mode to **By radius filter**, type `3` in the radius threshold field (the hole radius is 2 mm, so it is below the threshold) and click **OK**.



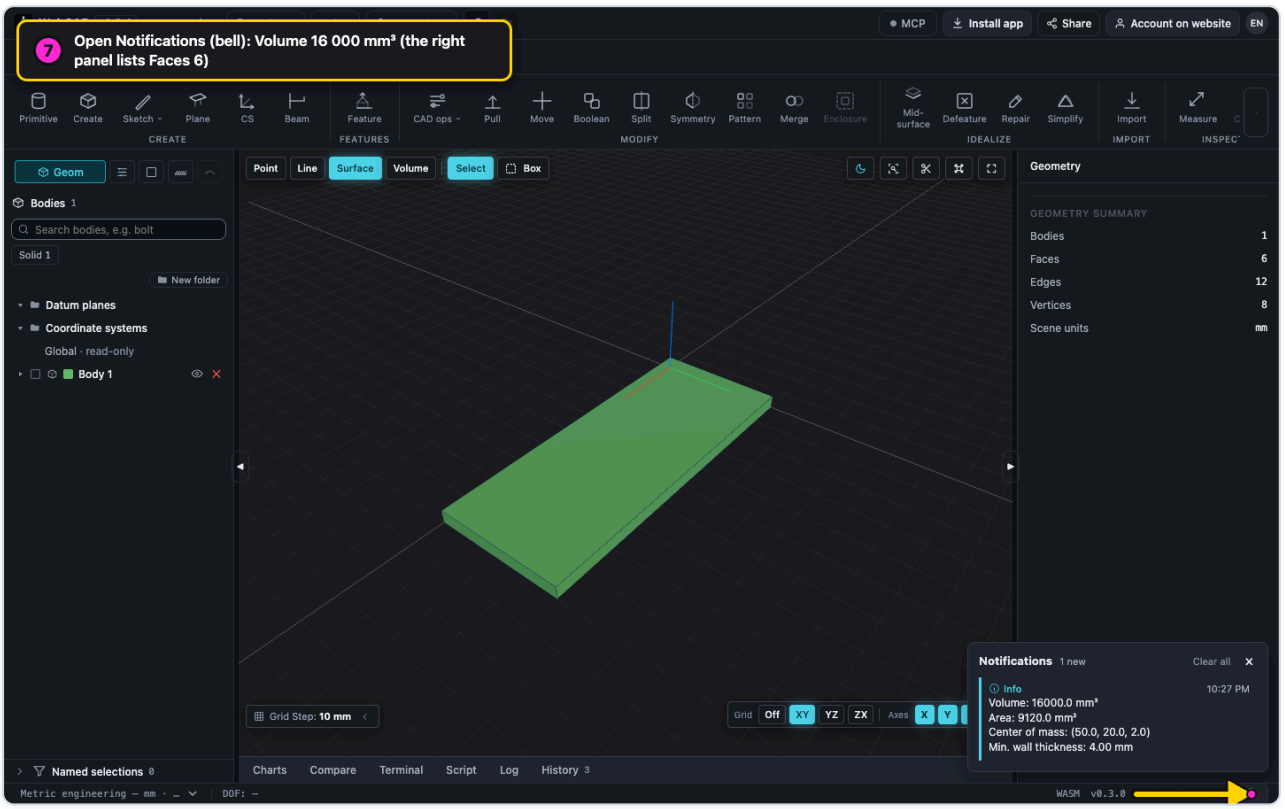
STEP 6 Open the Measure command of Body 1 again

Right-click **Body 1**, hover **Geometry**, click **Measure (volume/area/thickness)**.



STEP 7 Read the volume after the hole is gone

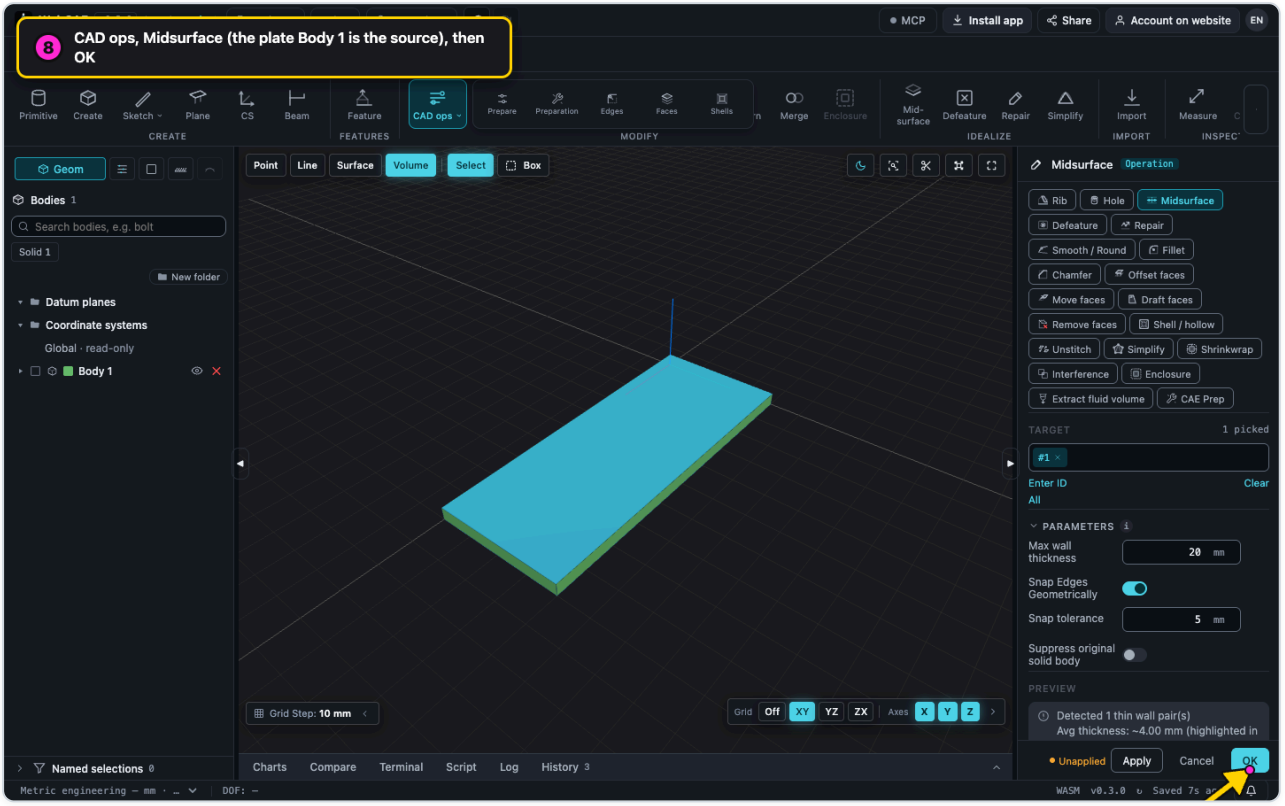
Open **Notifications** (bell): **Volume** 16000.0 mm^3, and the summary lists **Faces** 6.



STEP 8

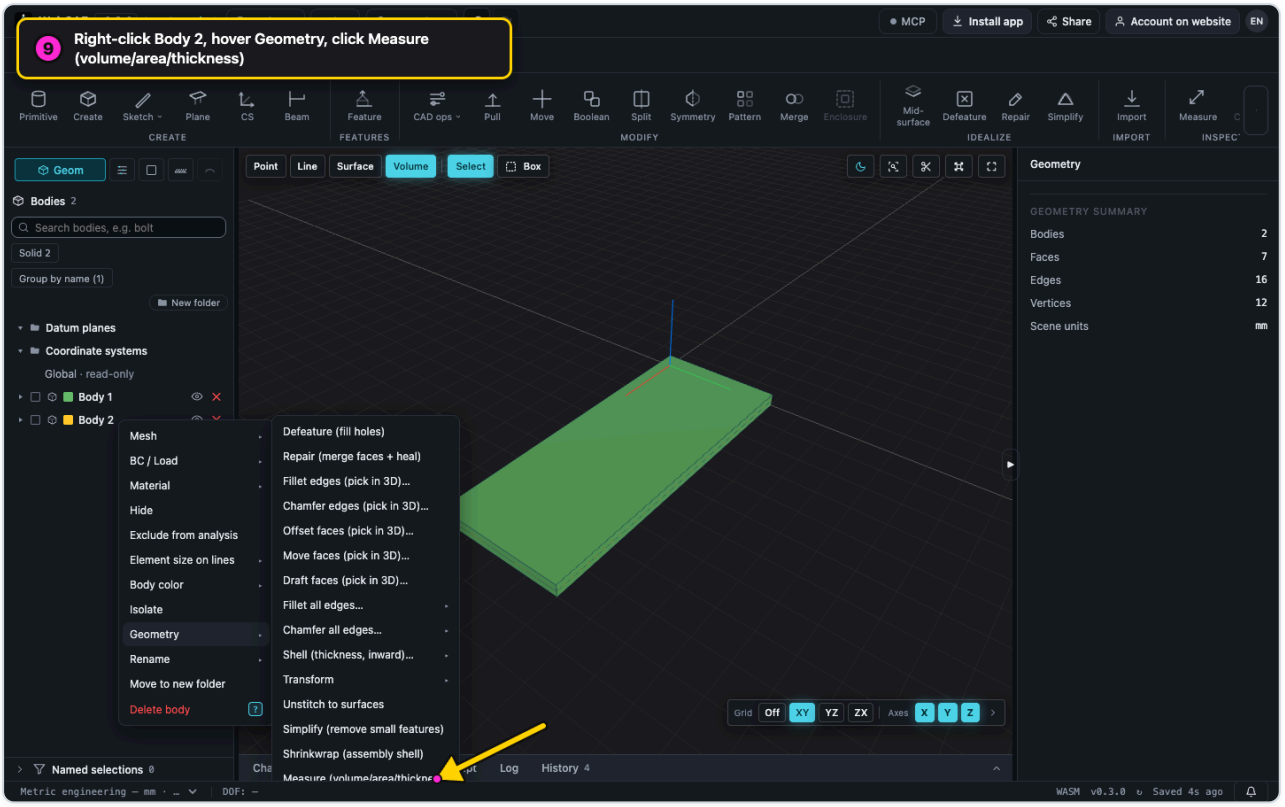
Extract the mid-surface

Click **CAD ops** and choose **Midsurface**. The plate is the source; click **OK**. A sheet body appears in the tree next to the plate (WebCAE keeps the solid; hide or delete it by hand if you only want the shell).



STEP 9 Open the Measure command of the sheet

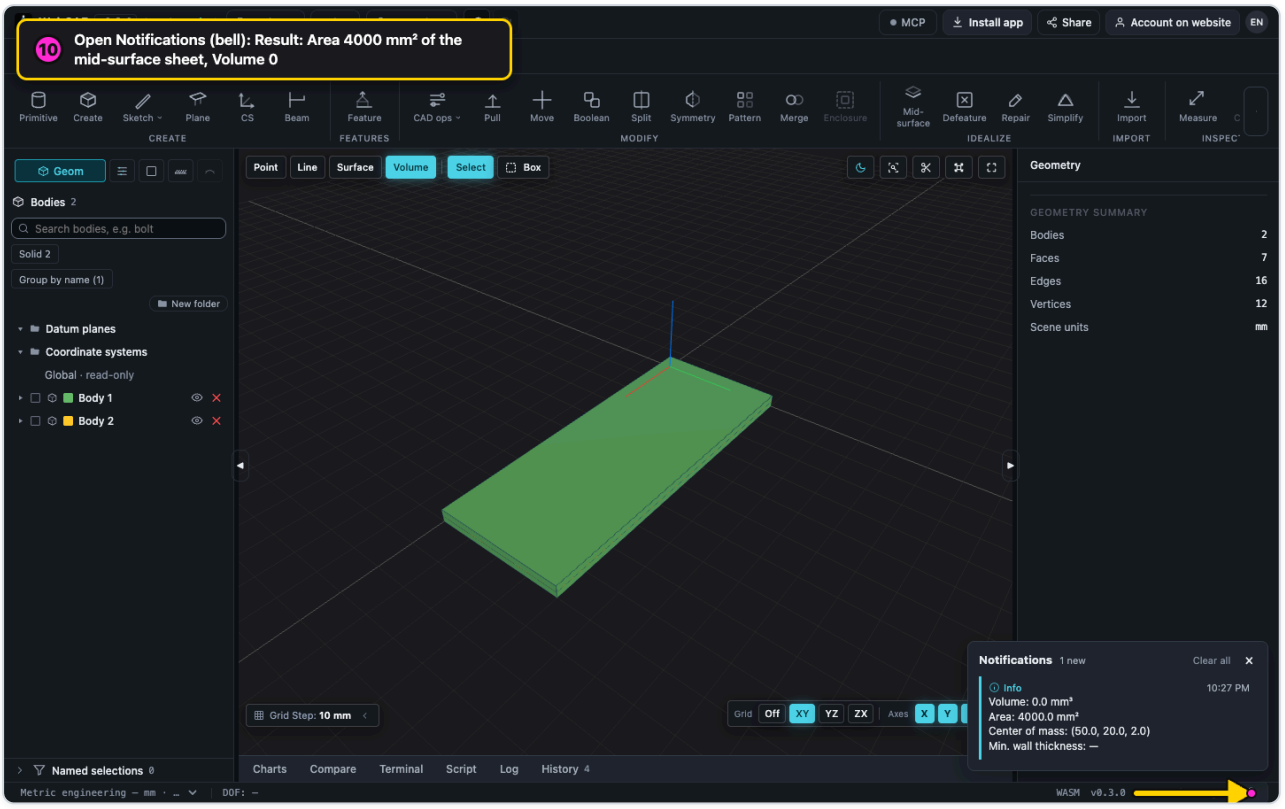
Right-click the new sheet body (**Body 2**), hover **Geometry**, click **Measure (volume/area/thickness)**.



STEP 10

Result: area of the mid-surface

Open **Notifications** (bell). **Area**: 4000.0 mm^2 and **Volume** 0.0: a surface has no volume. The sheet lies halfway through the plate thickness ($Z = 2$ mm).



VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Difference (web)
Volume of the plate with the hole	$100 \times 40 \times 4 - \pi \times 2^2 \times 4 = 15949.7\ \mathrm{mm}^3$	15949.7 mm^3	— (geometry is OpenCASCADE in WASM, no native entry)	-0.00 %
Faces with the hole	7	7	—	—
Volume after Remove faces	$100 \times 40 \times 4 = 16000\ \mathrm{mm}^3$	16000.0 mm^3	—	0.00 %
Faces after Remove faces	6	6	—	—
Area of the mid-surface	$100 \times 40 = 4000\ \mathrm{mm}^2$	4000.0 mm^2	— (the midsurface command has no script form, so no desktop number)	0.00 %

FORMULA

Plate volume $V = LWt = 100 \times 40 \times 4 = 16000\ \mathrm{mm}^3$, hole $\pi r^2 t = \pi \times 2^2 \times 4 = 50.3\ \mathrm{mm}^3$. Sheet area $A = LW = 4000\ \mathrm{mm}^2$. The numbers are read from **Measure (volume/area/thickness)**, steps 4, 7 and 10; the face counts from the geometry summary on the right of the window.

TROUBLESHOOTING & NOTES

Common mistakes

- **Threshold below the hole radius:** the radius threshold must be at least the hole radius (2 mm); 3 mm works, 1 mm leaves the hole.
- **Measuring the wrong body in step 9:** the solid plate is still there after Midsurface; measure the sheet body, otherwise you read a volume of 16 000 mm^3 and an area of 6 000 mm^2 (the full surface of the plate).
- **Hole on the wrong face:** the target must be the top face (Normal +Z); a hole axis pointing up from the top face cuts nothing.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#16 — Assembly Interference Resolution and CFD Enclosure (F16)**.

WebCAE 101

LESSON 16 · F16

Interference Resolve and a CFD Enclosure around the Whole Assembly

Goal Find and remove an overlap between two bodies, join them, and build a fluid-domain box (Enclosure) around the whole assembly.

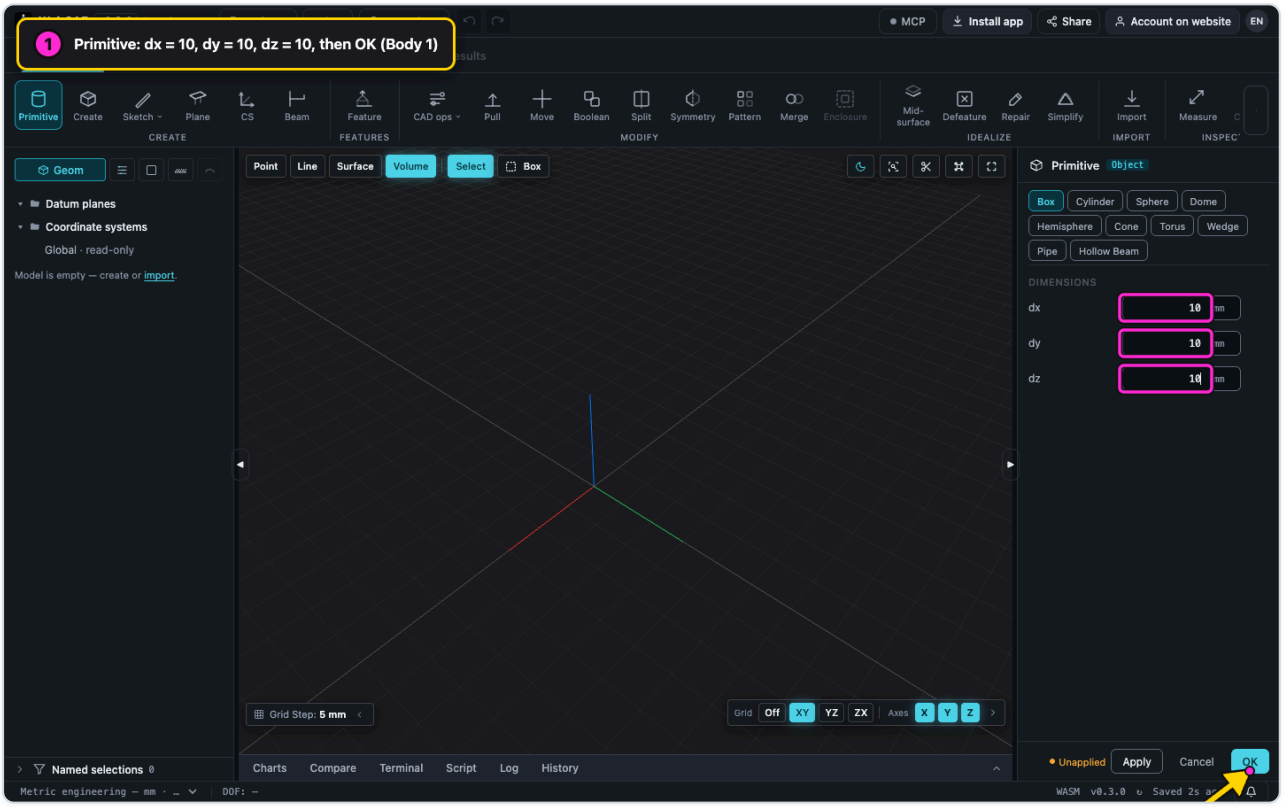
Outcome An assembly of two $10 \times 10 \times 10$ mm cubes without overlap (second cube 800 mm^3) and an Enclosure body around both with a volume of 2512 mm^3 (WebCAE: 2512.0).

8–9 min | WebCAE web (browser), geometry engine OpenCASCADE in WASM

Run: 2026-11-04-32ffaed2 / F16

STEP 1 Create the first cube

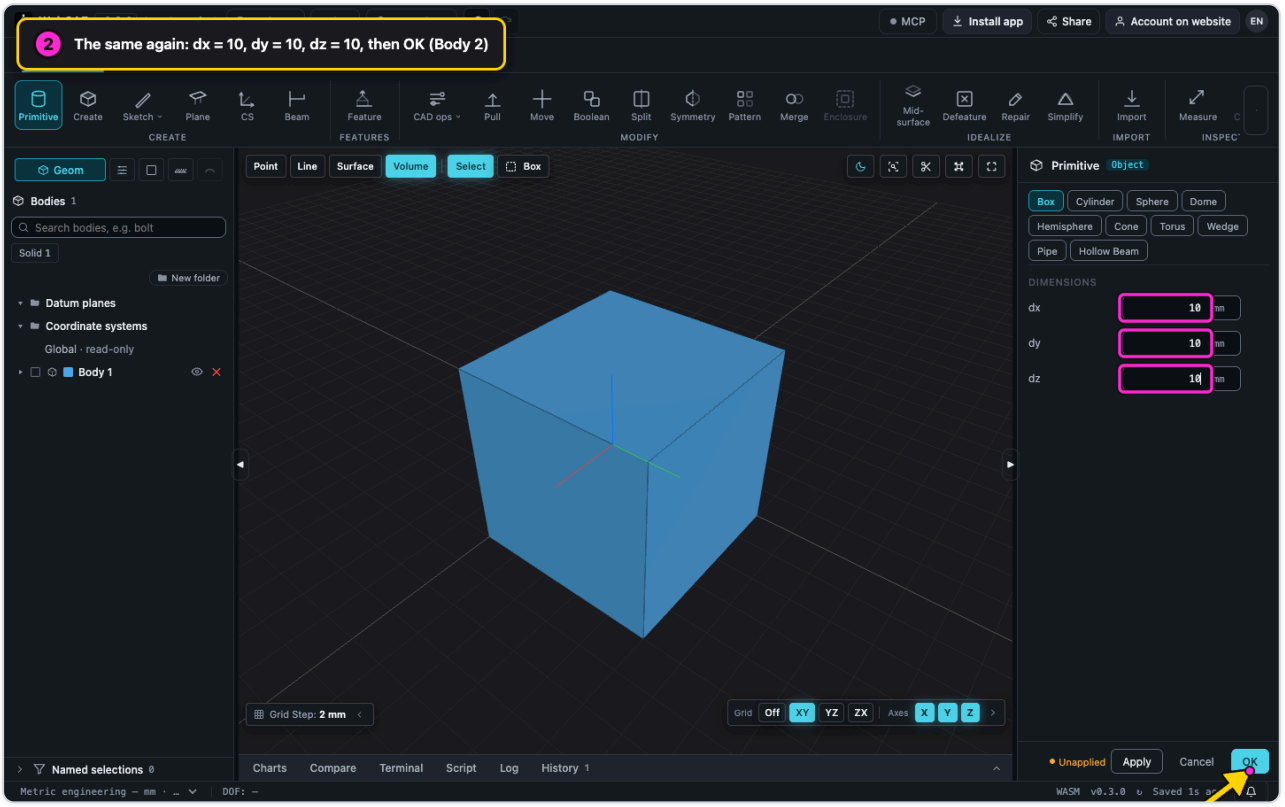
Open **Geometry**, click **Primitive** (shape **Box**), type `10` in **dx**, `10` in **dy**, `10` in **dz**, click **OK**. Body 1 appears at the origin.



STEP 2

Create the second cube

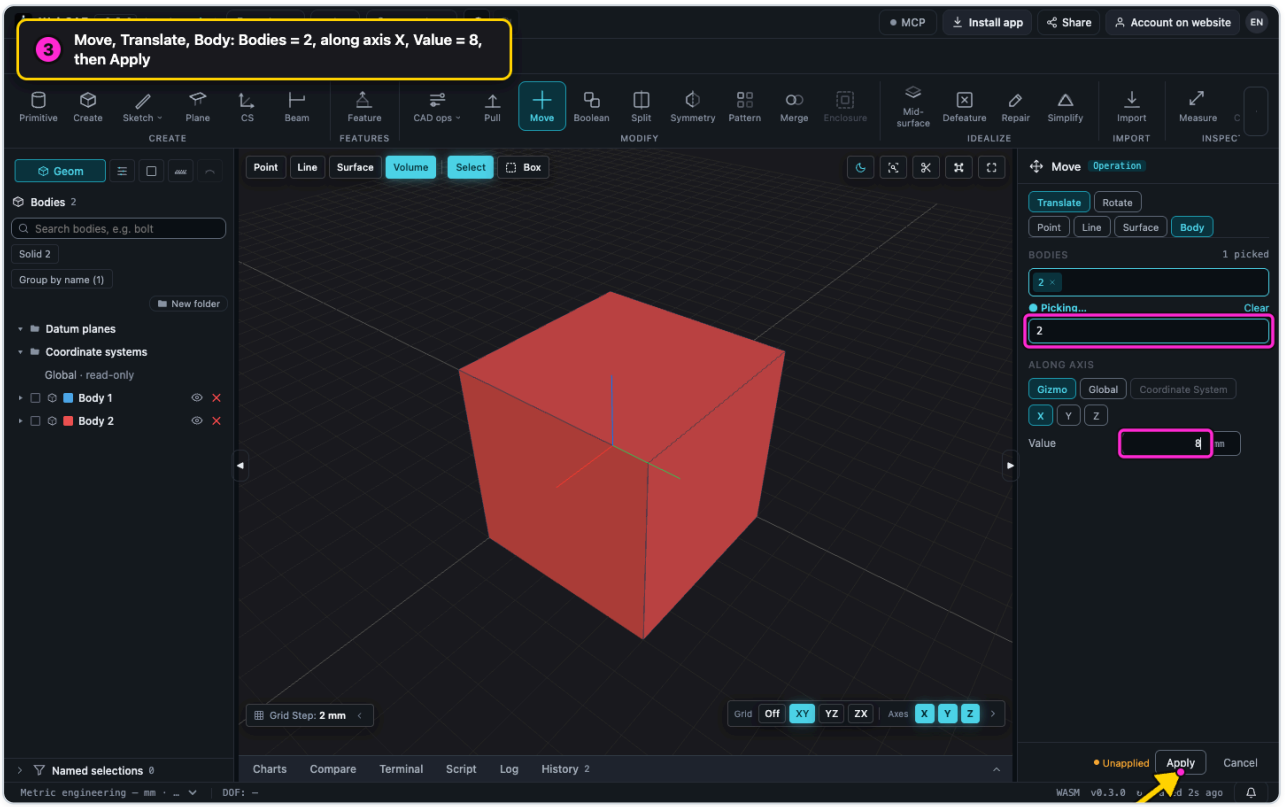
Repeat: **Primitive**, `10`, `10`, `10`, **OK**. Body 2 is created at the same place, exactly on top of Body 1.



STEP 3

Move the second cube along X

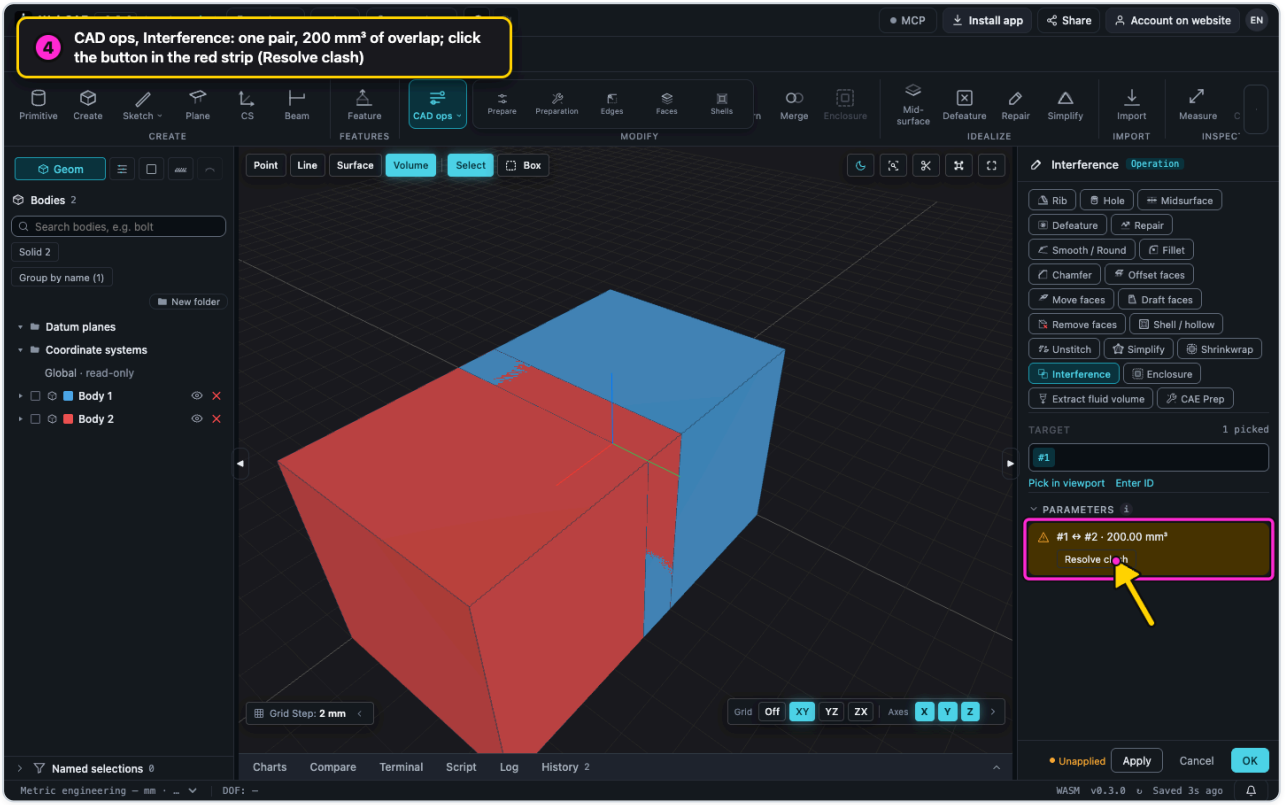
Click **Move** in the ribbon. In the panel keep **Translate**, click **Body** (the pick mode, not Point, Line or Surface), type `2` in the **Body #** box and press Enter (**Bodies** shows 1 picked), under **Along axis** keep **Gizmo** and click **X**, type `8` in **Value**, click **Apply**, then **Cancel**. Body 2 now spans X from 8 to 18 mm, so it overlaps Body 1 (0 to 10 mm) by 2 mm.



STEP 4

Find the overlap and resolve it

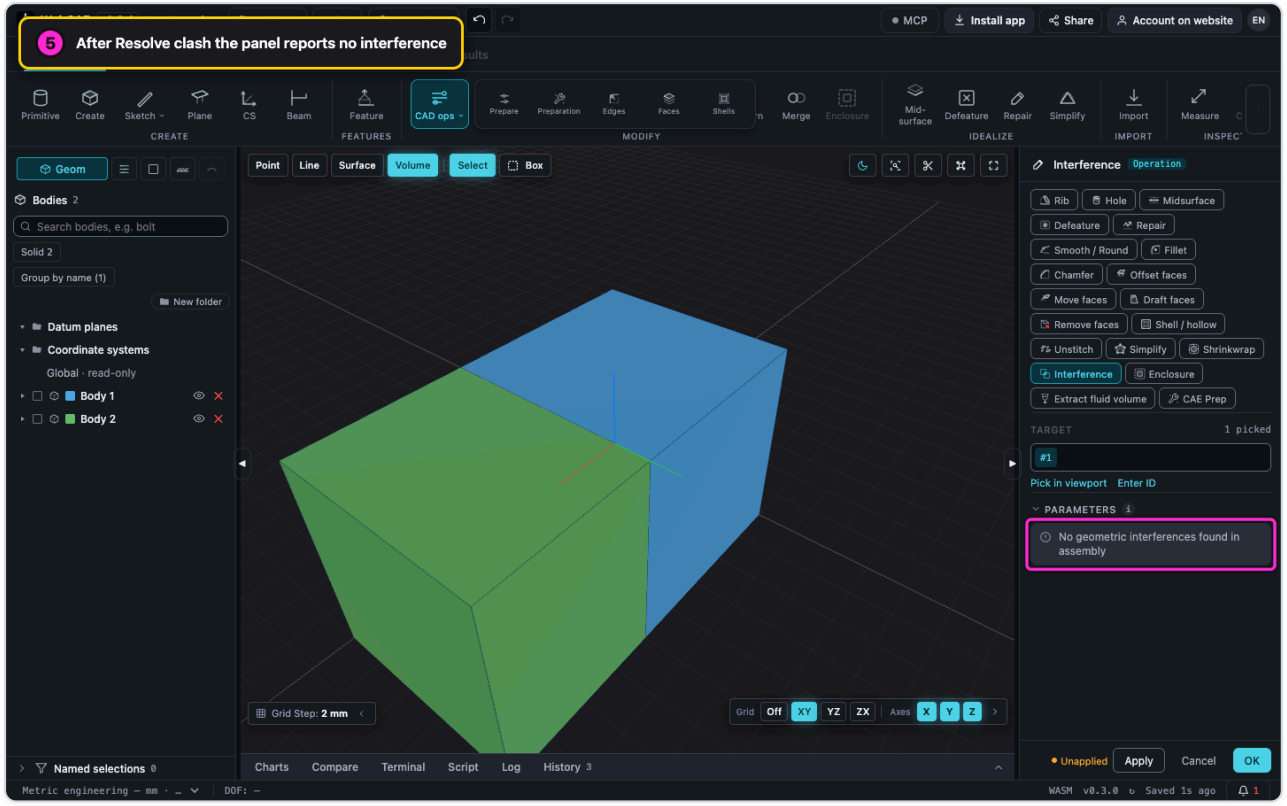
Click **CAD ops** in the ribbon and choose **Interference**. The panel checks the assembly and lists one pair in a red strip: **#1 ↔ #2** with the overlap volume 200.00 mm^3 ($2 \times 10 \times 10 = 200$). Click the button in that strip (**Resolve clash**).



STEP 5

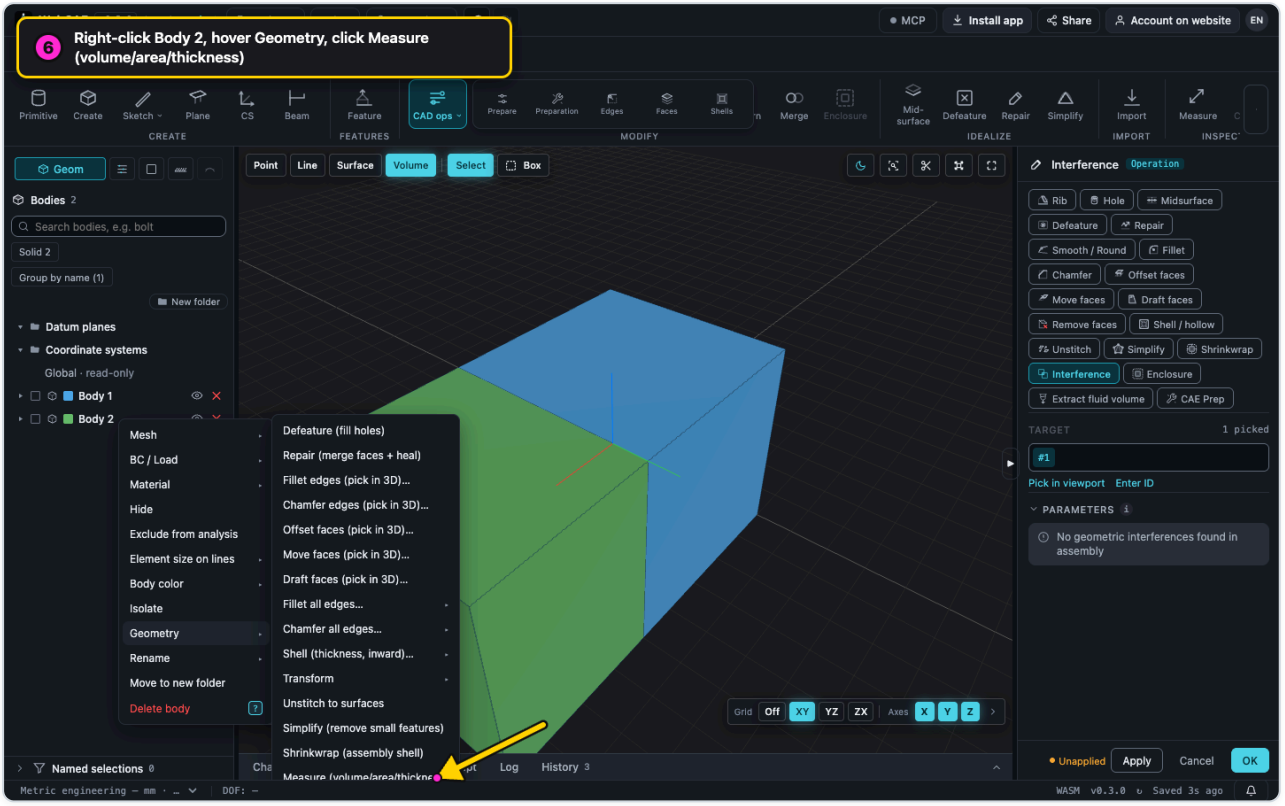
Check that no overlap is left

After the resolve the panel shows the strip **No geometric interferences found in assembly** instead of the pair.



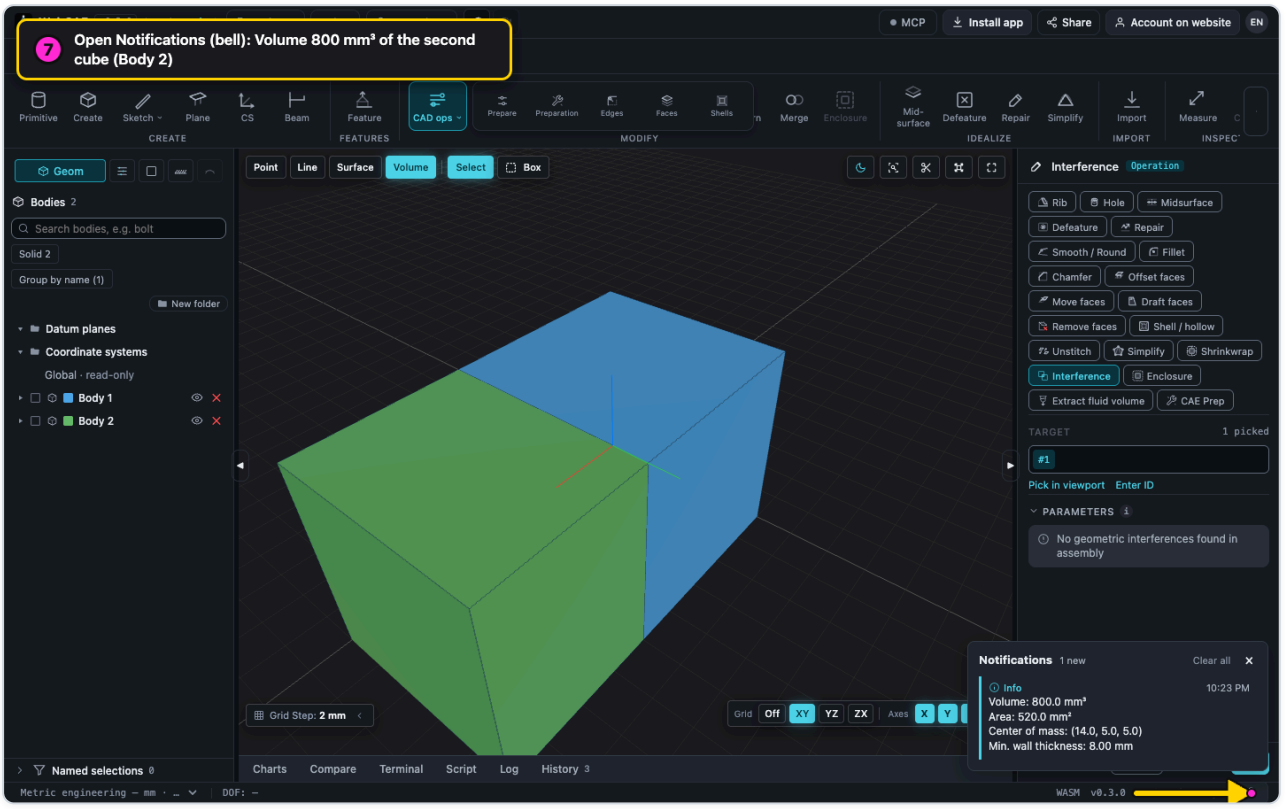
STEP 6 Open the Measure command of the second cube

Right-click **Body 2** in the tree (the cube that was moved), hover **Geometry** and click **Measure (volume/area/thickness)** at the bottom of the submenu.



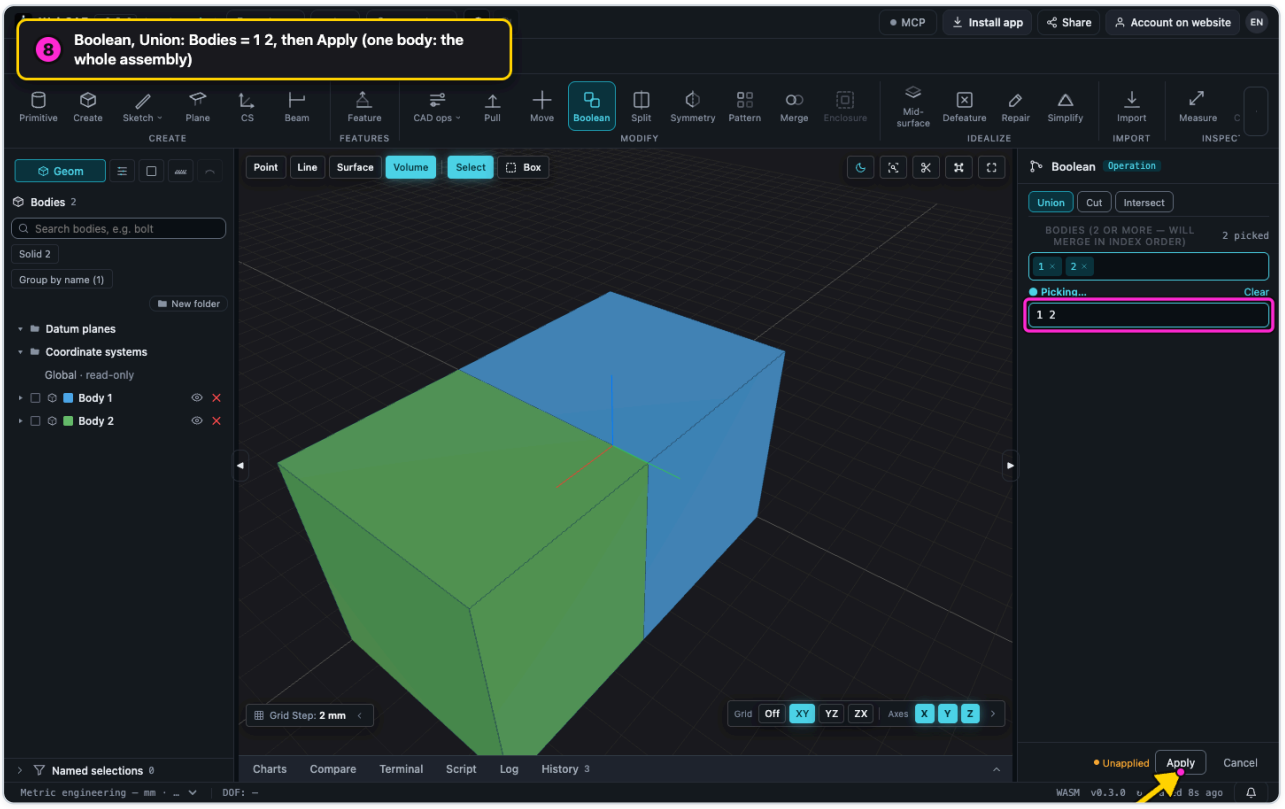
STEP 7 Read the volume of the second cube

Click the bell at the bottom right to open **Notifications**. **Volume**: 800.0 mm^3: Resolve cut the 2 mm overlap out of the second cube, leaving $8 \times 10 \times 10 = 800$.



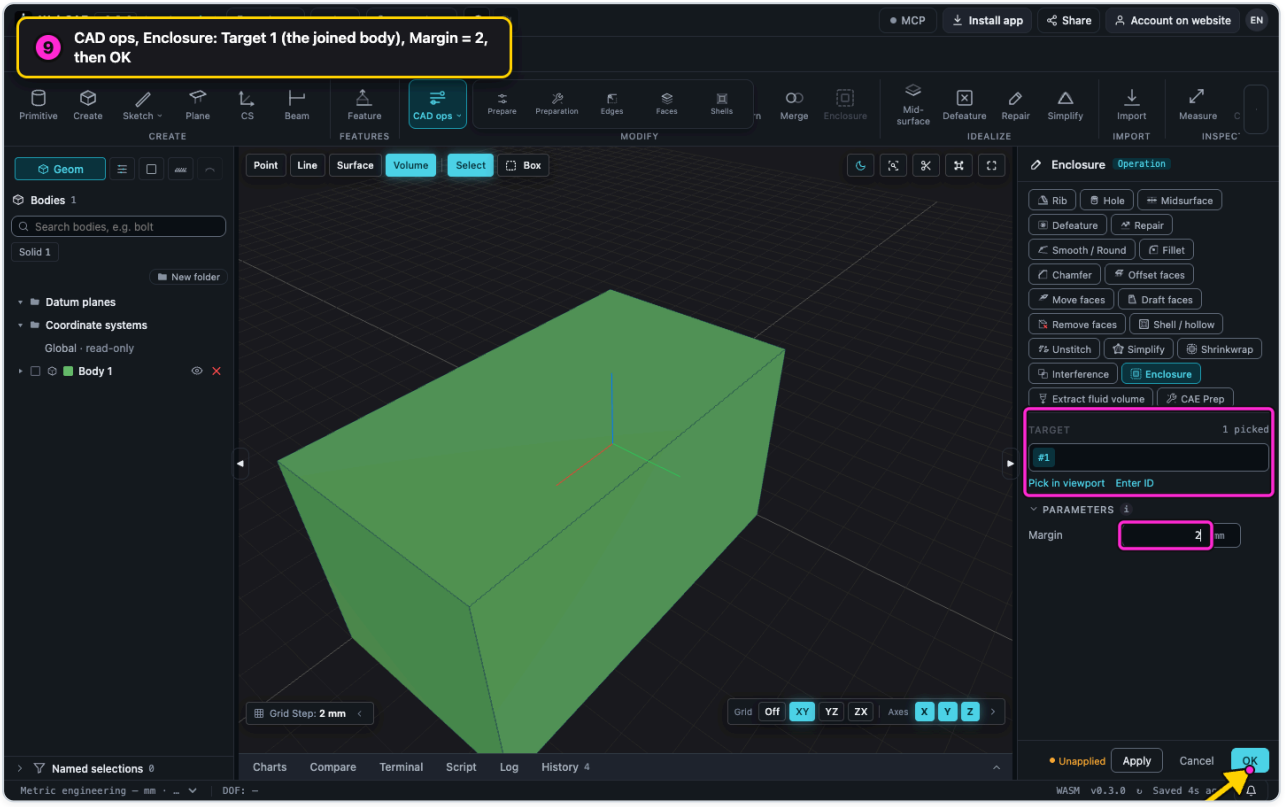
STEP 8 Join the two cubes

Enclosure works on one body, so join the assembly first. Click **Boolean** in the ribbon, choose **Union**, type `1 2` in the **Body #** box and press Enter, click **Apply**, then **Cancel**. One body remains: $18 \times 10 \times 10$ mm, 1800 mm^3.



STEP 9 Build the enclosure around the whole assembly

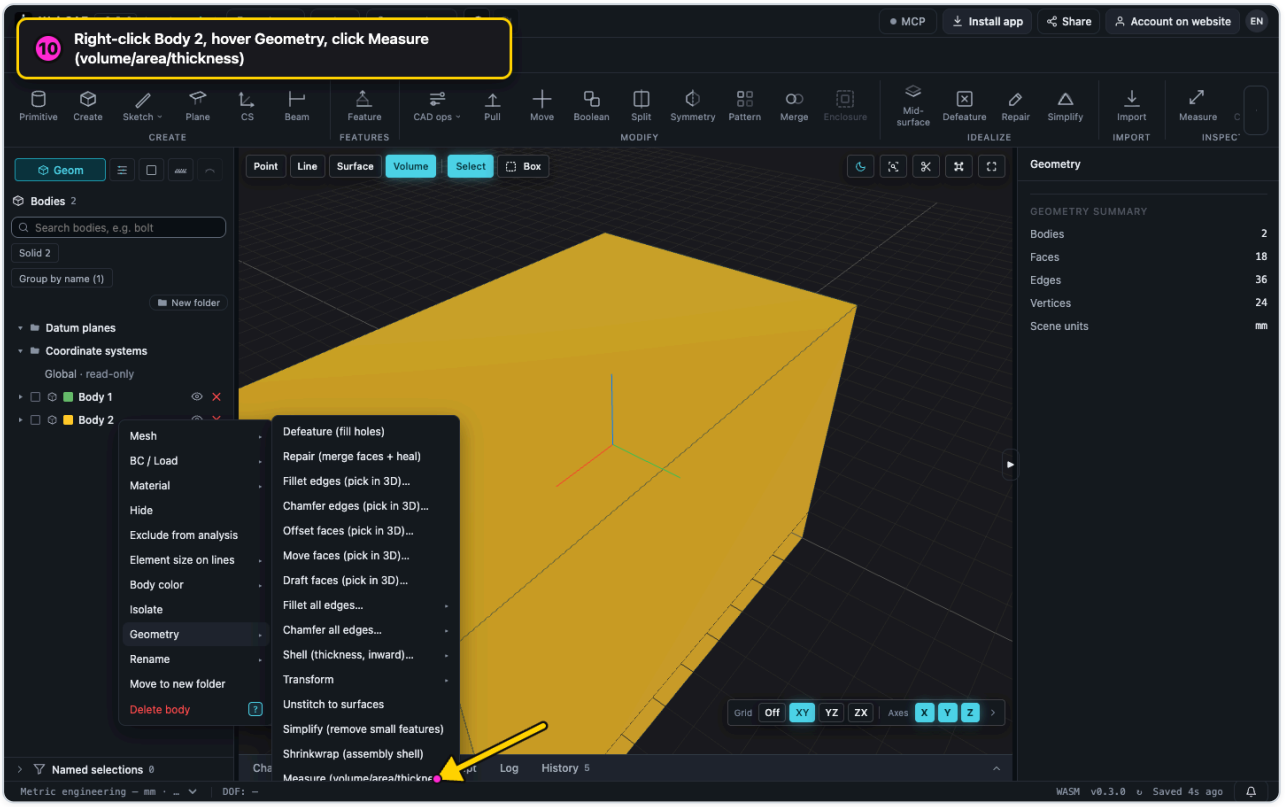
Click **CAD ops** and choose **Enclosure**. In **Target** click **Enter ID**, type `1` (the joined body) and press Enter, type `2` in **Enclosure margin** and click **OK**. A new body **Enclosure(...)** appears: a box 2 mm larger on every side than the assembly, minus the assembly itself, that is a fluid domain around both cubes.



STEP 10

Open the Measure command of the enclosure

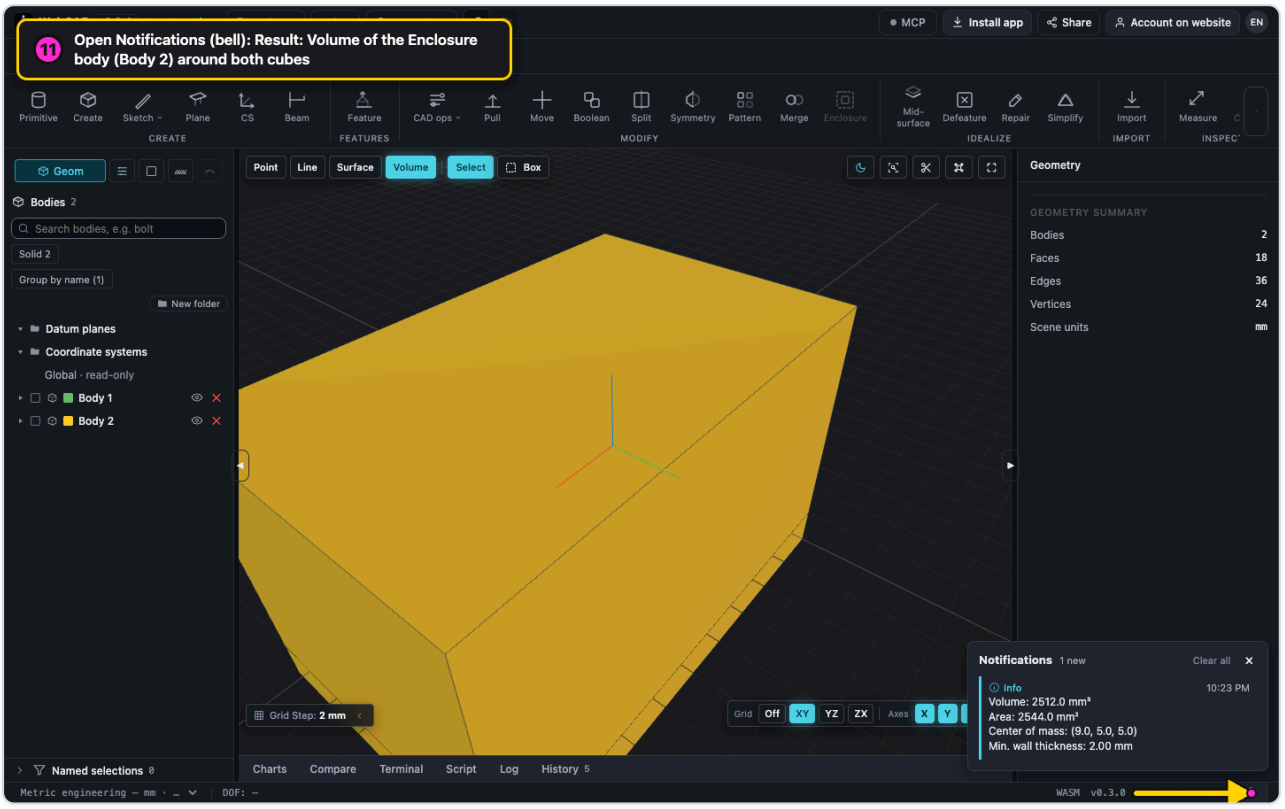
Right-click the new enclosure body (**Body 2**), hover **Geometry**, click **Measure (volume/area/thickness)**.



STEP 11

Result: volume of the fluid domain

Open **Notifications** (bell). **Volume**: 2512.0 mm^3. By hand: the outer box is $(18+4)\times(10+4)\times(10+4)=22\times14\times14=4312$, minus the assembly 1800, gives 2512.



VERIFICATION BENCHMARK

Check against the formula

Quantity	By hand	WebCAE web (browser)	WebCAE desktop (native)	Difference (web)
Overlap of the cubes	$2 \times 10 \times 10 = 200\ \mathrm{mm}^3$	200.00 mm^3	— (geometry is OpenCASCADE in WASM, no native entry)	0.00 %
Pairs in the overlap list	1	1	—	—
Volume of the second cube after Resolve	$1000 - 200 = 800\ \mathrm{mm}^3$	800.0 mm^3	—	0.00 %
Volume of the enclosure around both cubes	$22 \times 14 \times 14 - 1800 = 2512\ \mathrm{mm}^3$	2512.0 mm^3	—	0.00 %

FORMULA

Margin $m = 2$ mm on each side: outer box $(L + 2m)(W + 2m)(H + 2m)$ with the assembly $L \times W \times H = 18 \times 10 \times 10$ gives $22 \times 14 \times 14 = 4312$; the fluid volume is this minus the 1800 mm^3 of the assembly. The desktop numbers in docs/2026-11-05-tutorials-101-wave1-native-runs-findings.md describe a different set-up (enclosure around one cube, 1744 mm^3) and are not comparable here: the geometry engine is the same WASM OpenCASCADE on both platforms.

TROUBLESHOOTING & NOTES

Common mistakes

- **Enclosure on one cube only:** Enclosure takes a single body; selecting only Body 1 gives a fluid box around that cube ($14 \times 14 \times 14 - 1000 = 1744\ \mathrm{mm}^3$) and leaves the other cube inside it. Join the cubes first (step 8).
- **Move in Surface mode:** in the Move panel click **Body**, otherwise the pick takes a face and nothing moves.
- **Skipping Resolve before Union:** a union of overlapping cubes also works, but the second cube would never be corrected and step 7 would not apply.

RECOMMENDED PROGRESSION

Where to go next

Congratulations on completing the WebCAE 101 series. Explore the full DSL Command Reference and open the WebCAE Workbench to solve your own engineering problems.

WebCAE 101

LESSON 17 · F17

Spur Gear Pair: Static Tooth Contact Under Driving Torque

Goal Import a two-body involute spur gear pair ($z_1 = 12$, $z_2 = 16$, module $m = 4.0\ \mathrm{mm}$, pressure angle $\alpha = 20^\circ$, face width $b = 6\ \mathrm{mm}$, center distance $a = 56\ \mathrm{mm}$), mesh both gears with quadratic tetrahedra (**P2 / Tet10**), detect and assign unilateral **Frictionless (Penalty)** tooth contact, support both bores with **Roller / Normal** ($u_n = 0$), hold back faces in u_z, stop the driven-wheel keyway laterally with **Normal** support, and apply **90 MPa** pressure on the four driving spline faces of the pinion (equivalent clean torque $T \approx 66.46\ \mathrm{N \cdot m}$). Verify elastic $|u|_{\max}$, peak σ_{vM}, p_{contact}, and integrated normal tooth force $F_n = T/r_{b1}$ in the **fixed meshing position**. WebCAE contact assemblies use small-displacement Mortar contact: **prescribed shaft rotation or a full mechanism revolution is not modeled** (attempts beyond about 5.7° remote rotation are rejected).

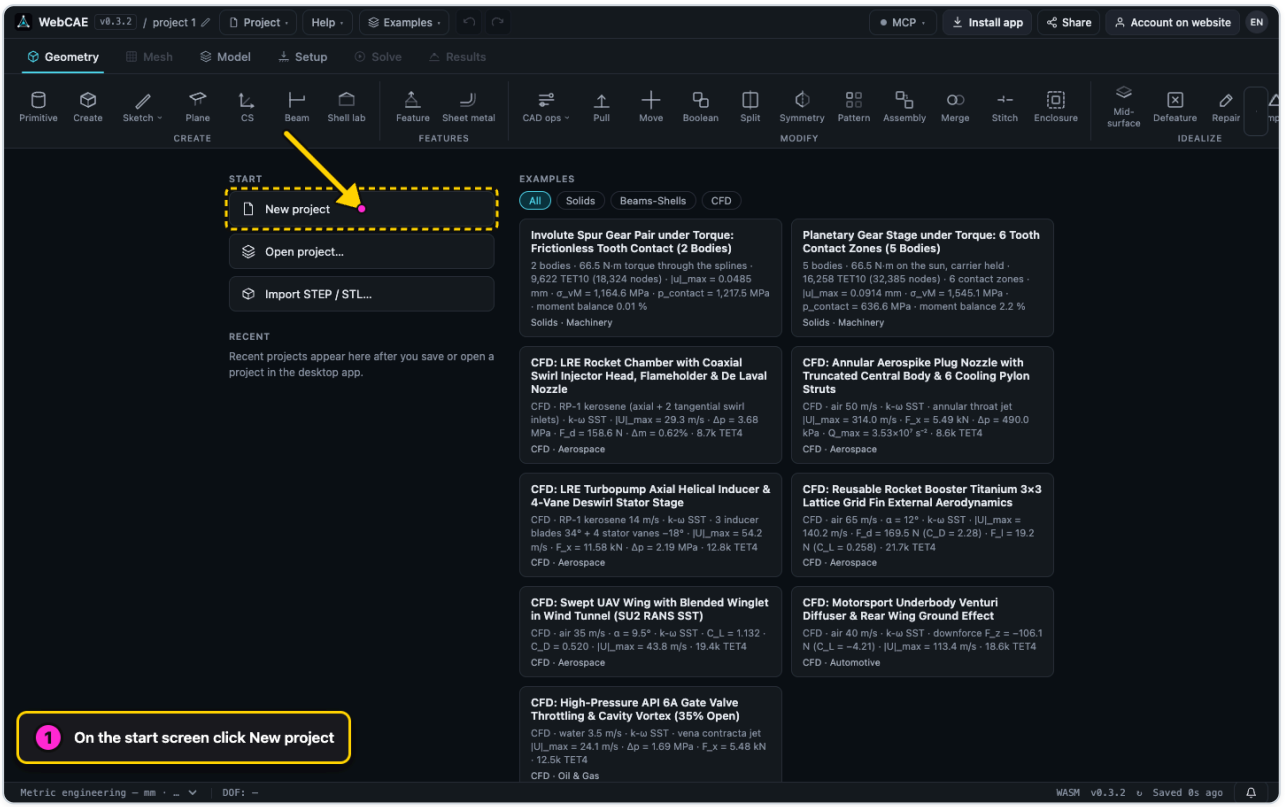
Outcome Converged Mortar contact after the native solver ramps spline pressure across **20** incremental load steps ($\lambda = 5\% \ldots 100\%$; **Step 20/20**, $\lambda = 100\%$ in the Results **LOAD STEP** slider and bottom timeline). Pre-built showcase bundles `gear_mesh_contact` / `spur_gear_pair` also ship **16-step** rotational tooth-engagement histories (60° / −45°). Elastic $|u|_{\max} = 0.0501\ \mathrm{mm}$, peak von Mises stress $\sigma_{\mathrm{vM,max}} = 1477.3\ \mathrm{MPa}$ (Results legend: nodal P2 field on the browser mesh; not comparable with the native solver field), peak contact pressure $p_n = 1226.4\ \mathrm{MPa}$, integrated normal tooth force $|R| = 3061.2\ \mathrm{N}$ vs $F_n = T/r_{b1} = 2946.9\ \mathrm{N}$ (3.88 % difference). **Tet10: 10938** elements.

7–8 min | WebCAE native contact solver (`webcae_solve`) | Run: 2026-11-04-32ffaed2 / F17

STEP 1 Start a new project

Open WebCAE and click **New project** on the Start screen (the arrow on the frame).

WHY Starts a clean document with the unified entity numbering journal reset to 1.

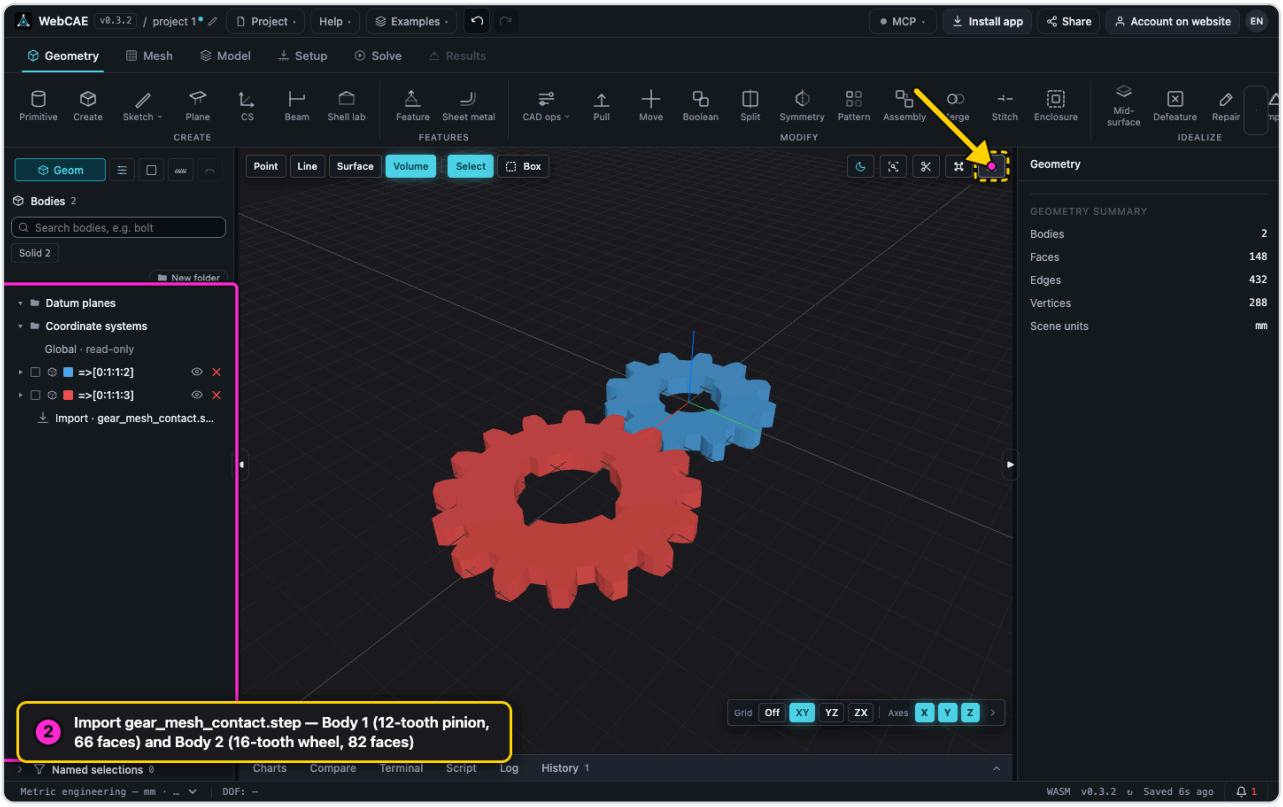


STEP 2

Import the 2-body spur gear STEP assembly

In the **Geometry** tab click **Import** and select `gear_mesh_contact.step`. Once both bodies appear in the tree (**Body 1** is the 12-tooth driving pinion at $(0, 0, 0)$ with 66 faces; **Body 2** is the 16-tooth driven wheel at $x = 56\ \text{mm}$ with 82 faces), click **Fit all content to screen** at the top right of the viewport.

> **WHY** The two gears are imported in their exact theoretical meshing alignment with their engaged involute flanks touching along the line of action.

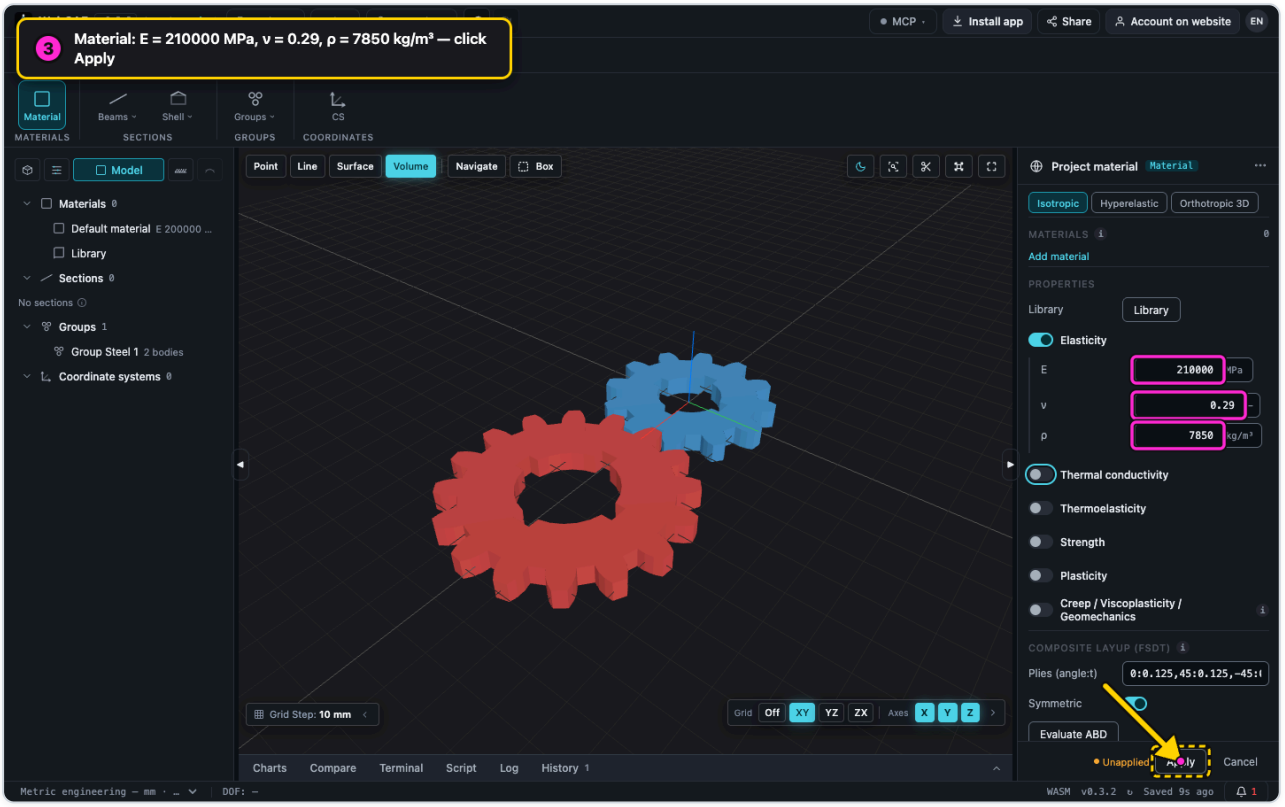


STEP 3

Define the case-hardened gear steel material

Open the **Model** tab, click **Material**, enter Young's modulus $E = 210\,000\ \mathrm{MPa}$, Poisson's ratio $\nu = 0.29$, and density $\rho = 7\,850\ \mathrm{kg/m^3}$, then click **Apply** (the arrow on the frame).

WHY Case-hardened alloy gear steel (18CrNiMo7-6 / AISI 8620) provides the elastic stiffness governing Hertzian tooth flank contact and root bending deflection.

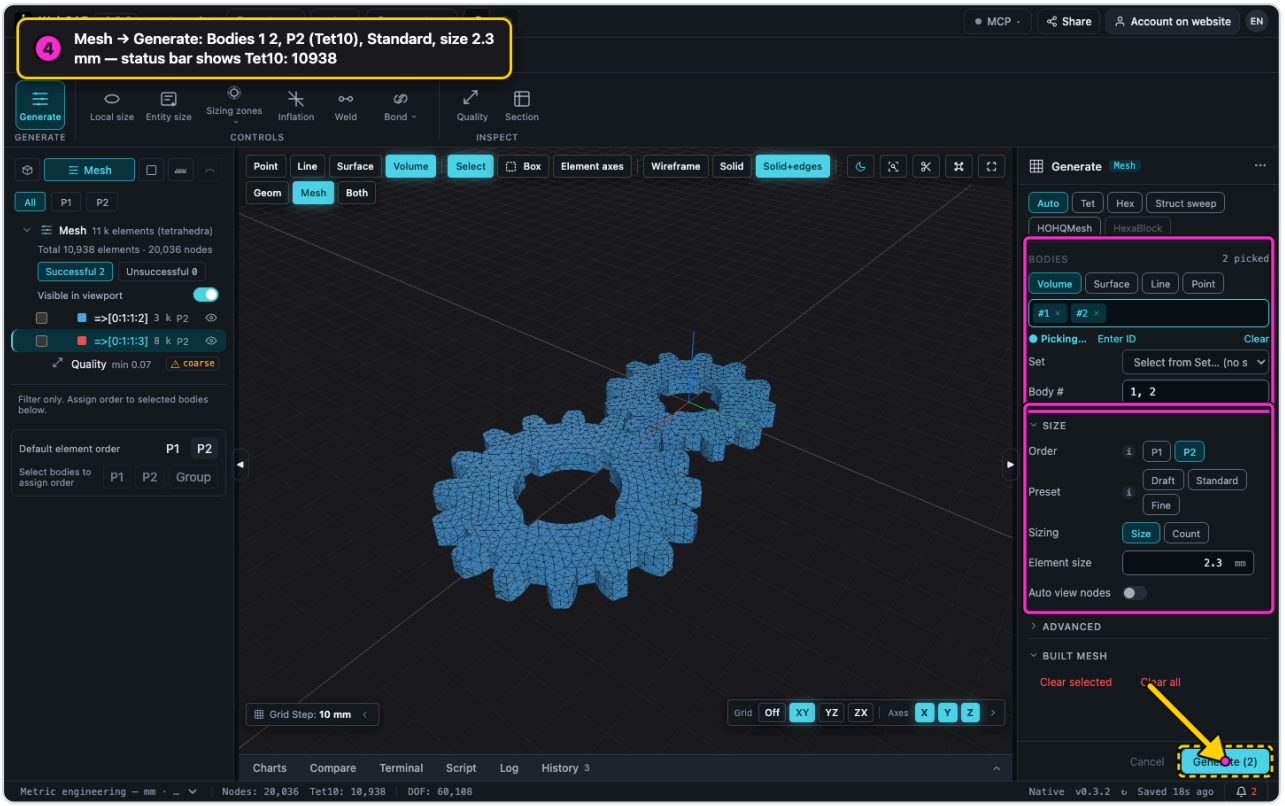


STEP 4

Mesh both gears with quadratic tetrahedra (P2 / Tet10)

Open the **Mesh** tab, click **Generate**, click **Enter ID** under **Target**, type `1 2` and press **Enter**. Select element order **P2** (10-node quadratic tetrahedra) and set the element size to `2.3` mm, then click **Generate**. The status bar shows **Tet10: 10938** (20036 nodes).

WHY Quadratic **Tet10** elements accurately capture curved involute tooth flanks and linear stress gradients across the tooth root without shear locking.

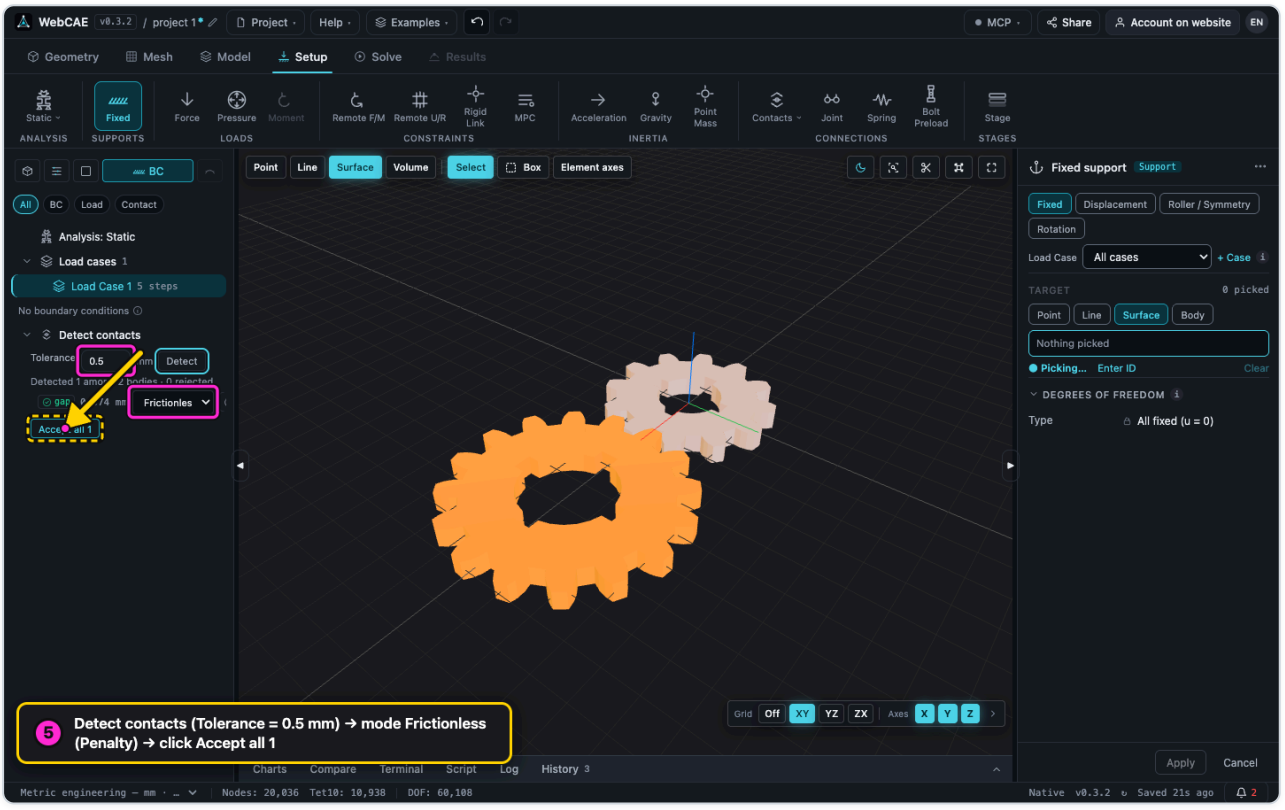


STEP 5

Detect the contacting tooth flanks and choose Frictionless (Penalty)

Open the **Setup** tab and click **Detect contacts** in the tree. Set **Tolerance (mm)** to `0.5` and click **Detect**. In the detected proposal row (**Body 1 ⇔ Body 2**), select **Frictionless (Penalty)** from the contact mode list and click **Accept all 1** (the arrow on the frame).

WHY Unlike **Bonded (Exact MPC)**, which glues both sides in tension and compression, **Frictionless (Penalty)** enforces unilateral normal non-penetration ($g_n \geq 0, p_n \geq 0$) while allowing tangential sliding along the involute profile and free separation on the non-working back flanks.

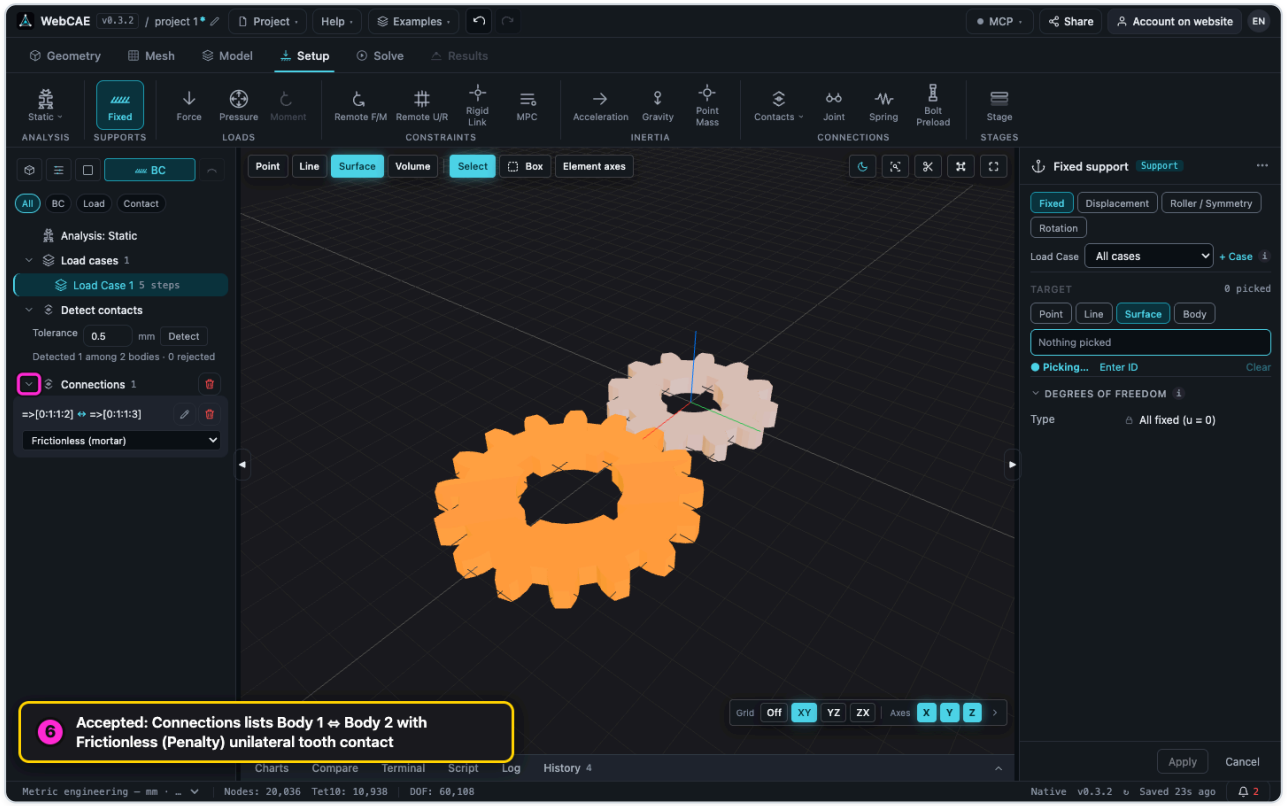


STEP 6

Check the accepted nonlinear contact pair in the tree

Under **Connections** in the Setup tree, the active pair **Body 1 ⇔ Body 2** with **Frictionless (Penalty)** is listed (framed).

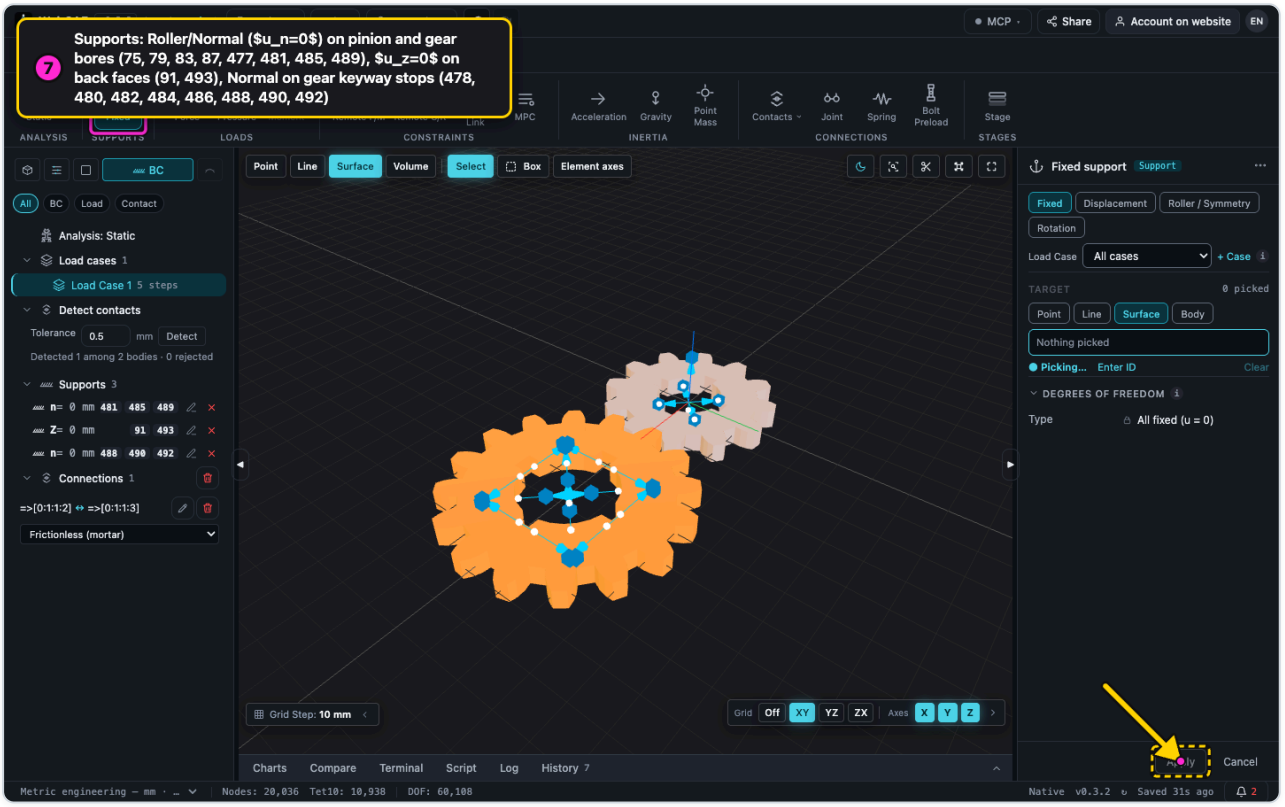
WHY Confirms that the nonlinear contact interface is registered for the assembly solver.



STEP 7 Support both shafts and the driven-wheel keyway (Roller / Normal + u_z)

In the **Setup** ribbon choose **Fixed** → **Roller / Normal** (`variant-roller`), click **Enter ID**, type the pinion and gear bore quadrants (tags `pinion_bore` , `gear_bore` in `gear_mesh_contact.tags.json`), and click **Apply**. Choose **Displacement**, lock **X** and **Y**, leave **Z** constrained ($u_z = 0$) on both back faces (`pinion_back` , `gear_back`), and click **Apply**. Return to **Roller / Normal** and apply it to the gear keyway stop faces (`gear_keyway_stop`).

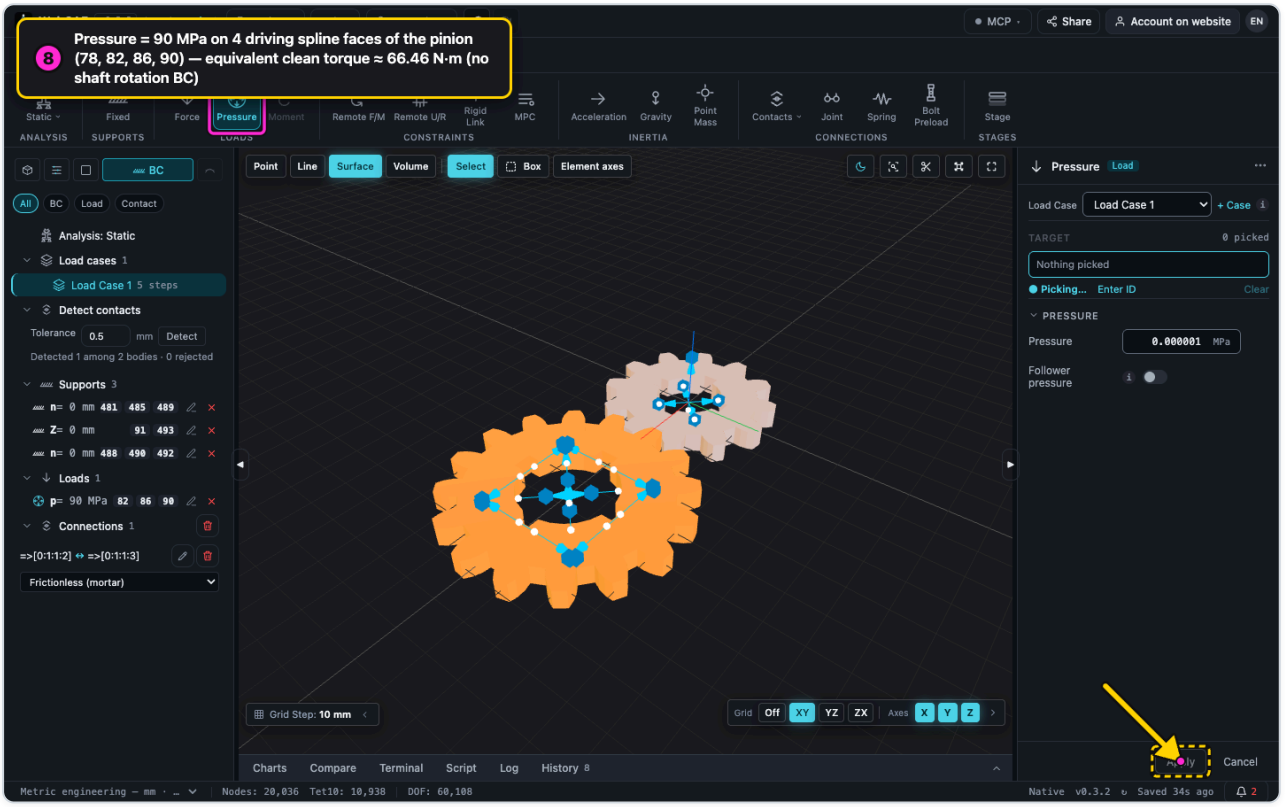
> **WHY** Matches the verified `gear_mesh_contact` assembly: each bore is a roller support (axis fixed, spin free), thrust is taken by the back faces, and the driven-wheel keyway cannot slide sideways off the shaft.



STEP 8 Apply driving torque via 90 MPa pressure on the pinion splines

Click **Pressure** (`button[data-tile-id=pressure]`) in the Setup ribbon, click **Enter ID**, type the four driving spline faces (`pinion_keyway_drive`), enter **90** MPa, and click **Apply**. This reproduces the clean torque $T \approx 66.46\ \mathrm{N \cdot m}$ of the native case without prescribing shaft rotation.

> **WHY** Delivers torsional load through face pressure on the splines. **Do not** use **Remote Motion** with large R_z here — contact assemblies reject rotations beyond the small-displacement limit and do not roll the gear mesh through multiple teeth.

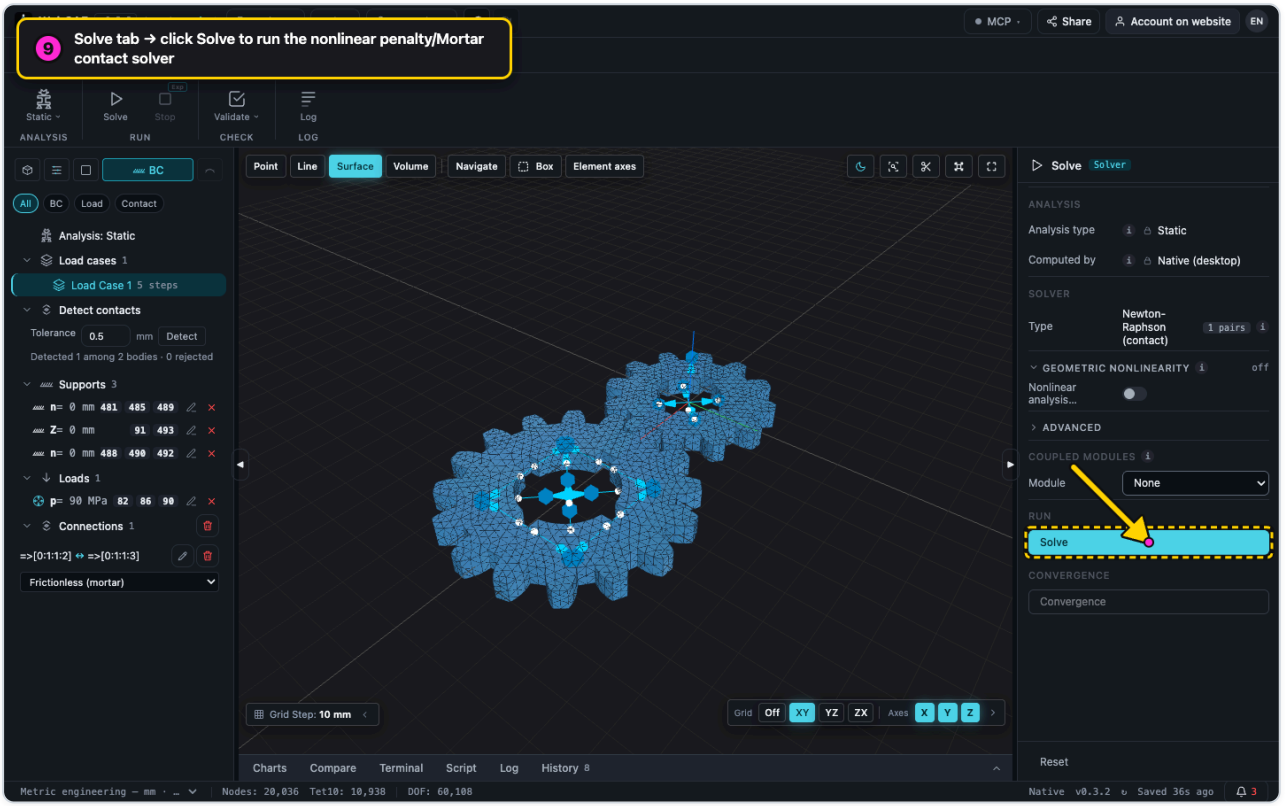


STEP 9

Solve the static nonlinear contact assembly

Open the **Solve** tab and click **Solve** (the arrow on the frame). WebCAE assembles the quadratic 2-body system, ramps the spline pressure to full value inside the native Mortar solver, and iterates the unilateral contact active set to convergence.

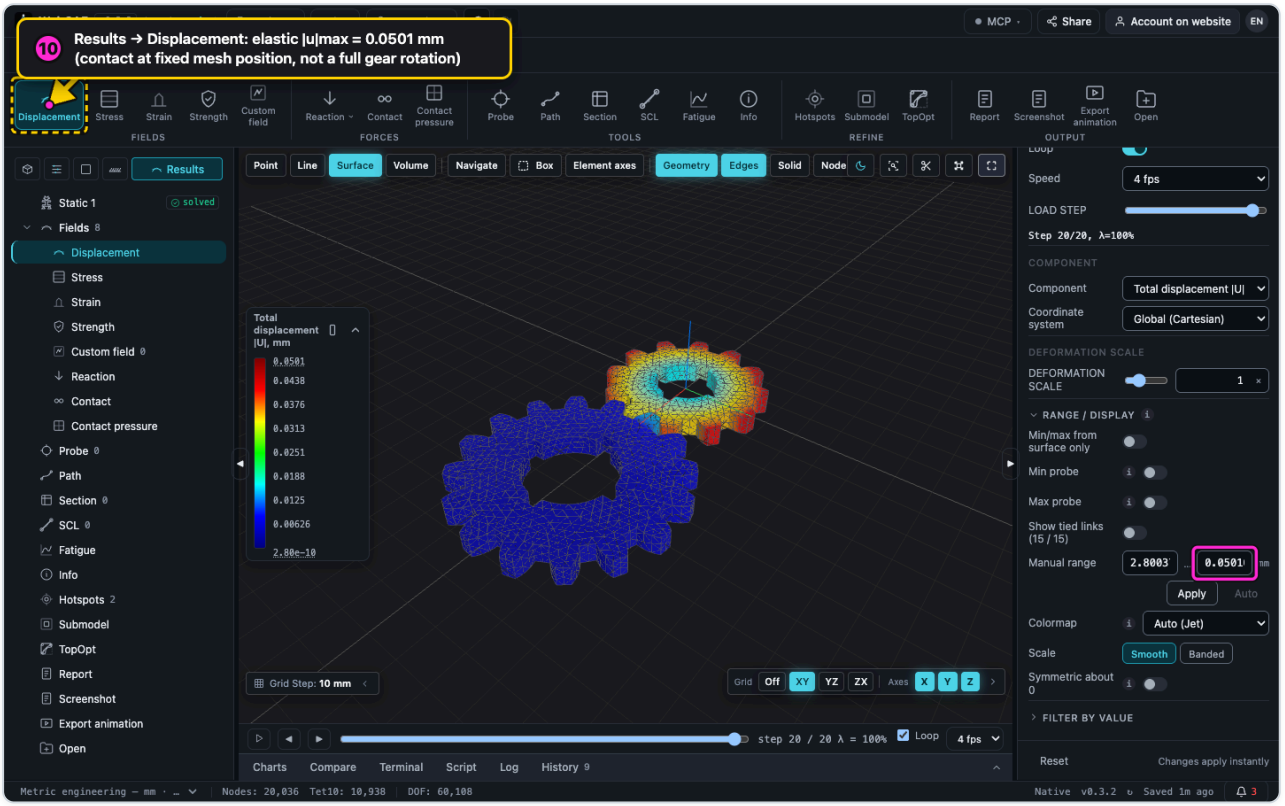
WHY Computes elastic tooth deformation and Hertzian contact in the **imported meshing pose**. Kinematic multi-tooth revolution is outside this analysis type.



STEP 10 Inspect elastic displacement $|u|$

In the **Results** tab click **Displacement** and set **Deformation scale = 1**. The **Manual range** maximum reads **0.0501 mm** — small elastic motion at the engaged flanks and roots, not a visible 90° shaft turn.

> **WHY** Confirms the small-displacement contact response. Large chordal displacements from finite gear rotation are **not** part of this solve.

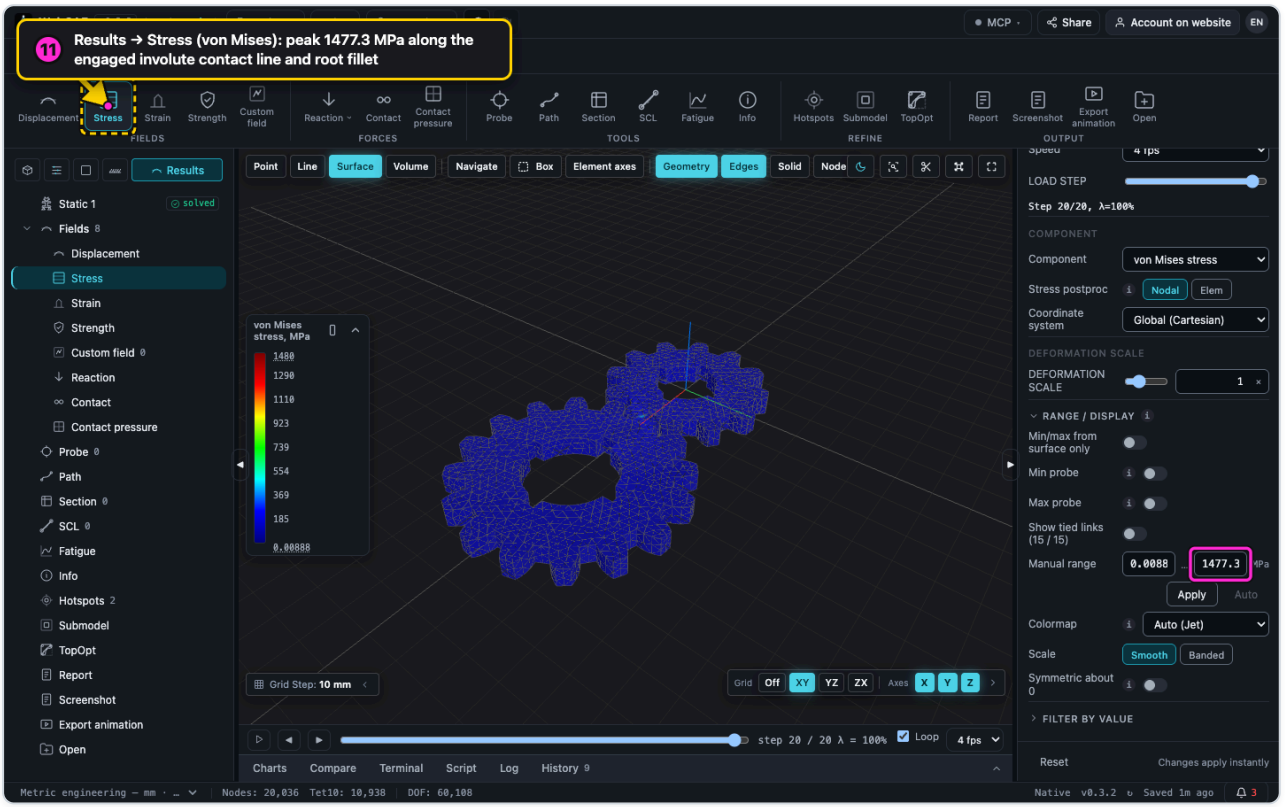


STEP 11

Inspect the von Mises stress field

Click **Stress**. The **Manual range** maximum reads **1477.3 MPa**, localized along the engaged involute contact line and at the root fillets of the meshing teeth.

WHY Shows peak bending/Hertzian combination at the active flank. Browser meshing runs somewhat hotter than the native verification mesh (1164.6 MPa reference on `gear_mesh_contact`).

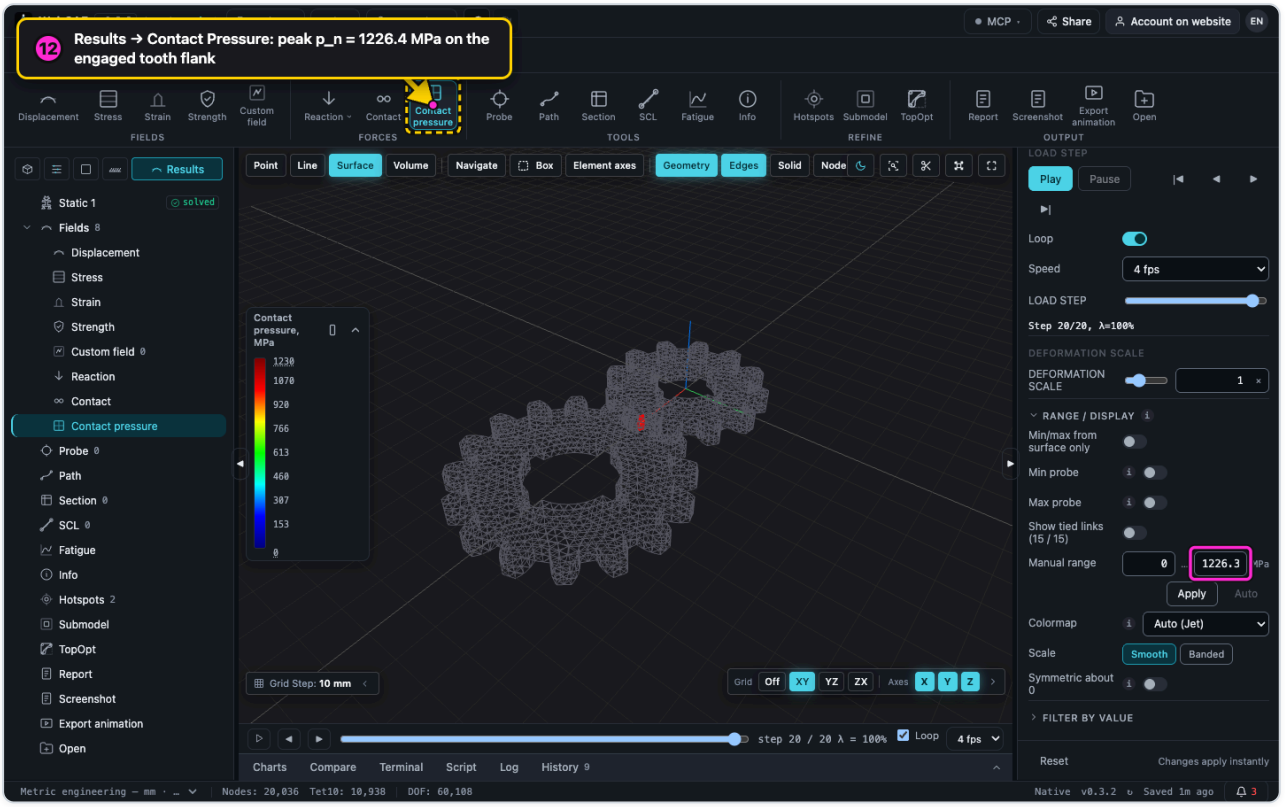


STEP 12

Inspect contact pressure on the active tooth flank

Click **Contact Pressure**. The **Manual range** maximum reads **1226.4 MPa** along the engaged flank, with zero pressure on separated back flanks.

WHY Visualizes unilateral p_n whose integral over active facets yields $|R| = 3061.2\ \mathrm{N}$, comparable to $F_n = T/r_{b1} = 2946.9\ \mathrm{N}$.



VERIFICATION BENCHMARK

Check against the formula

Quantity	Analytical formula	WebCAE (native `webcae_solve`)	Difference	Where you see it
Elastic $\lvert u\rvert_{\max}$	Small-displacement tooth contact ($\lvert u\rvert \ll 1$ mm)	0.0501 mm	—	Step 10 (Manual range max)
Normal tooth force $\lvert R\rvert$	$F_n = \frac{T}{r_{b1}} = \frac{66.46}{24\cos 20^\circ \times 10^{-3}} = 2946.9$ N	3061.2 N	3.88 %	Integrated contact normal force (`contactSummary.forceN`)
Peak contact pressure p_n	Native `gear_mesh_contact` reference 1217.5 MPa	1226.4 MPa	0.73 %	Step 12 (Manual range max)
Peak von Mises stress σ_{vM} (legend)	Not compared: the legend is the nodal P2 field on the browser mesh (10938 Tet10); the native 1164.6 MPa is the solver field on another mesh (9622 Tet10)	1477.3 MPa	—	Step 11
Driving torque (spline pressure)	90 MPa on four `pinion_keyway_drive` faces	66.46 N · m	—	Setup → Pressure

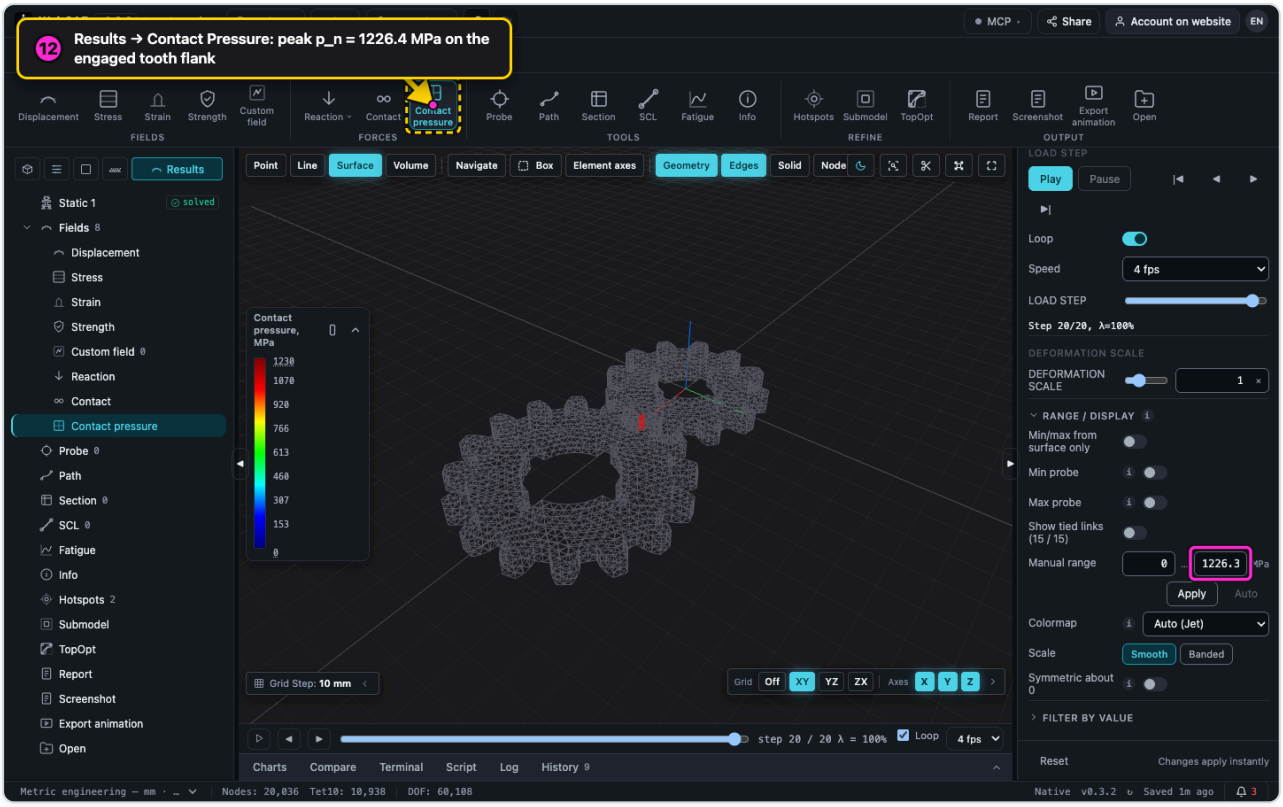
FORMULA

Analytical check (force balance).

Pitch radius $r_{p1} = \frac{1}{2} m z_1 = 24.0$ mm, base radius $r_{b1} = r_{p1}\cos 20^\circ = 22.5526$ mm. Spline pressure $p = 90$ MPa on the four driving faces gives a clean torque $T \approx 66.46$ N · m (same construction as `cases/assemblies/gear_mesh_contact/case.mts`). The normal tooth force along the line of action is

$$F_n = \frac{T}{r_{b1}} = 2946.9 \text{ N}$$

WebCAE integrates $\lvert R\rvert = 3061.2$ N on the Mortar interface (3.88 %). **Kinematic rotation** of either shaft through many teeth is **not** simulated: contact uses small-displacement theory; use multibody kinematics outside CAE for revolution angles.



TROUBLESHOOTING & NOTES Common mistakes

- **Remote Motion with large R_z on gear bores:** contact assemblies reject prescribed rotations beyond about 5.7° (`contactRotationExceedsSmallDisplacement`) and do not animate a full mesh revolution.
- **Fully fixing both bores with $u_x = u_y = u_z = 0$:** prevents the driven gear from reacting torque through tooth contact; use **Roller / Normal** on bores and u_z on backs.
- **Choosing Bonded (Exact MPC) instead of Frictionless (Penalty):** bonded contact glues non-working back flanks and blocks realistic involute separation.
- **Contact tolerance 0.2 mm on curved flanks:** use **0.5 mm** so the engaged involute patch is detected.

RECOMMENDED PROGRESSION Where to go next

Proceed to lesson **#18 — Planetary Gear Stage in Static Tooth Contact (F18)** for a five-body stage with six frictionless meshes and spline torque on the sun gear.

WebCAE 101

LESSON 18 · F18

Planetary Gear Stage: Static Tooth Contact Under Sun Torque

Goal Import a five-body epicyclic stage (sun $z_s = 12$, three planets $z_p = 9$, ring $z_r = 30$, $m = 4.0\ \mathrm{mm}$, $\alpha = 20°$, $b = 6\ \mathrm{mm}$), mesh with **P2 / Tet10**, accept six **Frictionless** Mortar pairs, support all bores and the ring with **Roller / Normal**, constrain all back faces in u_z, and apply **90 MPa** on the sun spline drive faces ($T \approx 66.46\ \mathrm{N \cdot m}$). Verify elastic $|u|_{\max}$, σ_{vM}, and p_{contact} in the fixed meshing pose. **Full sun or carrier rotation is not modeled** in contact assemblies (small-displacement Mortar contact only).

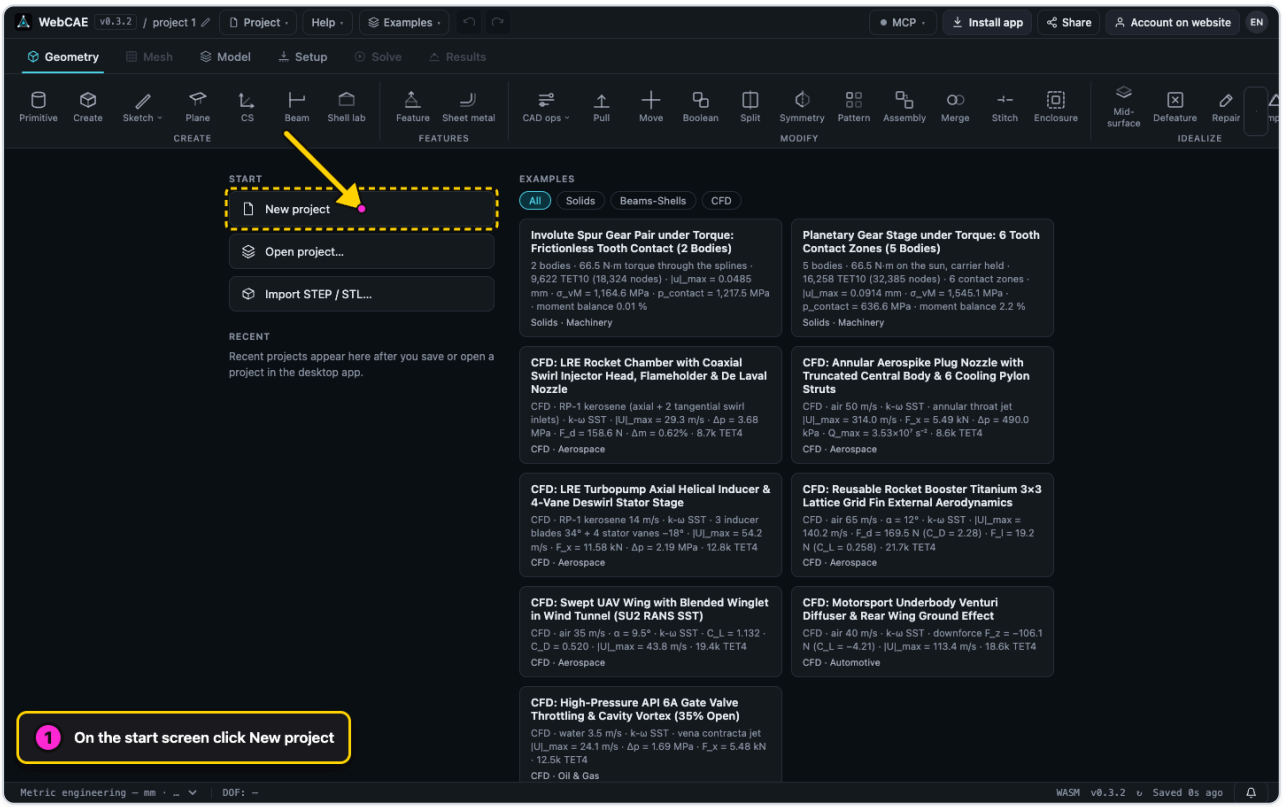
Outcome Converged Mortar contact after the native solver ramps spline pressure across **20** incremental load steps ($\lambda = 5\% \ldots 100\%$; **Step 20/20**, $\lambda = 100\%$ in the Results **LOAD STEP** slider and bottom timeline). Pre-built showcase bundles `planetary_gear_contact` / `planetary_gear_stage` also ship **16-step** orbital tooth-engagement histories (63° sun, 18° carrier orbit). $|u|_{\max} = 0.1006\ \mathrm{mm}$, stress legend peak $\sigma_{\mathrm{vM}} = 2961.3\ \mathrm{MPa}$ (includes spline/keyway stress concentrations; tooth-region peak $r \geq 19\ \mathrm{mm}$ is $1662.0\ \mathrm{MPa}$, native reference $444.6\ \mathrm{MPa}$ on the verification mesh), $p_{\mathrm{contact,max}} = 739.4\ \mathrm{MPa}$, **Tet10: 18582** elements.

8–10 min | WebCAE native contact solver (`webcae_solve`) | Run: 2026-11-04-32ffaed2 / F18

STEP 1 Start a new project

Open WebCAE and click **New project** on the Start screen (the arrow on the frame).

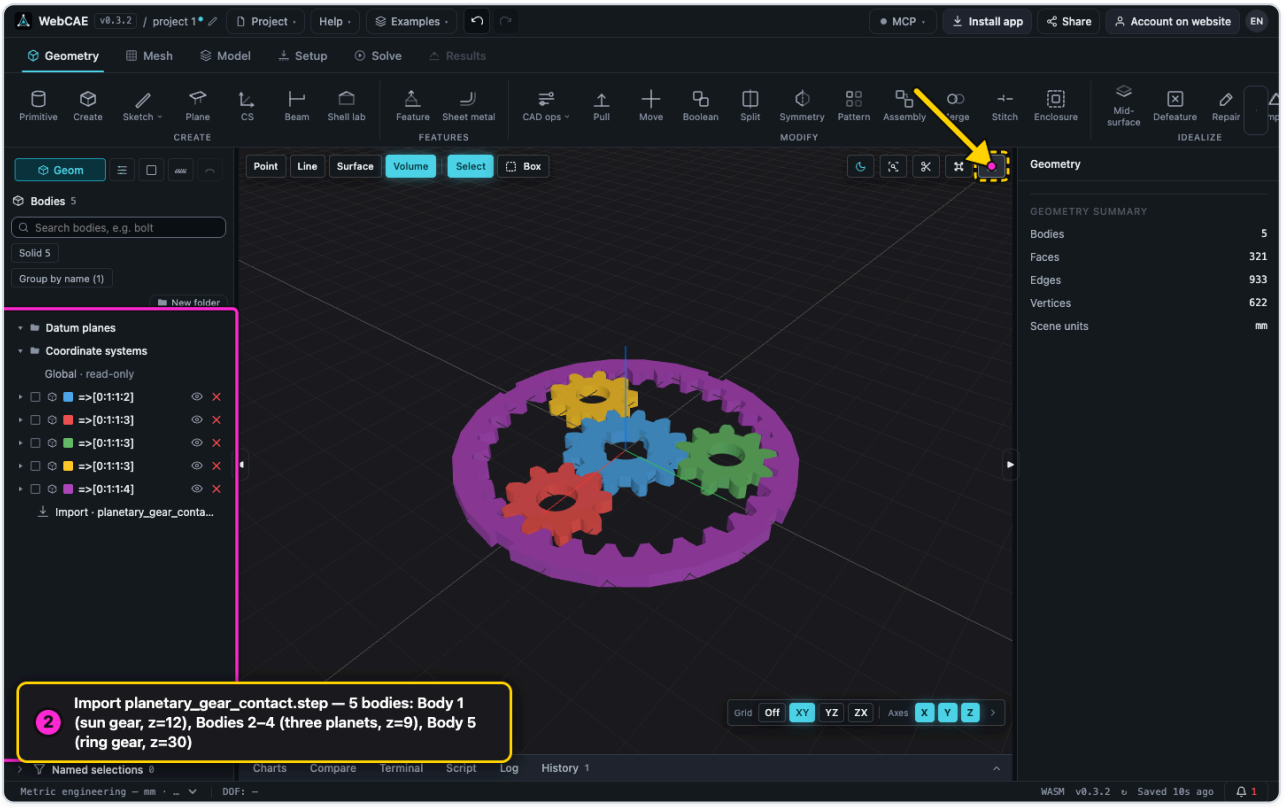
WHY Initializes an empty assembly workspace with entity numbering starting from 1.



STEP 2 Import the 5-body planetary gear STEP assembly

In the **Geometry** tab click **Import** and choose `planetary_gear_contact.step`. The tree lists five solid bodies: **Body 1** (12-tooth sun gear at $(0, 0, 0)$), **Body 2**, **Body 3**, **Body 4** (three 9-tooth planet gears on carrier circle $r_c = 42\ \text{mm}$ at $0°, 120°, 240°$), and **Body 5** (30-tooth internal ring gear). Click **Fit all content to screen**.

WHY All five gears are positioned in simultaneous tooth engagement along both the external sun–planet and internal planet–ring lines of action ($z_r = z_s + 2z_p = 12 + 2 \times 9 = 30$).

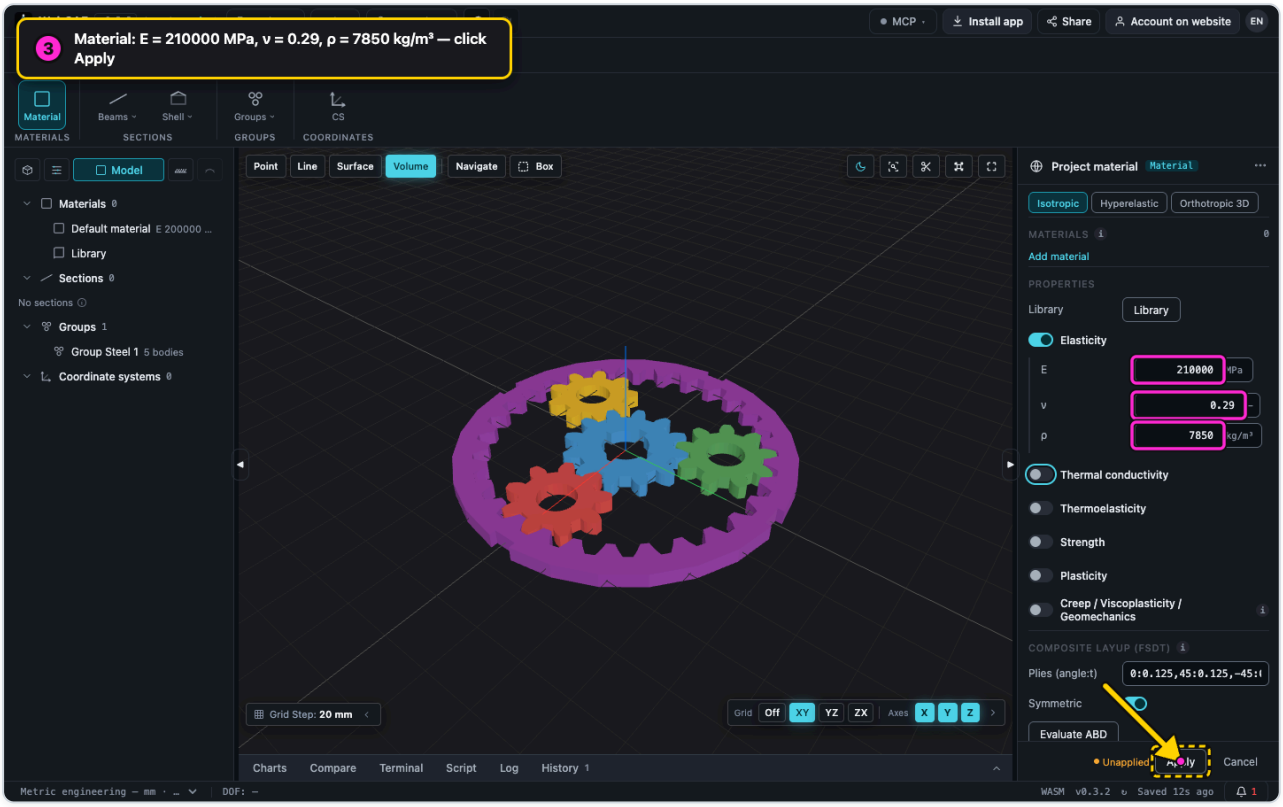


STEP 3

Define the gear steel material

Open the **Model** tab, click **Material**, set $E = 210\,000\ \mathrm{MPa}$, $\nu = 0.29$, and $\rho = 7\,850\ \mathrm{kg/m^3}$, then click **Apply** (the arrow on the frame).

WHY Applies case-hardened gear steel properties across all five bodies of the planetary stage.

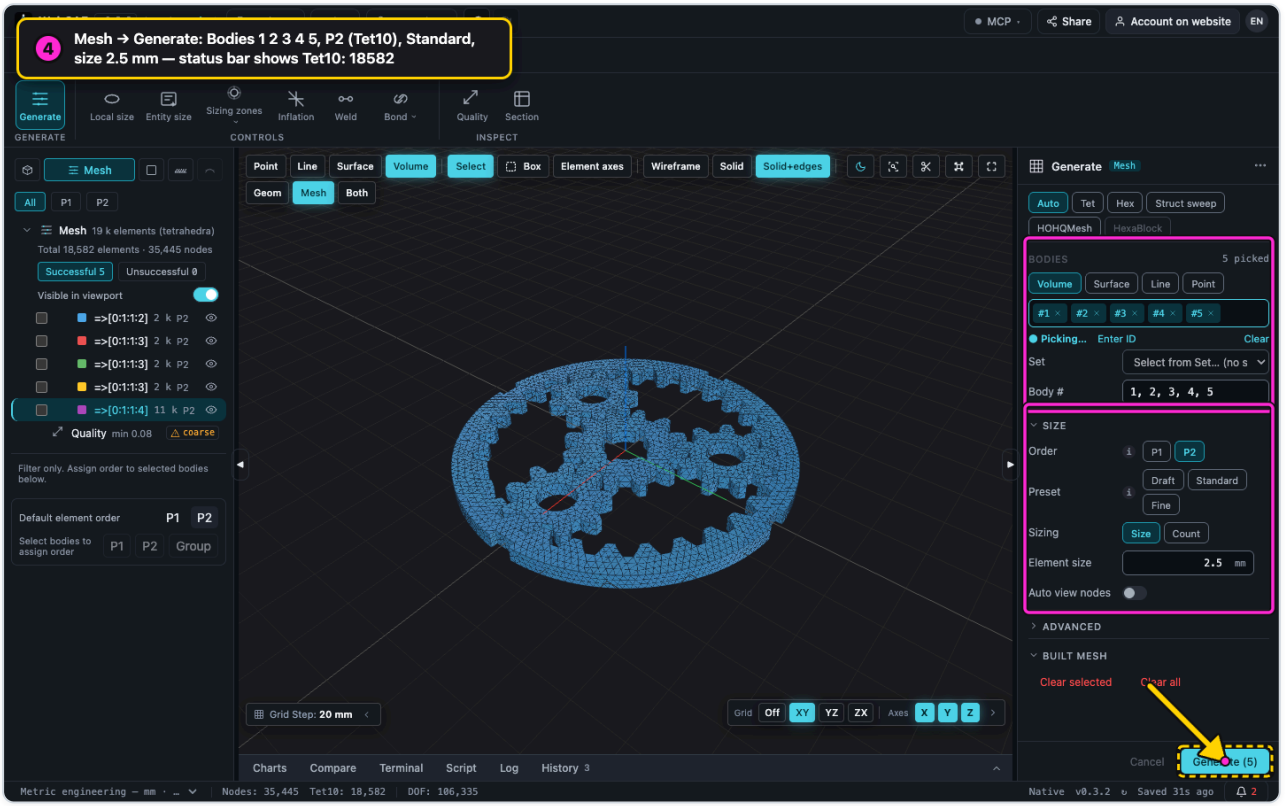


STEP 4

Generate a quadratic (P2 / Tet10) mesh on all 5 bodies

Open the **Mesh** tab, click **Generate**, click **Enter ID**, type `1 2 3 4 5` and press **Enter**. Select element order **P2** (**Tet10**) and set the element size to `2.5` mm, then click **Generate**. The status bar reports **Tet10: 18582** (35445 nodes).

> **WHY** Quadratic tetrahedra resolve both the convex external tooth flanks (sun and planets) and the concave internal tooth flanks (ring gear).

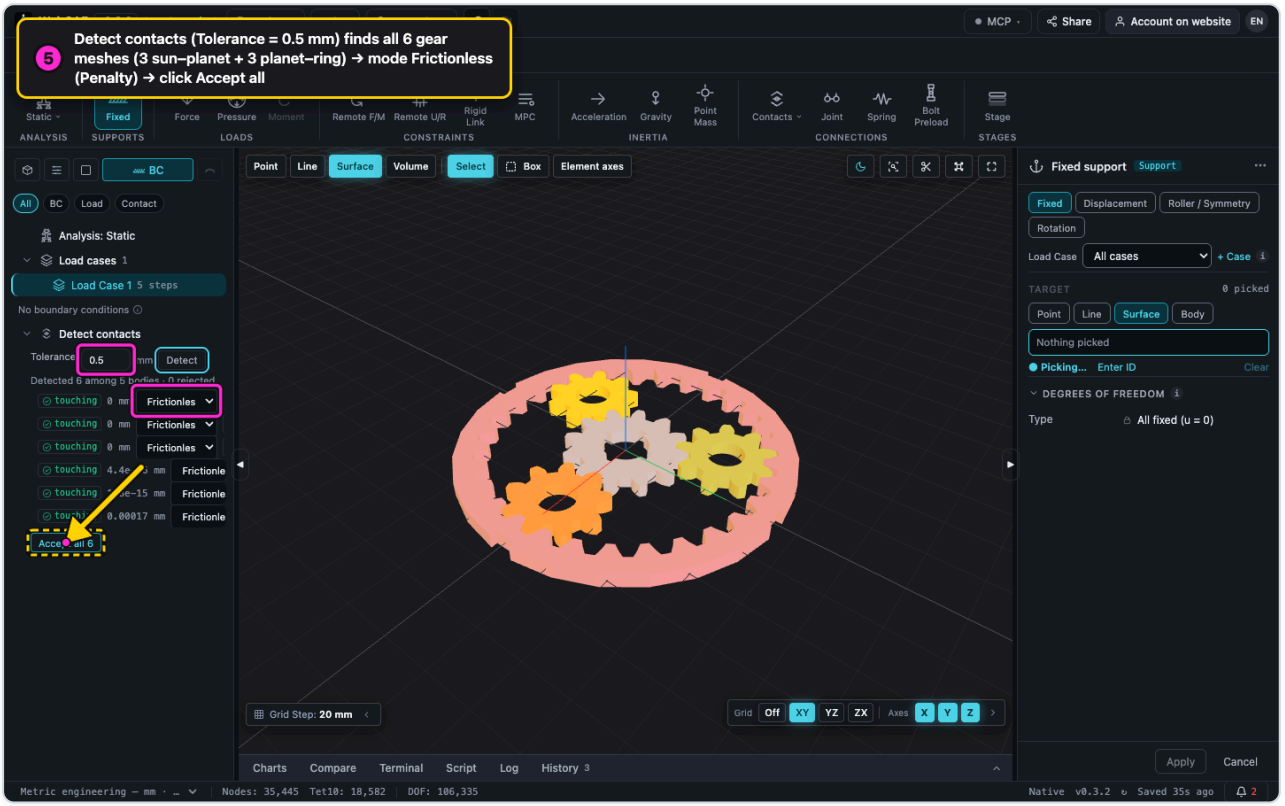


STEP 5

Detect all 6 gear mesh interfaces and set Frictionless (Penalty)

Open the **Setup** tab, click **Detect contacts** in the tree, set **Tolerance (mm)** to `0.5`, and click **Detect**. WebCAE finds all **6** gear mesh interfaces (3 sun–planet pairs + 3 planet–ring pairs). Set every proposal dropdown to **Frictionless (Penalty)** and click **Accept all 6** (the arrow on the frame).

> **WHY** Each planet simultaneously engages the central sun gear and the outer ring gear through unilateral, frictionless involute contact.

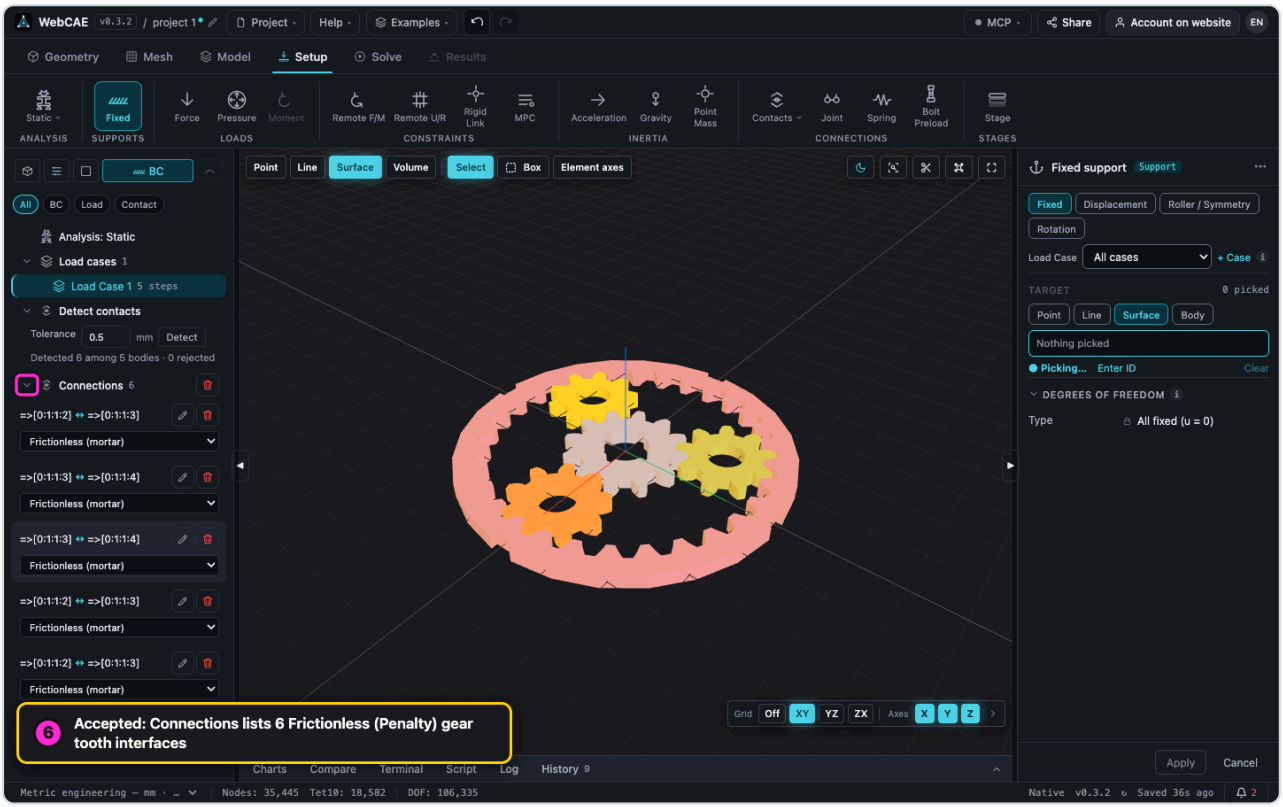


STEP 6

Check the 6 accepted contact pairs in the tree

Under **Connections** in the Setup tree, all 6 **Frictionless (Penalty)** contact pairs (**Body 1 ⇔ Body 2**, **Body 1 ⇔ Body 3**, **Body 1 ⇔ Body 4**, **Body 2 ⇔ Body 5**, **Body 3 ⇔ Body 5**, **Body 4 ⇔ Body 5**) are listed.

> **WHY** Verifies that all three parallel load paths from the sun gear through the planets to the ring gear are active.

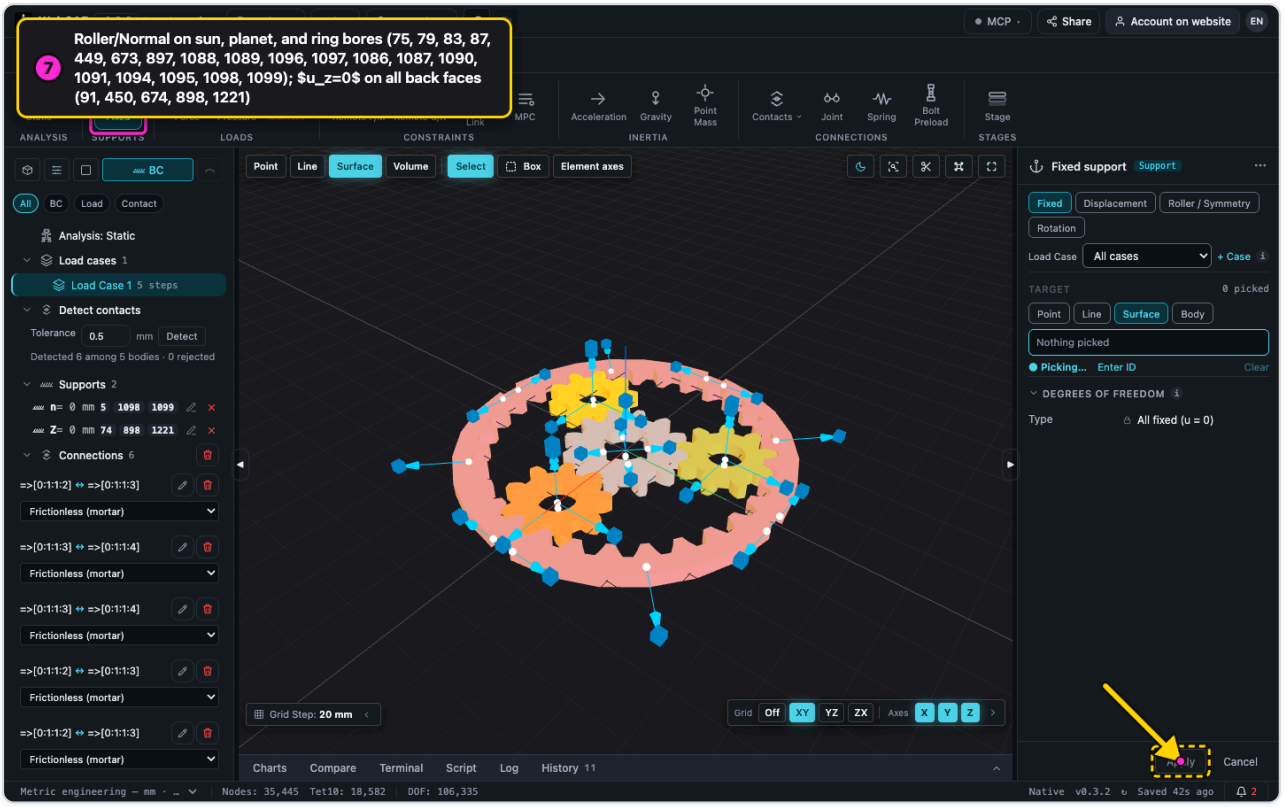


STEP 7

Support all bores, ring housing, and back faces

Choose **Fixed** → **Roller / Normal**, pick sun bore, all three planet bores, ring outer cylinder, and ring keyway stops (`sun_bore` , `planet*_bore` , `ring_outer` , `ring_keyway_stop` in `planetary_gear_contact.tags.json`), and click **Apply**. Choose **Displacement**, lock **X** and **Y**, constrain **Z** on every back face of bodies 1–5, and click **Apply**.

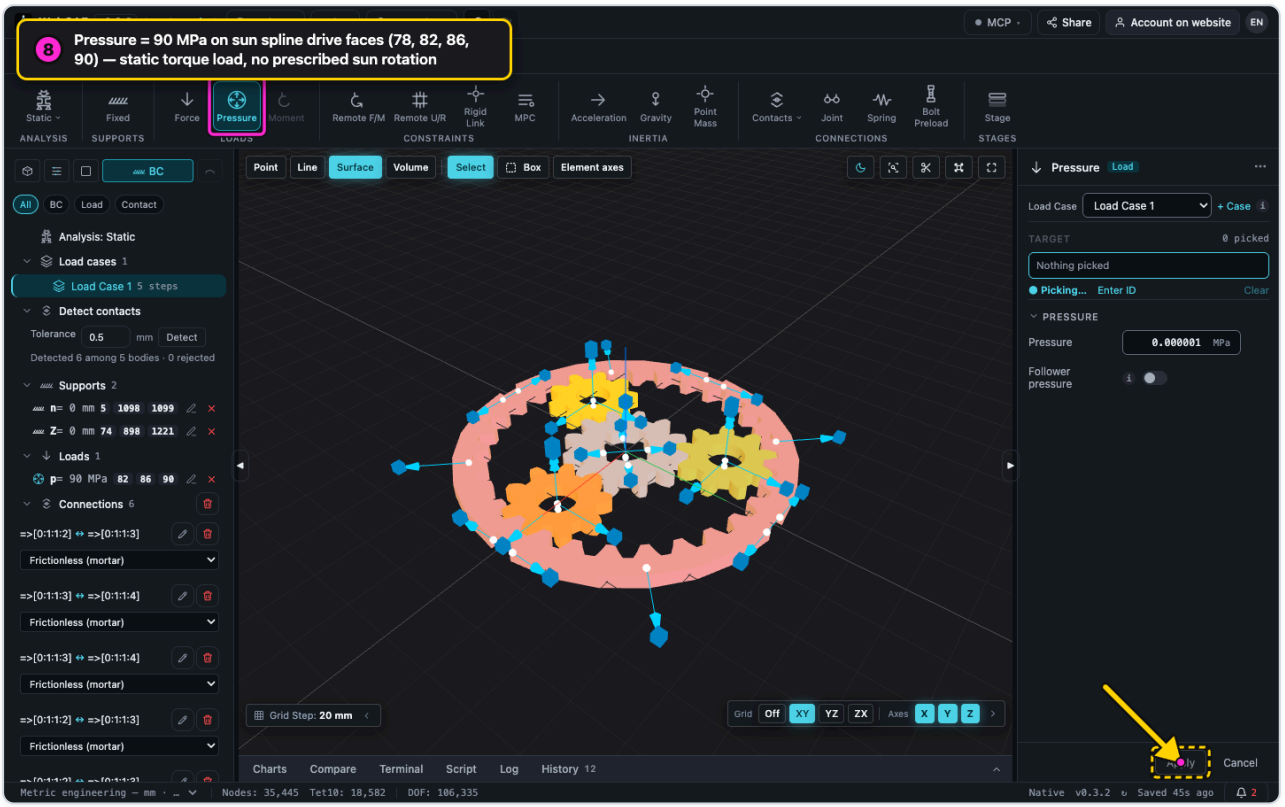
WHY Same support pattern as `planetary_gear_contact/case.mts` : axes held in normal direction, spin about z free, thrust on backs, ring held from outside.



STEP 8 Apply sun driving torque via 90 MPa spline pressure

Click **Pressure**, select the four sun spline drive faces (`sun_keyway_drive`), enter **90** MPa, and click **Apply**.

WHY Loads the stage torsionally without **Remote Motion**. Large prescribed R_z on the sun bore is rejected in contact assemblies and does not roll planets around the carrier.

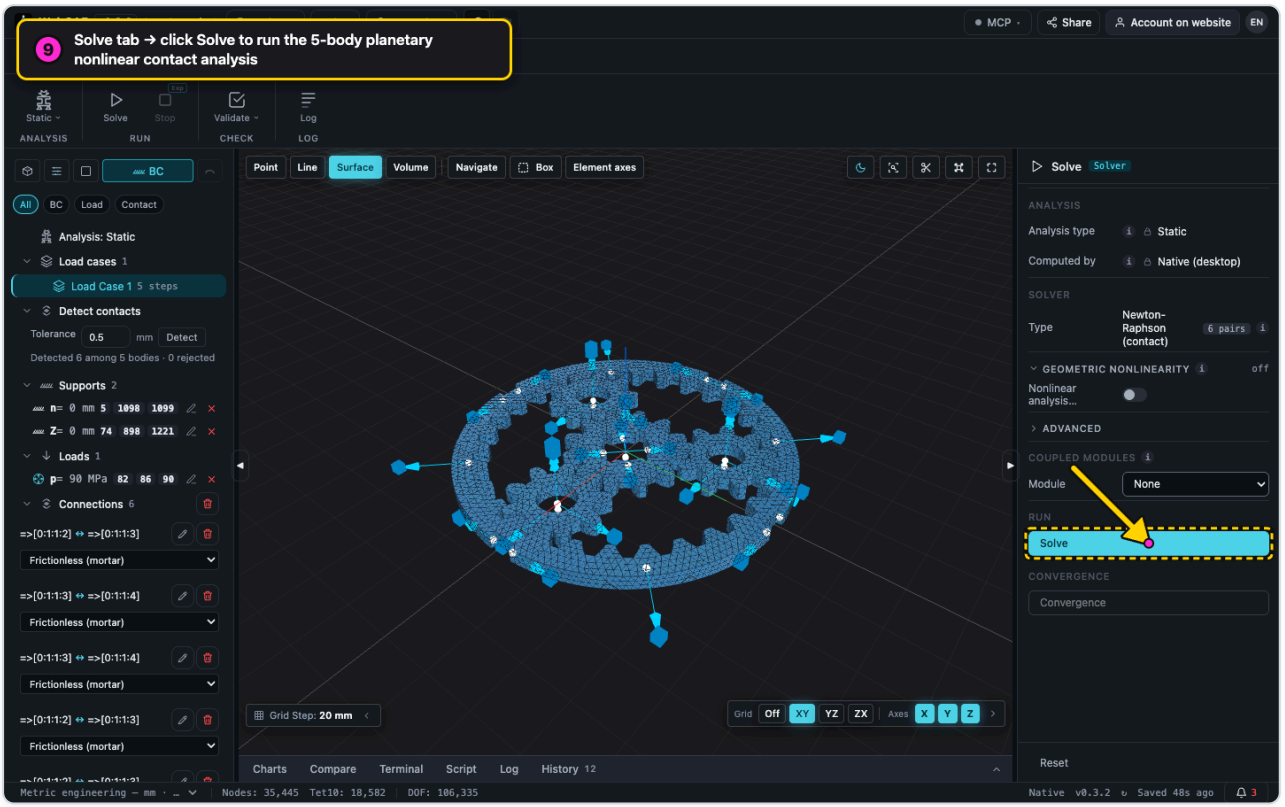


STEP 9

Solve the five-body static contact assembly

Open **Solve** and click **Solve**. The native Mortar path ramps spline pressure and balances all six frictionless interfaces at the imported tooth positions.

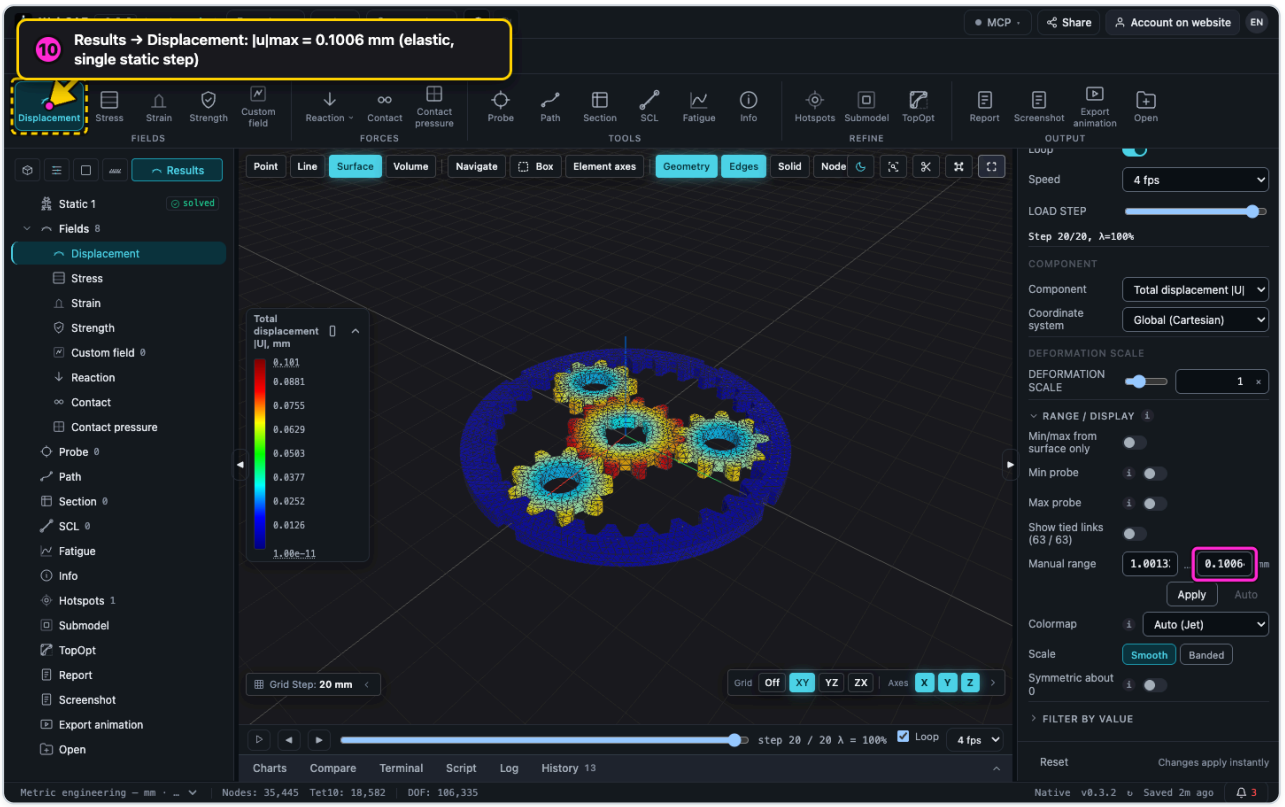
WHY Computes elastic three-path load split in a **fixed** epicyclic meshing pose, not a full 180° carrier orbit.



STEP 10 Inspect elastic displacement $|u|$

In **Results** click **Displacement** with **Deformation scale = 1**. **Manual range** maximum **0.1006 mm** — tooth and body elasticity only.

WHY Confirms small-displacement contact. Orbital motion of planets is not an output of this analysis type.

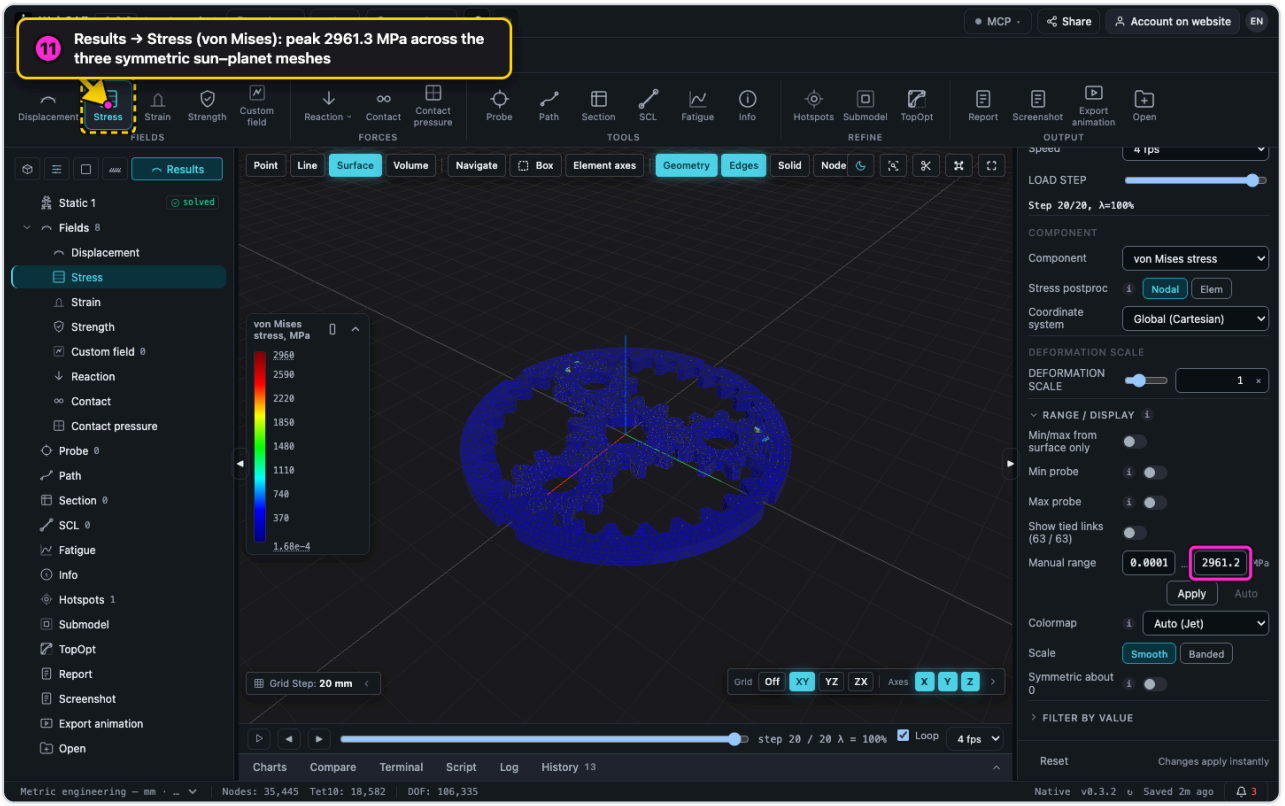


STEP 11

Inspect von Mises stress

Click **Stress**. **Manual range** maximum **2961.3 MPa** (legend includes spline/keyway peaks). Engaged tooth roots are lower ($\approx 1662\ \mathrm{MPa}$ for $r \geq 19\ \mathrm{mm}$ on the browser mesh; native reference $1545.6\ \mathrm{MPa}$).

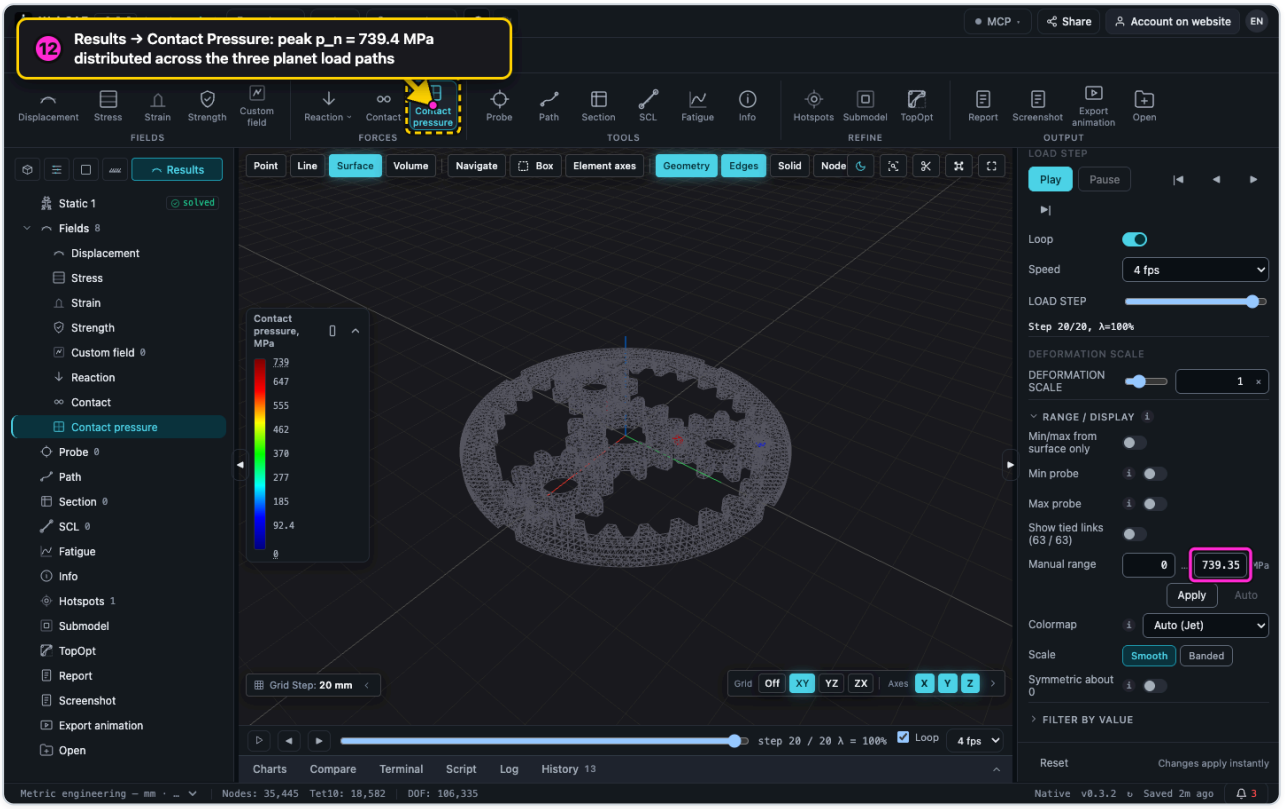
WHY Three symmetric sun–planet and planet–ring paths carry stress; interpret legend max together with tooth-region peaks when comparing to native benchmarks.



STEP 12 Inspect contact pressure across six meshes

Click **Contact Pressure**. **Manual range** maximum **739.4 MPa** on active flanks across the three planet load paths.

WHY Shows unilateral $p_n > 0$ on working flanks only; zero on open backs.



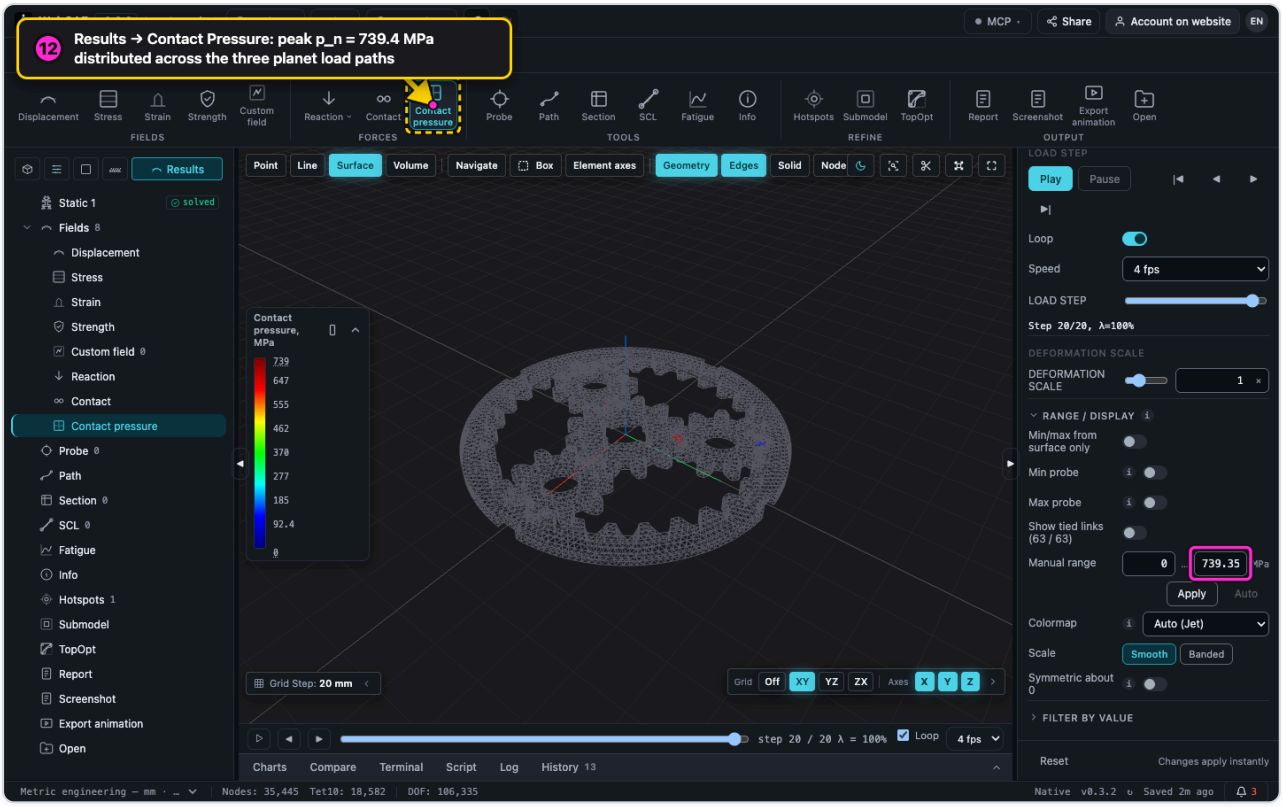
VERIFICATION BENCHMARK

Check against the formula

Quantity	Analytical formula	WebCAE (native `webcae_solve`)	Difference	Where you see it	
Elastic $\lvert u\rvert_{\max}$	$\lvert u\rvert < 1$ mm (static contact)	0.1006 mm	—	Step 10	
Peak contact pressure p_n	Unilateral contact ($p_n > 5$ MPa)	739.4 MPa	—	Step 12	
σ_{vM} (tooth region, $r \geq 19$ mm)	Native `planetary_gear_contact` reference	1545.6 MPa	1662.0 MPa	7.5 %	Post-process filter on browser mesh
σ_{vM} (Results legend)	Includes spline peaks on browser mesh	2961.3 MPa	—	Step 11 Manual range	
Spline torque	90 MPa on `sun_keyway_drive`	≈ 66.46 N · m	—	Setup → Pressure	

FORMULA

Scope note. This lesson uses the verified static BC set from `cases/assemblies/planetary_gear_contact/case.mts` (roller bores, ring normal support, u_z on backs, spline pressure). Willis epicyclic **kinematics** ($\theta_c = \frac{z_s}{z_s+z_r}\theta_{\mathrm{sun}}$ for a free carrier) is **not** reproduced in the FEA mesh: contact pairs stay in the imported engagement position. Compare $\lvert u\rvert$, p_n, and tooth-region σ_{vM} to the native case file, not to chordal tip displacement from a 180° turn.



TROUBLESHOOTING & NOTES

Common mistakes

- **Remote Motion on the sun ($R_z = 180^\circ$):** rejected or misleading for contact assemblies; use spline **Pressure** like the benchmark case.
- **Fully fixing the ring with $u_x = u_y = u_z = 0$ on the entire outer surface:** the benchmark holds the ring with **Normal** on the outer cylinder and keyway stops, not a solid clamp of every face.
- **Omitting a planet from the mesh target list (`1 2 3 4 5`):** breaks the 3-fold load path symmetry.
- **Contact tolerance below 0.5 mm:** six curved interfaces need **0.5 mm** detection tolerance.

RECOMMENDED PROGRESSION

Where to go next

Congratulations on completing the **WebCAE 101** interactive curriculum! Explore the **Case Library** on the Start screen to inspect pre-built industrial assemblies, CFD turbomachinery, and multiphysics benchmarks, or open the **DSL Reference** (`/docs/dsl`) to automate parametric gear and assembly studies from scripts.

WebCAE 101

LESSON 19 · F19

Register the official WebCAE MCP Server in Claude Code CLI (one-time setup):

Goal Connect a local AI coding or engineering agent (**Claude Code**, **Claude Desktop**, **Cursor**, **Codex**, or a custom MCP script) directly to a live **WebCAE** session — either in the browser (**Web/WASM**) or in the **WebCAE Desktop (Native)** app — over the local WebSocket Model Context Protocol bridge (`ws://127.0.0.1:8765`), and run a complete 3D CAD + P2 Tet10 FEA verification loop without manual clicking.

Outcome A live green **MCP: Connected** bridge between your local agent and WebCAE, automatic surface discovery via the unified numbering journal (`surface number 29` and `surface number 30`), a solved 1431-element quadratic `Tet10` cantilever model ($|u|_{\max} = 7.572\ \mathrm{mm}$ vs analytical $7.619\ \mathrm{mm}$, error 0.62%), a structured safety factor verdict returned directly to the agent, and a live 3D WebGL viewport screenshot captured via `cae_screenshot` .

~5 min | web (WASM) | Run: `2026-11-04-32ffaed2`

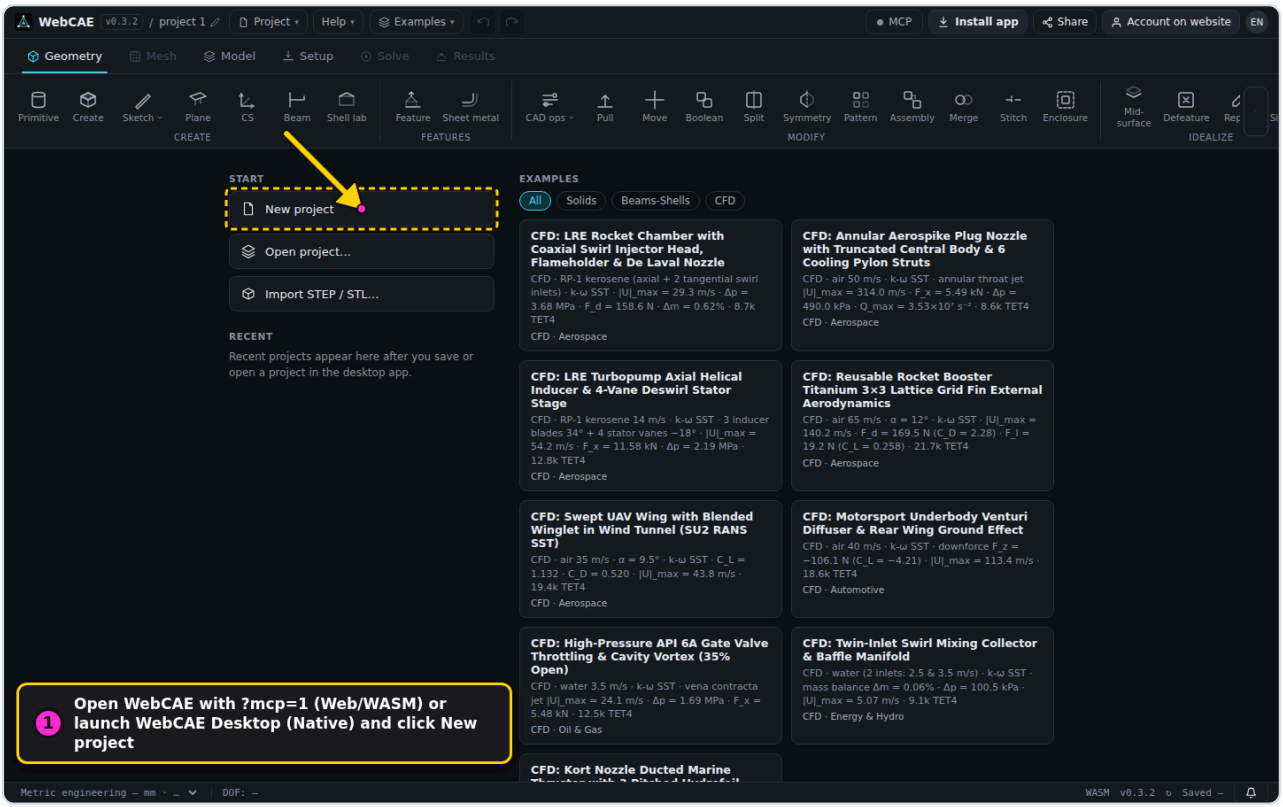
STEP 1

Register the WebCAE MCP Server Once in Your AI Agent

Register the official WebCAE MCP Server once in your local AI client (**Claude Code**, **Cursor**, **Claude Desktop**, or **Codex**). When your AI client starts a session, it automatically spawns the local MCP bridge on `ws://127.0.0.1:8765`. Open **WebCAE** (in the browser or Desktop app) and click **New project** on the start screen.

```
claude mcp add webcae -- npx -y @webcae/mcp-server
```

Why. The local MCP server (`@webcae/mcp-server` / `mcp-bridge/server.mjs`) speaks stdio JSON-RPC to your AI agent on one side and opens a local-only WebSocket hub on `ws://127.0.0.1:8765` on the other side, forwarding every engineering tool call to whichever WebCAE window (Web/WASM or Desktop Native) holds the active slot.



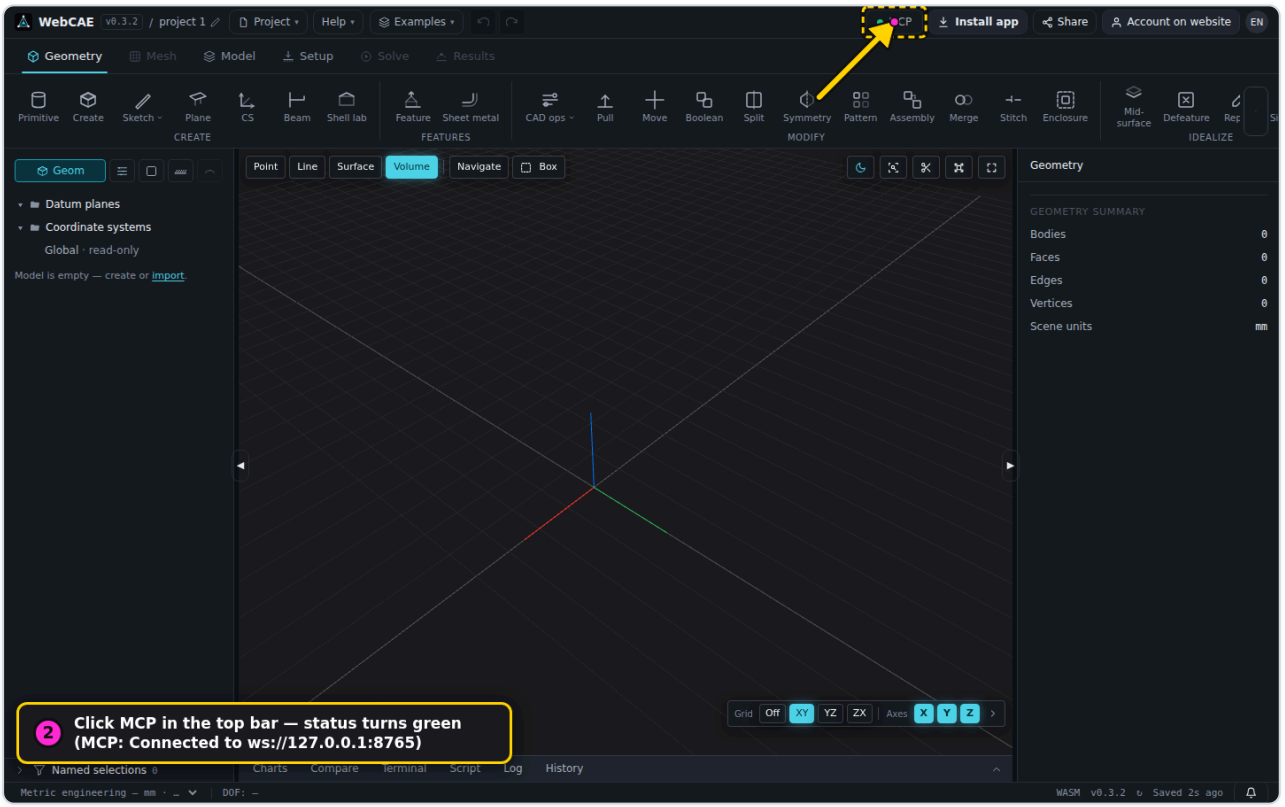
STEP 2

Click the MCP Button in the Top Bar to Connect

Click the **MCP** button in the top bar of WebCAE. WebCAE immediately connects to `ws://127.0.0.1:8765`, the status dot turns green (**MCP: Connected**), and your AI agent sees the live session.

```
{
  "mcpServers": {
    "webcae": {
      "command": "npx",
      "args": ["-y", "@webcae/mcp-server"]
    }
  }
}
```

Why. Clicking **MCP** in the top bar explicitly binds that browser tab or Desktop window to the local port `8765` without needing any URL query parameters. If multiple Claude/Cursor windows are open simultaneously, the first process becomes the **HUB** on port `8765` and additional processes automatically join as **RELAY** clients.



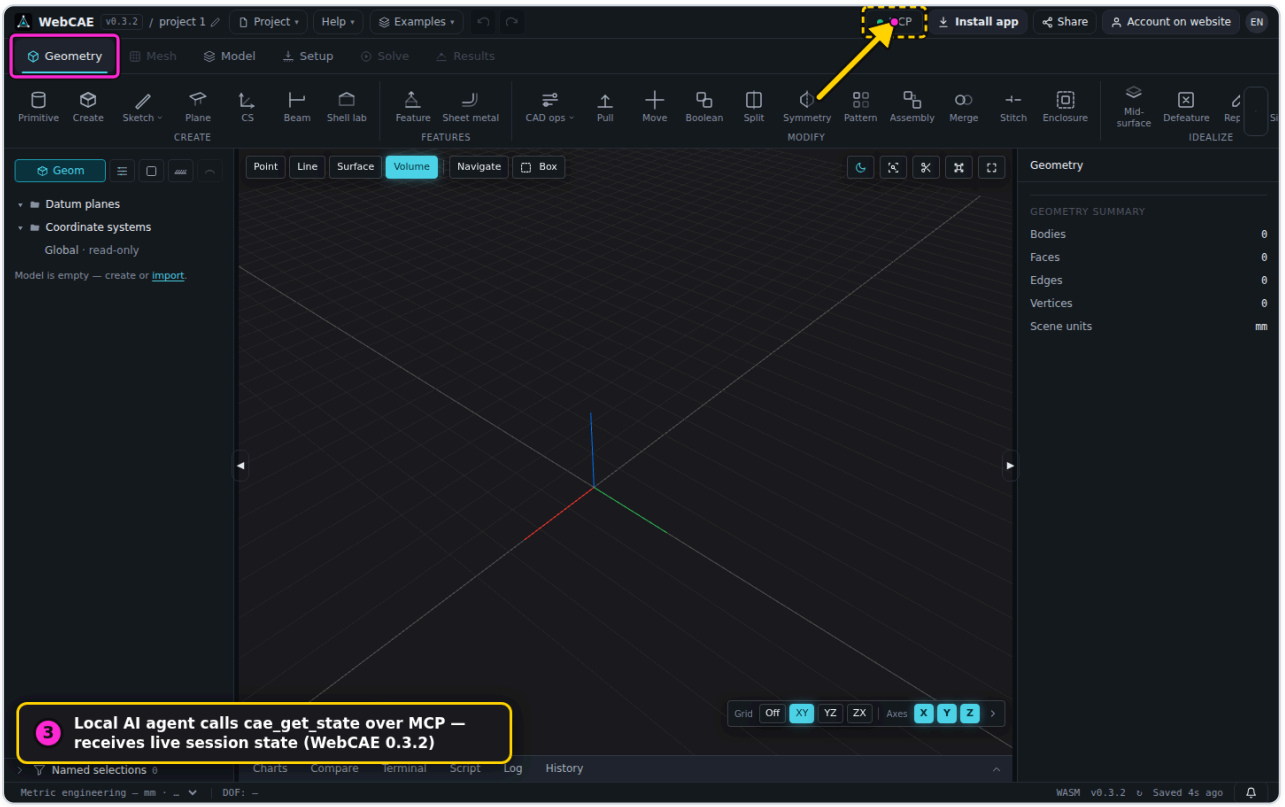
STEP 3

Inspect the Live Session via `cae_get_state`

From your AI agent, call `cae_get_state`. The bridge returns a compact JSON snapshot of the active project, including product version (`WebCAE 0.3.2`), current wizard step, active bodies, mesh status, and result staleness epoch.

```
// Agent MCP tool call:
cae_get_state({})
```

Why. `cae_get_state` strips multi-megabyte binary arrays (`positions`, `tetNodes`, `brepBytes`) and returns structured metadata plus the `resultsEpoch` staleness verdict so the agent never mistakes stale results for a fresh solve.



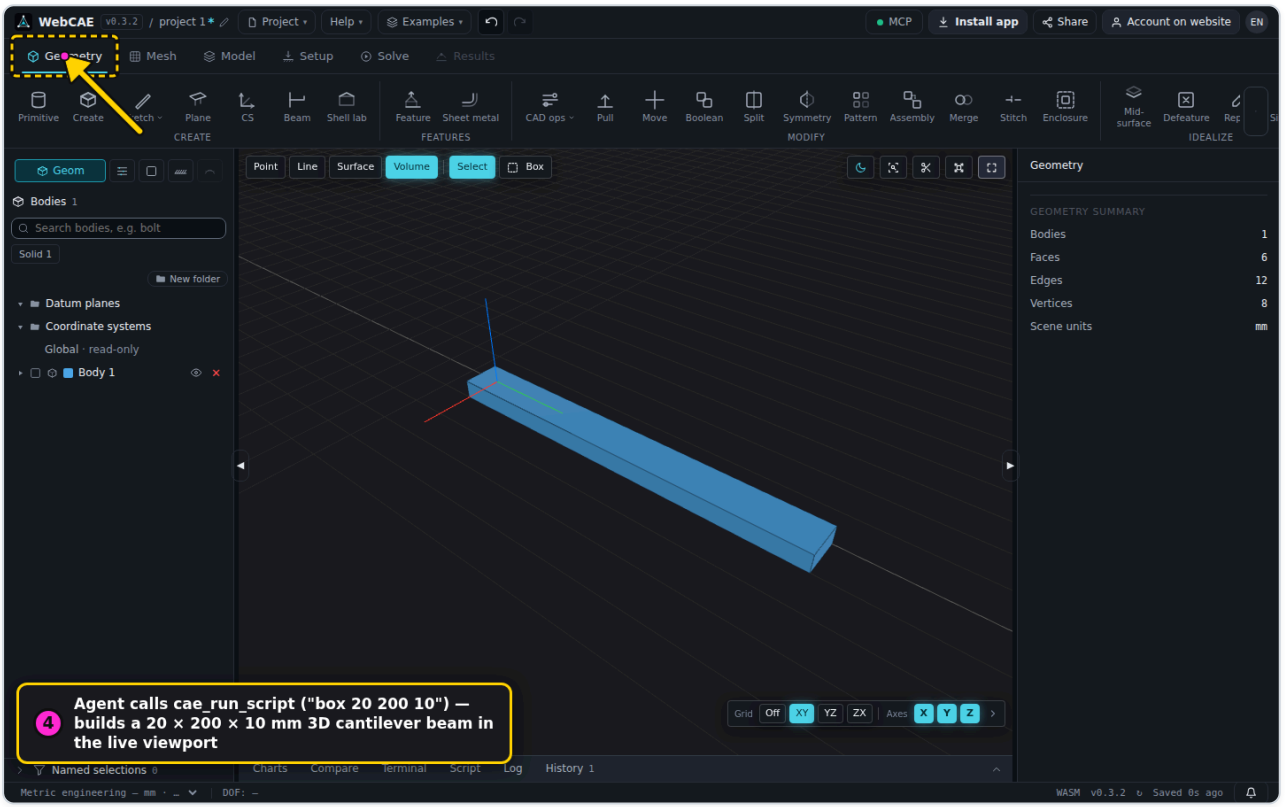
STEP 4

Build 3D Geometry via `cae_run_script`

Ask the agent to create a $20 \times 200 \times 10$ mm 3D cantilever beam using `cae_run_script`. The command executes through the exact same DSL command layer as the built-in script editor and immediately renders the 3D solid in the viewport.

```
box 20 200 10
```

Why. `mcpBridgeConnector.ts` routes `cae_run_script` through `window.cae.runScriptText()`, preserving undo/redo history, geometry artifacts, and the unified entity numbering journal.



STEP 5

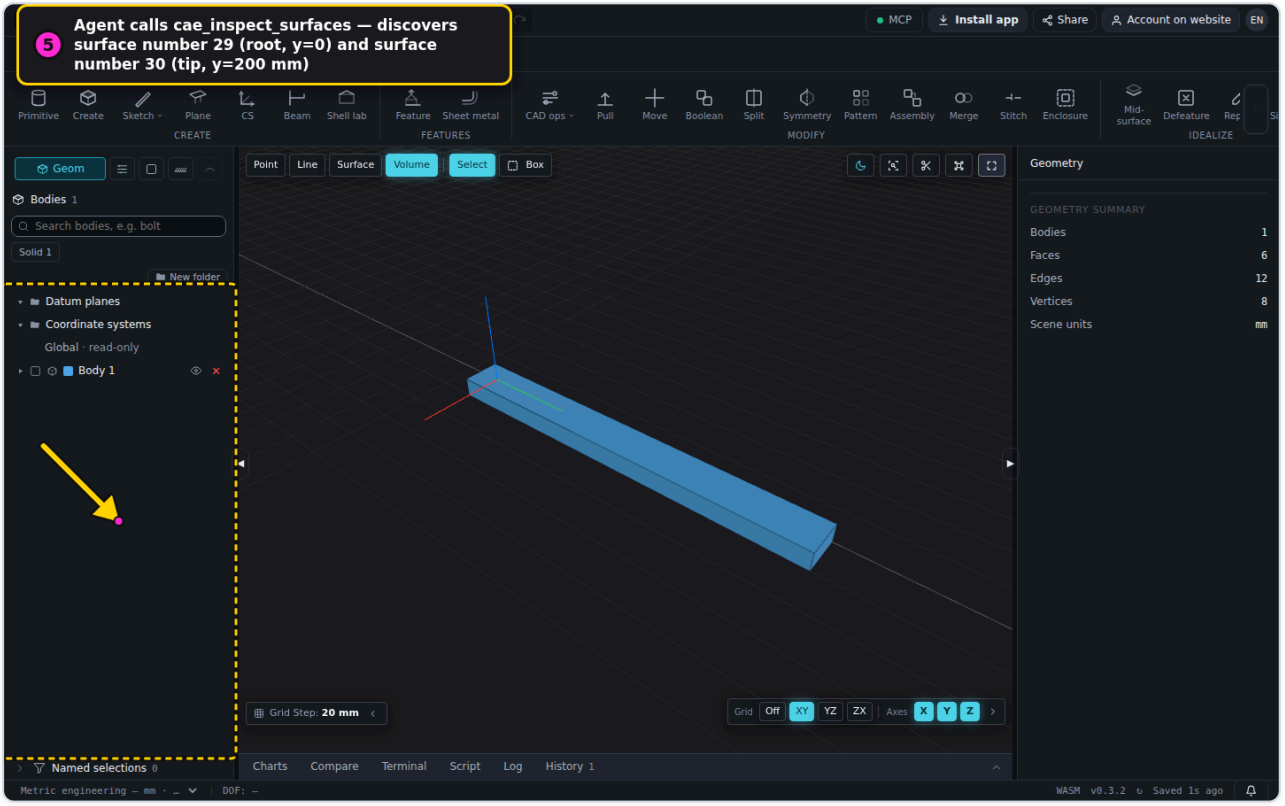
Query the Unified Numbering Journal via `cae_inspect_surfaces`

Before applying boundary conditions, the agent calls `cae_inspect_surfaces` and `cae_inspect_edges`. WebCAE returns all 6 faces and 12 edges with their exact `surfaceNumber`, outward normal vector, centroid, and area in mm^2:

- Clamped root face ($y = 0$ mm, normal $[0, -1, 0]$, area $200\ \mathrm{mm}^2$): `surface number 29`
- Loaded tip face ($y = 200$ mm, normal $[0, +1, 0]$, area $200\ \mathrm{mm}^2$): `surface number 30`

```
// Agent MCP tool call:
cae_inspect_surfaces({})
// -> root: surface number 29, tip: surface number 30
```

Why. In WebCAE, directional or bounding-box guesses are strictly forbidden. Querying `cae_inspect_surfaces` gives the agent the authoritative `surface number N` from the model tree journal.

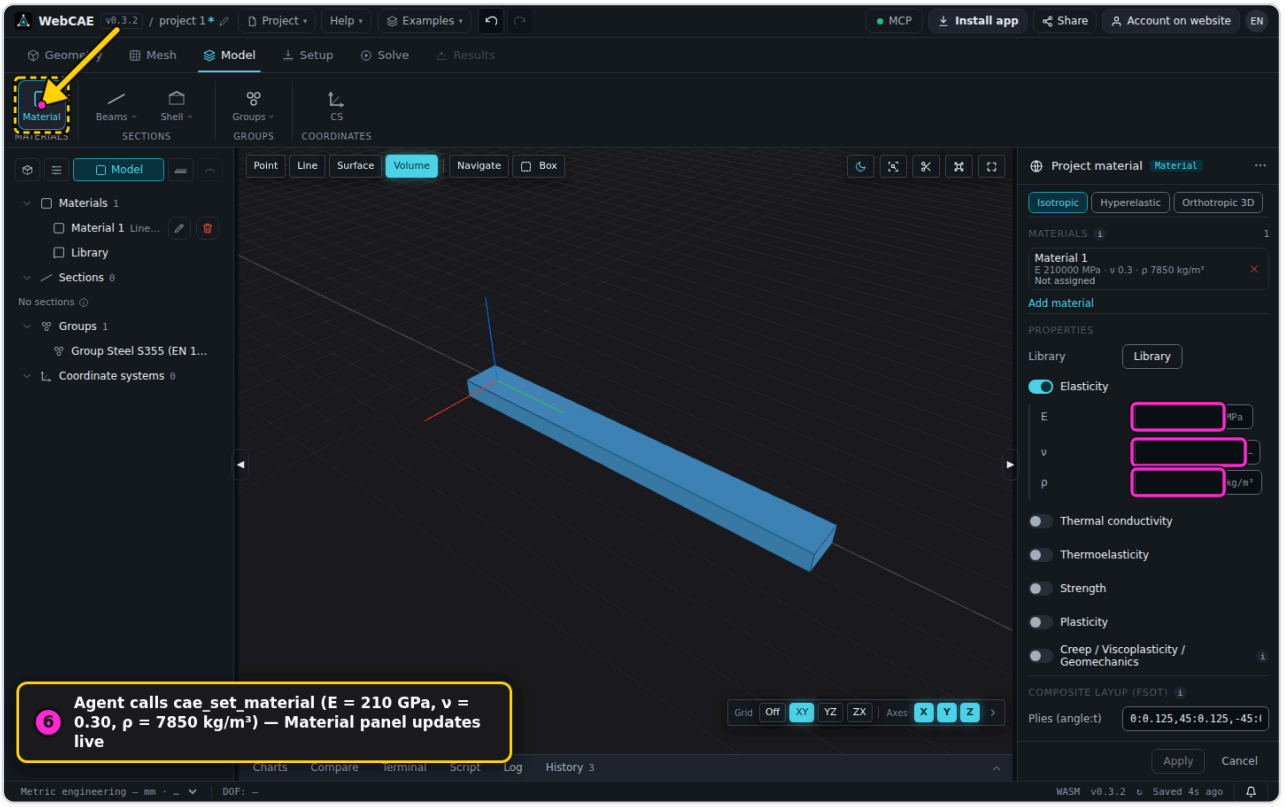


STEP 6 Assign Material Properties via `cae_set_material`

The agent calls `cae_set_material` with Structural Steel parameters ($E = 210\ \text{GPa}$, $\nu = 0.30$, $\rho = 7850\ \text{kg/m}^3$). Switching to the **Material** tab confirms the live update.

```
material E=210000 nu=0.30 density=7850
```

Why. `cae_set_material` validates physical bounds ($E > 0$, $-1 < \nu < 0.5$) and updates the project material state synchronously.



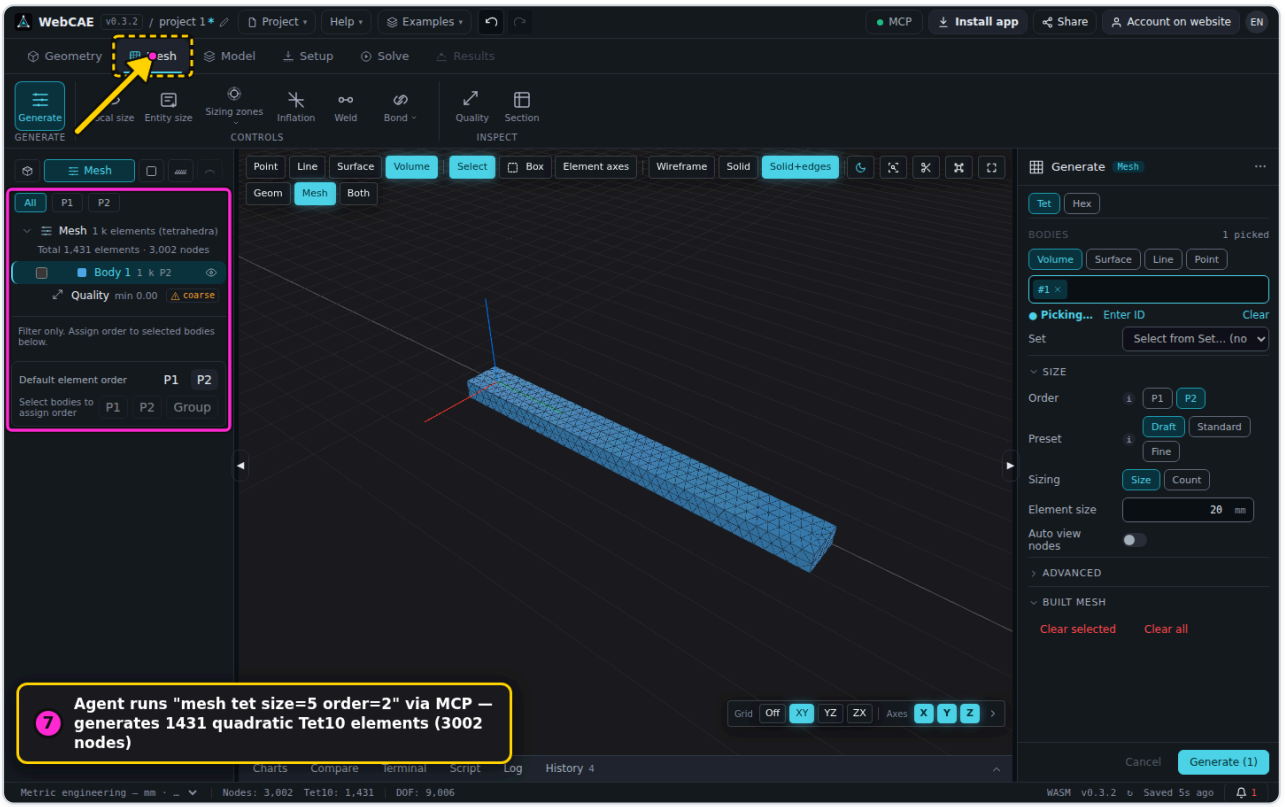
STEP 7

Generate a Quadratic P2 (Tet10) Mesh via MCP

The agent invokes `cae_run_script` with `mesh tet size=5 order=2` to mesh the beam with 10-node quadratic tetrahedra. The status bar reports **Tet10: 1431** (3002 nodes).

```
mesh tet size=5 order=2
```

Why. Quadratic `P2 (Tet10)` elements represent linear bending strain exactly across the $10\ \mathrm{mm}$ thickness without shear locking, whether solved in the browser via WebAssembly or in the desktop app via native OpenMP C++.



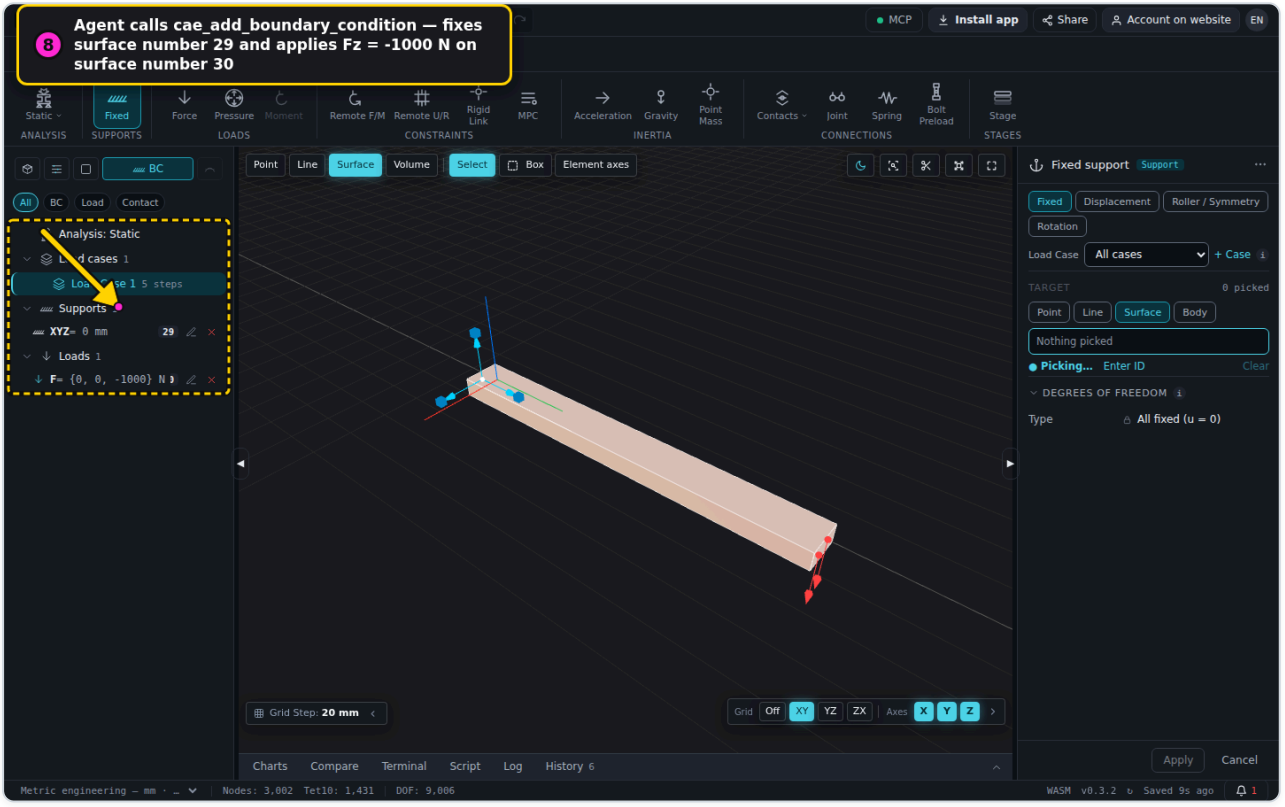
STEP 8 Apply Boundary Conditions on `surface number N` via `cae_add_boundary_condition`

Using the surface numbers discovered in Step 5, the agent calls `cae_add_boundary_condition` twice:

1. Full clamping (`bcType: "fix"`, `component: "all"`) on `surface number 29`
2. Transverse tip force $F_z = -1000\ \mathrm{N}$ (`bcType: "force"`, `fz: -1000`) on `surface number 30`

```
fix surface number 29
load surface number 30 force=-1000 component=z
```

Why. `resolveMcpNumberedTargets` in `mcpBridgeConnector.ts` resolves `uinum:surface:N` through the active `uiNumbering` journal and rejects any unnumbered or mismatched entity kind before solve.



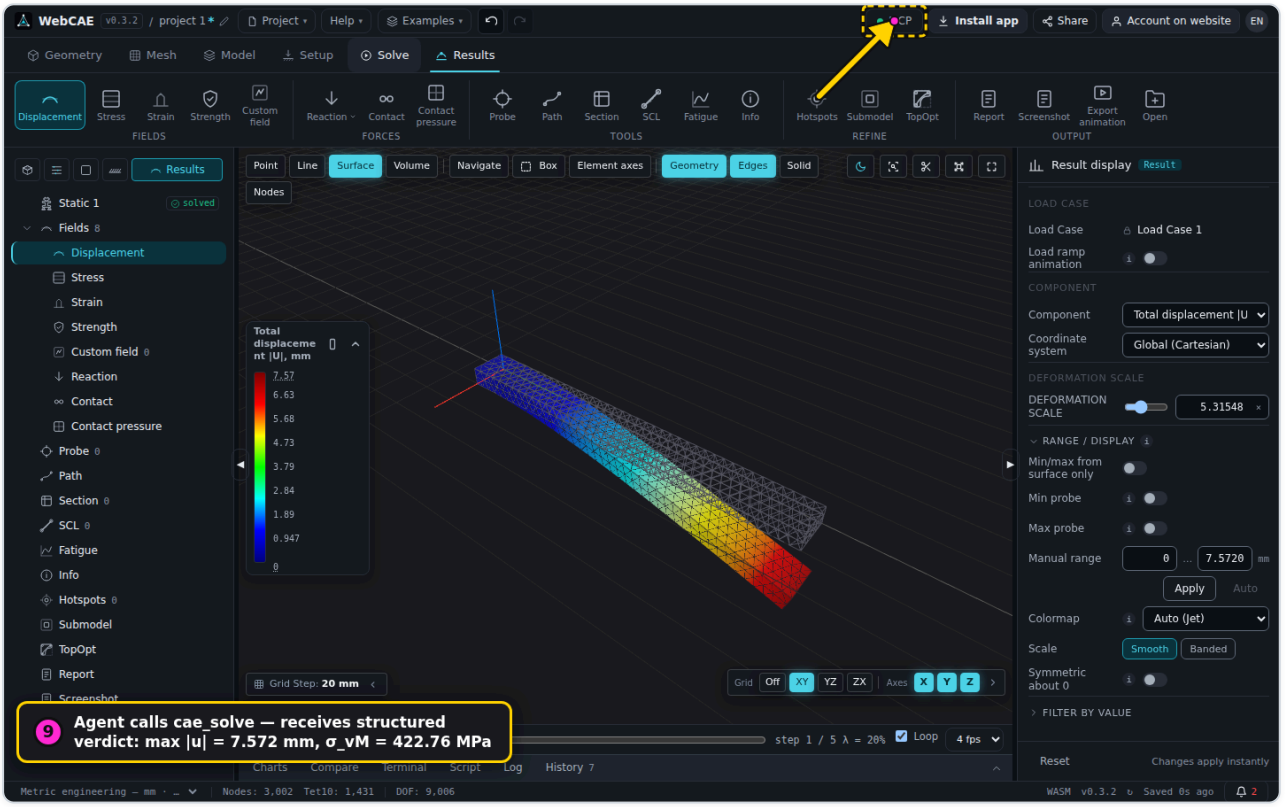
STEP 9 Run the Solver via `cae_solve` and Receive the Engineering Verdict

The agent calls `cae_solve`. WebCAE executes the finite element solver (WASM in browser or native `webcae_solve` in Desktop) and returns a structured engineering summary directly to the agent:

- `maxDisplacementMm`: **7.572 mm**
- `maxVonMisesStressMpa`: **422.76 MPa**
- `minSafetyFactor`: **0.59**

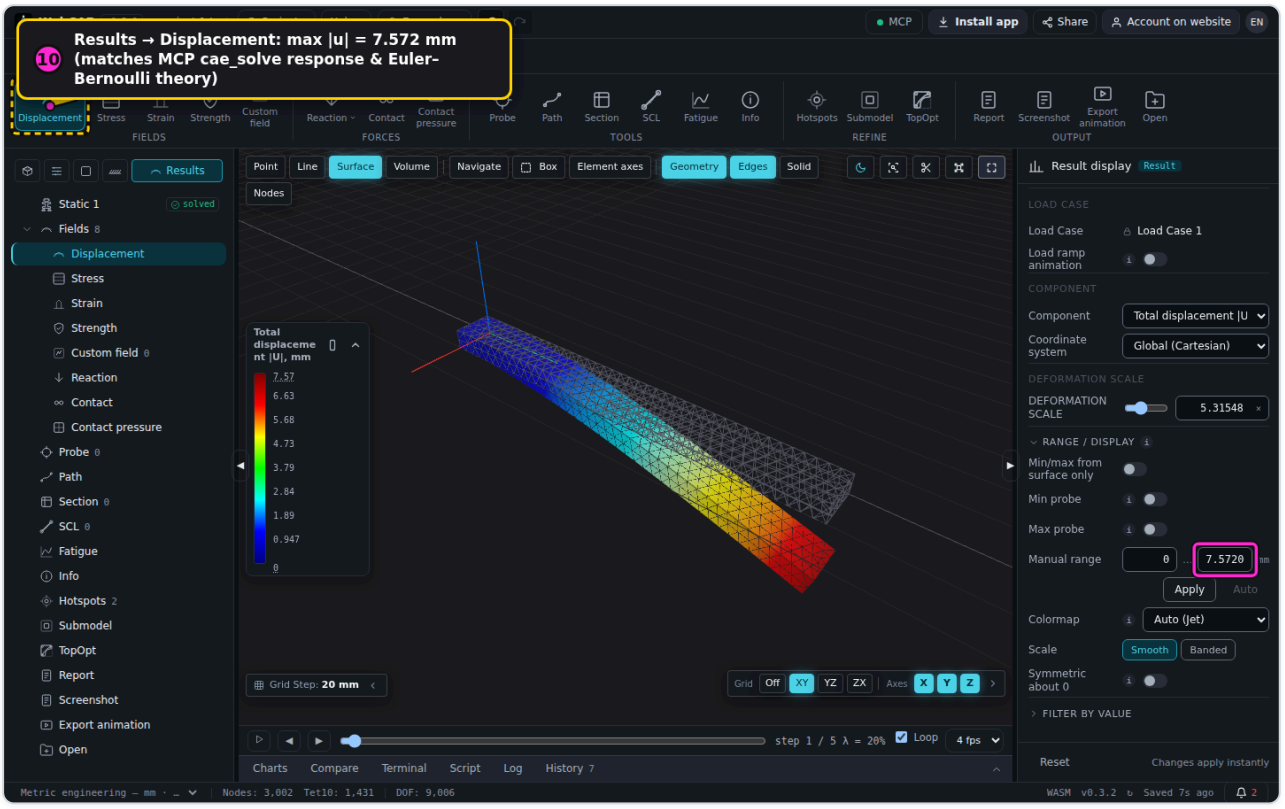
```
solve
```

Why. Returning a structured JSON verdict allows the AI agent to autonomously loop on design dimensions (for example, increasing beam thickness h until $\mathrm{FOS} \geq 1.5$) without scraping HTML.



STEP 10 Verify Tip Deflection $|u|$ in the Results Tab

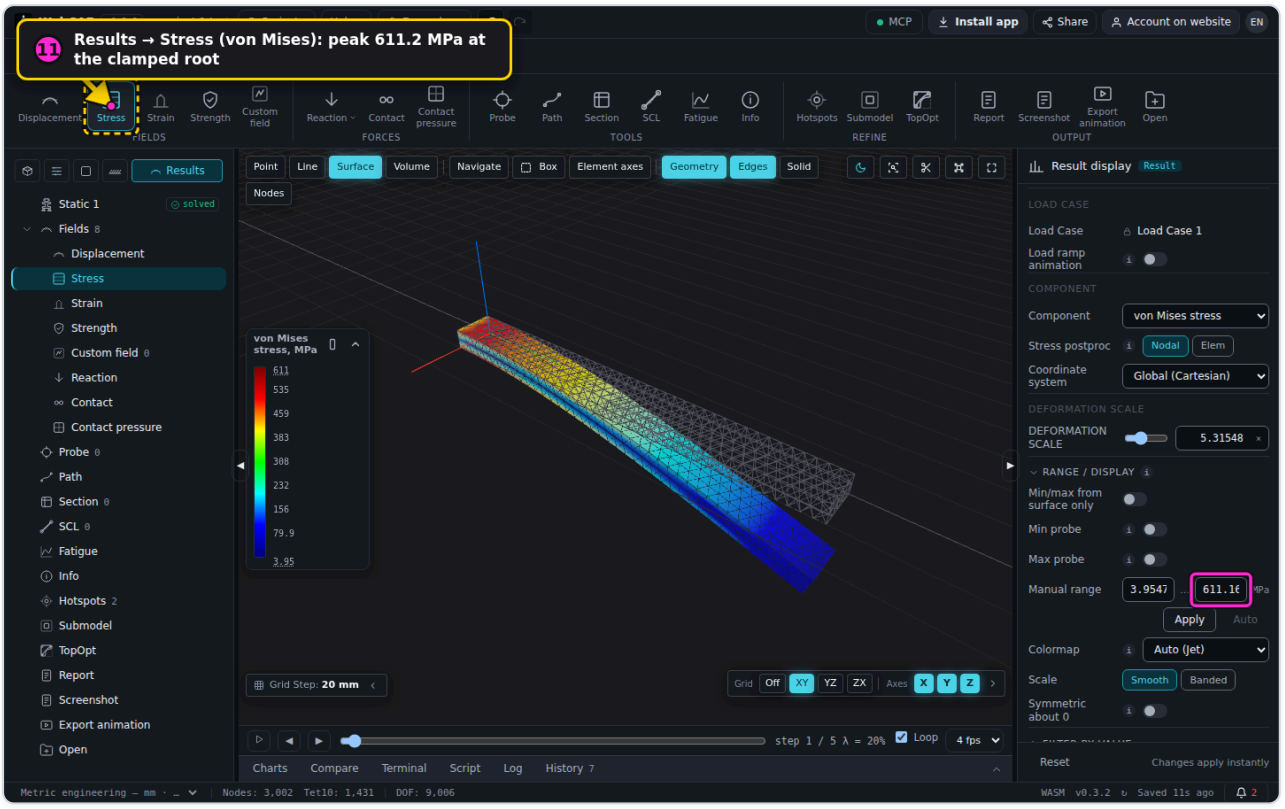
Open **Results → Displacement**. The 3D color contour and **CanvasLegend** display a peak tip displacement of $|u|_{\max} = 7.572\ \mathrm{mm}$, matching the JSON value returned to the MCP agent.



STEP 11

Verify Root Bending Stress σ_{vM}

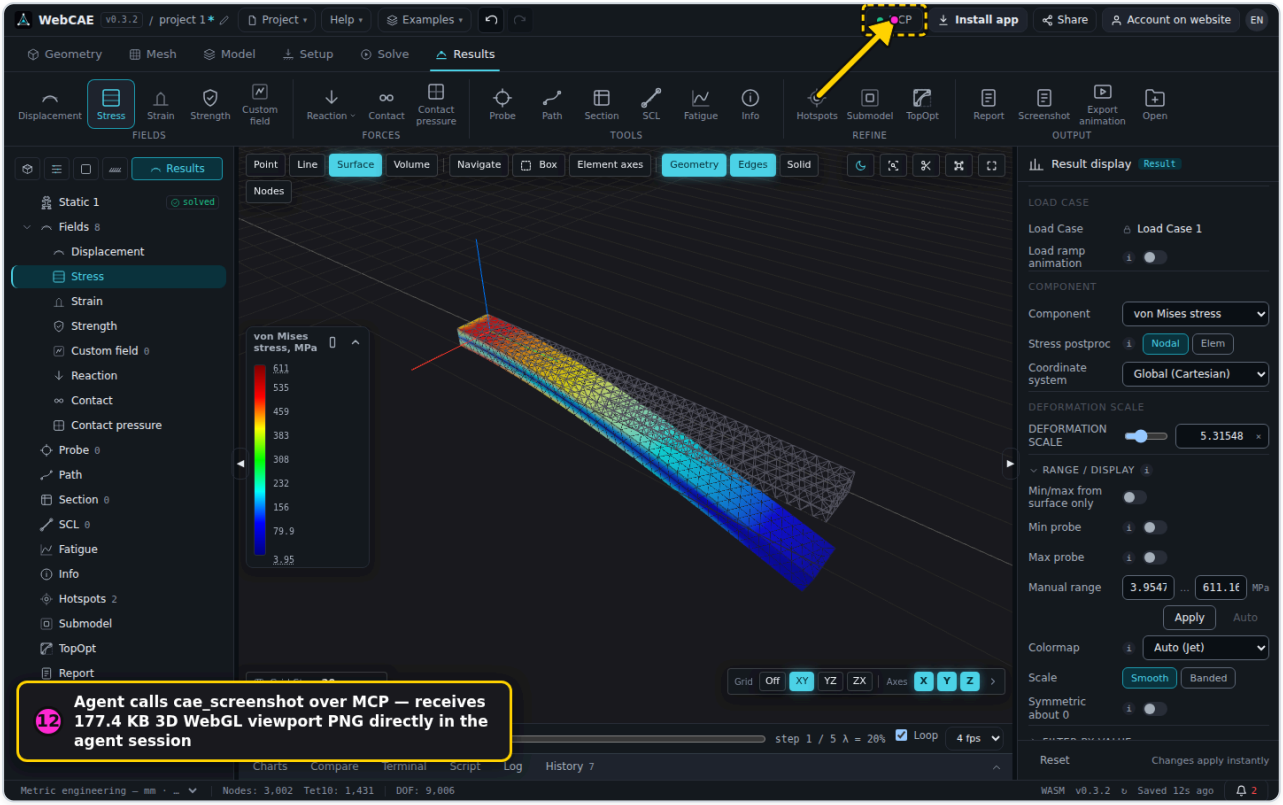
Click **Stress** in the Results ribbon. The peak von Mises stress at the clamped root ($y = 0$ mm) is $\sigma_{\mathrm{vM}} = 611.2$ MPa, in close agreement with Euler–Bernoulli bending theory ($\sigma = \frac{6FL}{bh^2} = 600.0$ MPa).



STEP 12

Capture the 3D Viewport Image via `cae_screenshot`

Finally, the agent calls `cae_screenshot`. WebCAE captures the live WebGL canvas and returns a base64 PNG (`177.4 KB`) so vision-capable agents (Claude Sonnet / Opus, GPT-4o, Gemini Pro) can visually inspect mesh quality, boundary condition arrows, and stress contours.



VERIFICATION BENCHMARK

Check against Euler–Bernoulli Beam Theory & MCP Synchronization

For a rectangular cantilever of width $b = 20\ \text{mm}$, thickness $h = 10\ \text{mm}$, length $L = 200\ \text{mm}$, Young's modulus $E = 210\,000\ \text{MPa}$, and tip load $F = 1000\ \text{N}$:

$$I = \frac{bh^3}{12} = 1666.67\ \text{mm}^4, \qquad w_{\text{EB}} = \frac{FL^3}{3EI} = 7.619\ \text{mm}, \qquad \sigma_{\text{EB}} = \frac{6FL}{bh^2} = 600.0\ \text{MPa}$$

Quantity	Reference / Analytical	WebCAE via MCP (WASM/Native)	Error
Tip deflection $\|u\|_{\max}$	$7.619\ \text{mm}$	$7.572\ \text{mm}$	$0.62\ \%$
Root bending stress σ_{vM}	$600.0\ \text{MPa}$	$611.2\ \text{MPa}$	$1.86\ \%$
MCP `cae_solve` vs UI Legend $\|u\|_{\max}$	$7.572\ \text{mm}$	$7.572\ \text{mm}$	$0.00\ \%$

TROUBLESHOOTING & NOTES

Common mistakes

- **Forgetting to click the MCP button in the top bar.** In the browser (Web/WASM), WebCAE does not auto-connect on page load so background tabs do not steal the active slot. Click **MCP** in the top bar so the dot turns green (**MCP: Connected**).
- **Guessing face names instead of calling `cae_inspect_surfaces`.** Directional aliases are rejected by the MCP validator. Always call `cae_inspect_surfaces` first and reference `surface number N` from the unified numbering journal.
- **Multiple windows competing for the MCP slot.** When a second WebCAE window clicks **MCP**, the previous window transitions to **MCP: Standby** (amber dot) so commands always go deterministically to the active window.

RECOMMENDED PROGRESSION

Where to go next

- Combine `cae_import_step_artifact` and `cae_inspect_surfaces` to let your AI agent run automated defeaturing and nonlinear contact checks on multi-body STEP assemblies (**Lessons #17 and #18**).

WebCAE 101

LESSON 20 · F20

ISO 7206-4 Total Hip Arthroplasty Implant (Ti-6Al-4V + CoCrMo)

Goal Import a two-body orthopedic total hip arthroplasty (THA) implant assembly (Body 1: Ti-6Al-4V ELI femoral stem with distal potted shaft and 135° CCD neck angle; Body 2: CoCrMo modular femoral head), mesh both components with quadratic tetrahedra (**P2 / Tet10**), detect and assign **Bonded (Exact MPC)** contact across the modular Morse taper interface, clamp the potted stem in PMMA bone cement up to $z = 50\ \text{mm}$ per ISO 7206-4, apply peak gait axial loading $F_z = -2\,300\ \text{N}$ via an **RBE2 Spider (Remote Load)** at the femoral head rotation center, and verify global equilibrium, stress distribution, and 10^7-cycle fatigue endurance per ISO 7206-4.

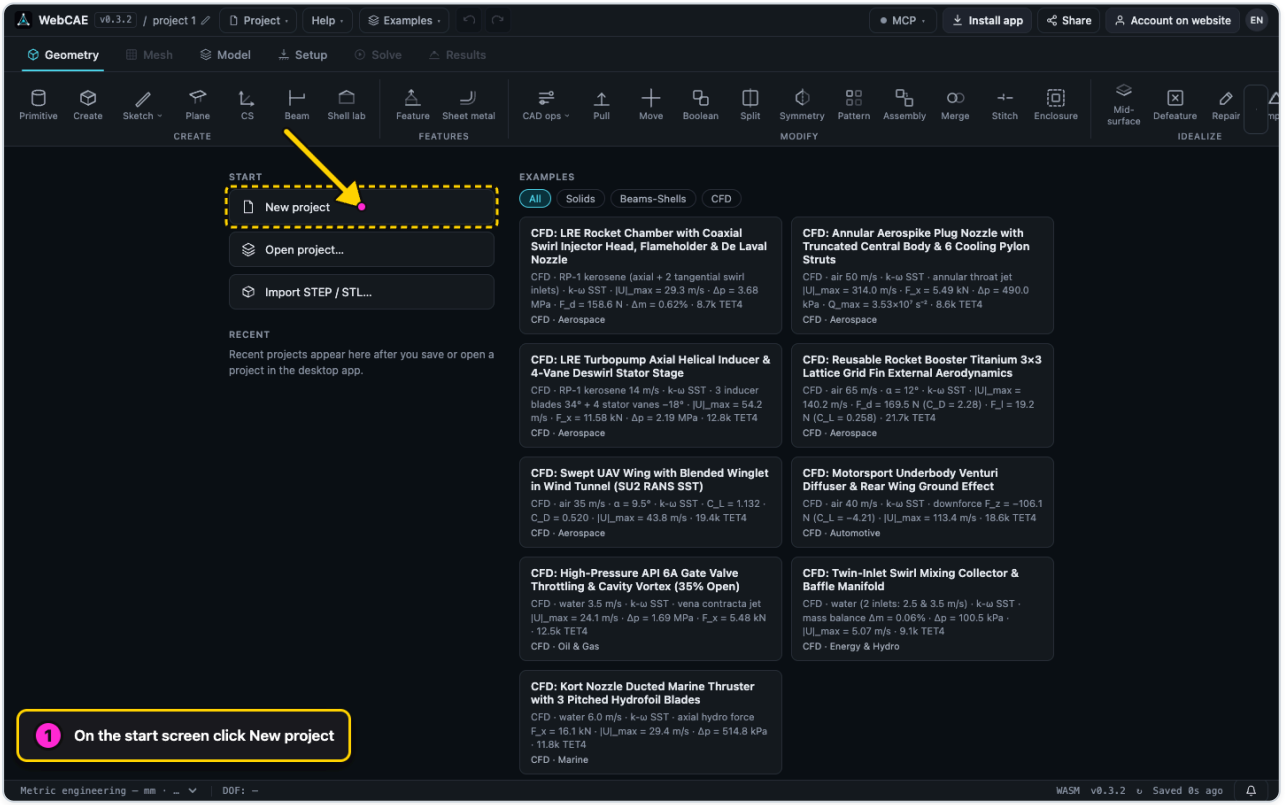
Outcome Total axial potting reaction force $R_z = 2300.0\ \text{N}$ perfectly balancing applied load $F_z = 2300.0\ \text{N}$ (0.00 % difference), applied cantilever bending moment $M_y = F_z \cdot e_x = 65.05\ \text{N} \cdot \text{m}$, peak head displacement $|\mathbf{u}| = 0.1941\ \text{mm}$, peak stem von Mises stress $\sigma_{\text{vM}} = 180.7\ \text{MPa}$, and fatigue endurance safety factor $n_f = 2.77 \geq 1.50$ exceeding ISO 7206-4 requirements for 10^7 load cycles.

6–7 min | WebCAE native structural solver (`webcae_solve`) | Run: 2026-11-04-32ffaed2 / F20

STEP 1 Start a new project

Open WebCAE and click **New project** on the Start screen (the arrow on the frame).

WHY Initializes a fresh document workspace and resets the unified entity numbering journal to 1.

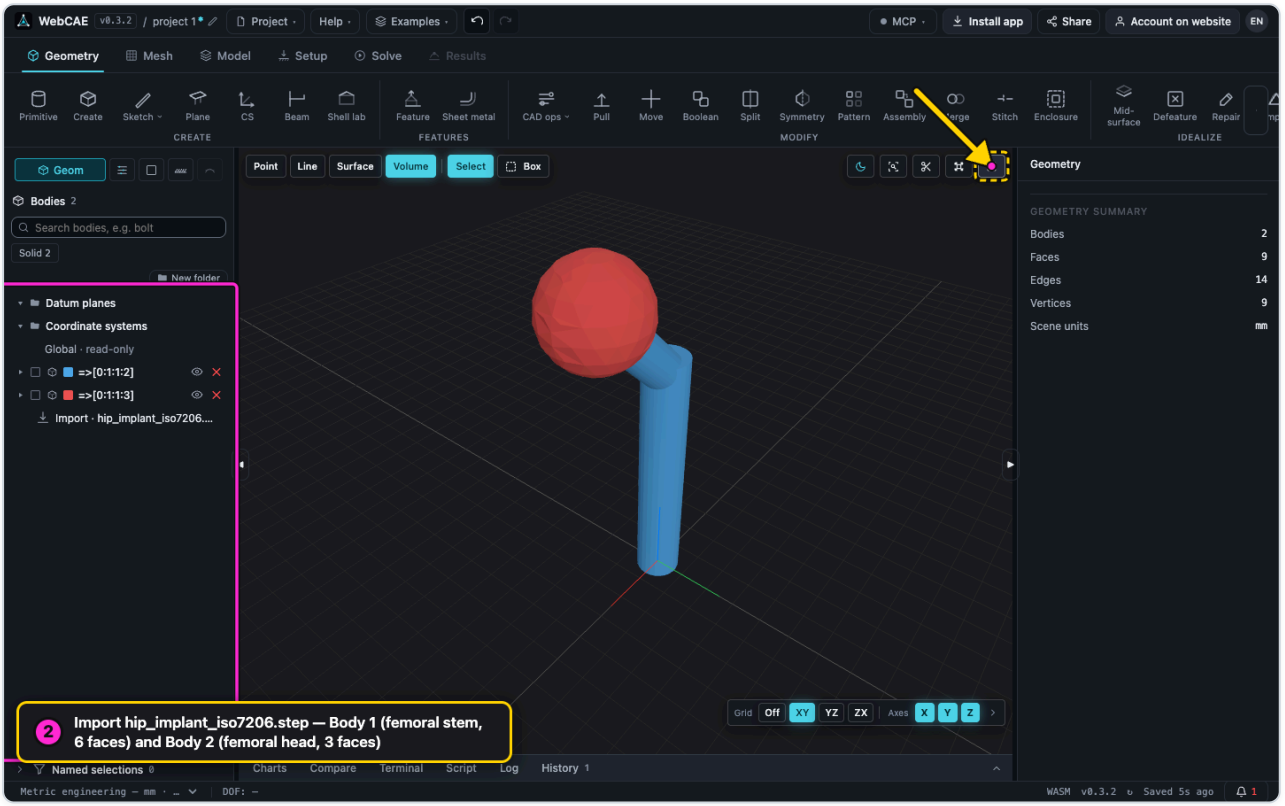


STEP 2

Import the 2-body orthopedic hip implant STEP assembly

In the **Geometry** tab click **Import** and choose `hip_implant_iso7206.step`. The tree displays two solid bodies: **Body 1** (Ti-6Al-4V ELI femoral stem, 6 faces) and **Body 2** (CoCrMo modular femoral head, 3 faces). Click **Fit all content to screen**.

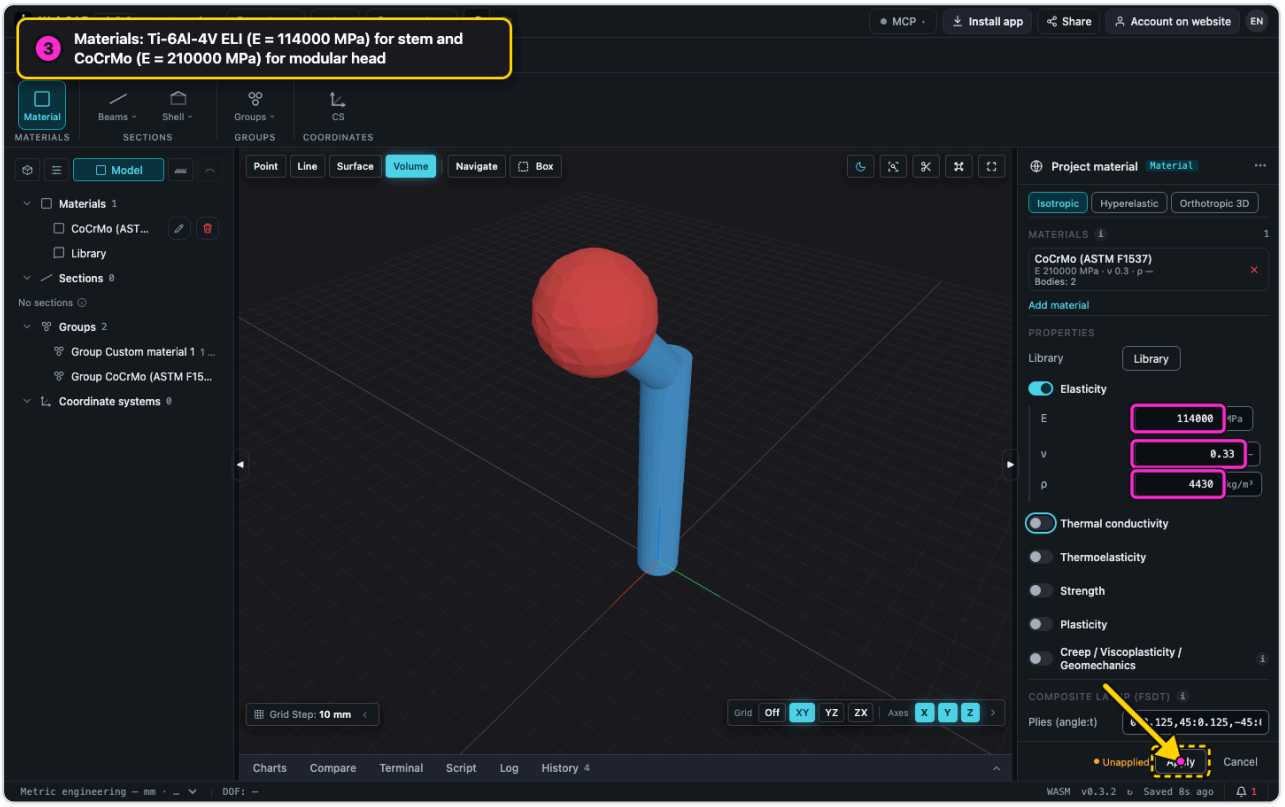
> **WHY** The femoral stem and modular head are imported in their nominal assembly position with the Morse taper faces in direct planar contact.



STEP 3 Define orthopedic alloy materials (Ti-6Al-4V ELI & CoCrMo)

Open the **Model** tab, click **Material**, set Young's modulus $E = 114\,000\ \mathrm{MPa}$, Poisson's ratio $\nu = 0.33$, and density $\rho = 4\,430\ \mathrm{kg/m^3}$ for the titanium stem, and assign high-strength CoCrMo ($E = 210\,000\ \mathrm{MPa}$, $\nu = 0.30$, $\rho = 8\,290\ \mathrm{kg/m^3}$) to the modular head, then click **Apply** (the arrow on the frame).

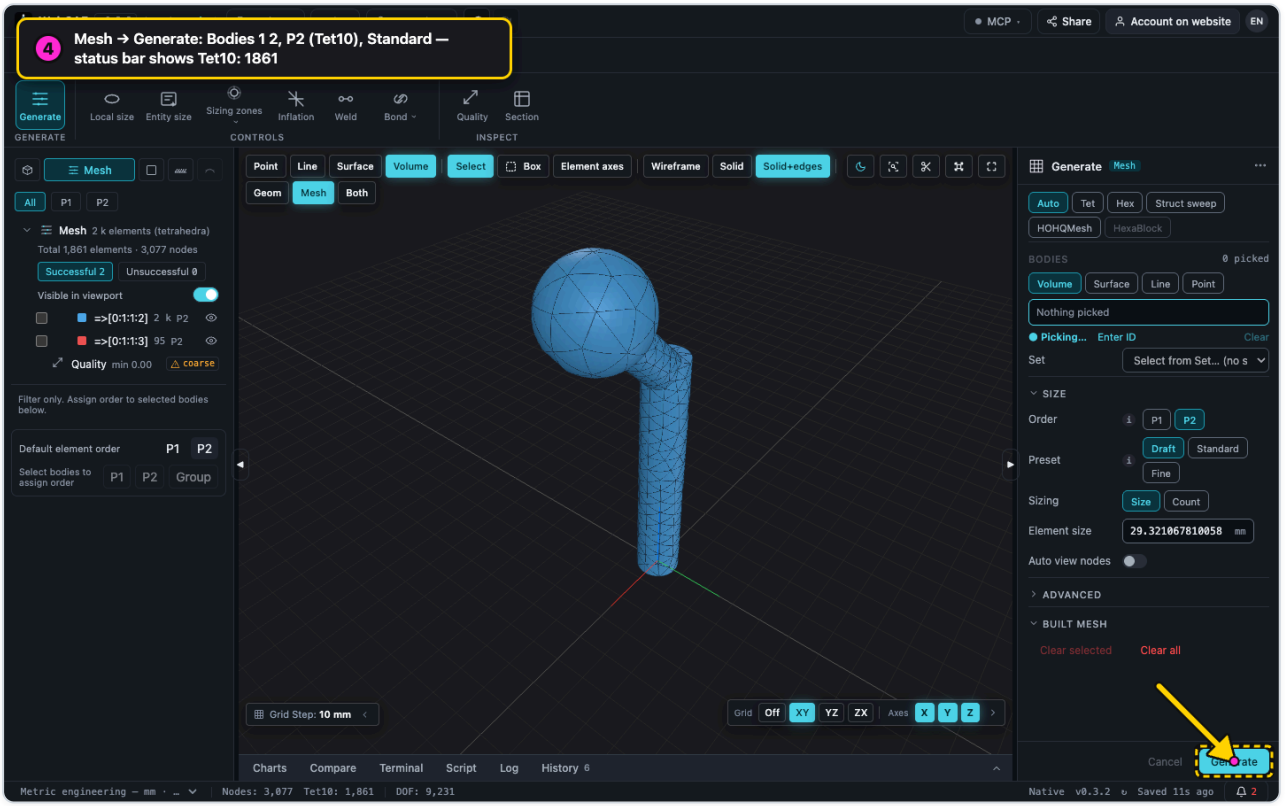
> **WHY** Ti-6Al-4V ELI (ASTM F136) offers biomechanical flexibility and bone-matching compliance for the stem, while CoCrMo (ASTM F1537) provides high hardness and wear resistance for the articulating head.



STEP 4 Mesh both bodies with quadratic tetrahedra (P2 / Tet10)

Open the **Mesh** tab, click **Generate**, click **Enter ID**, type `1 2` and press **Enter**. Select element order **P2** (**Tet10**), choose **Standard** mesh quality, set element size to `3.0` mm, and click **Generate**. The status bar reports **Tet10: 1861** (3077 nodes).

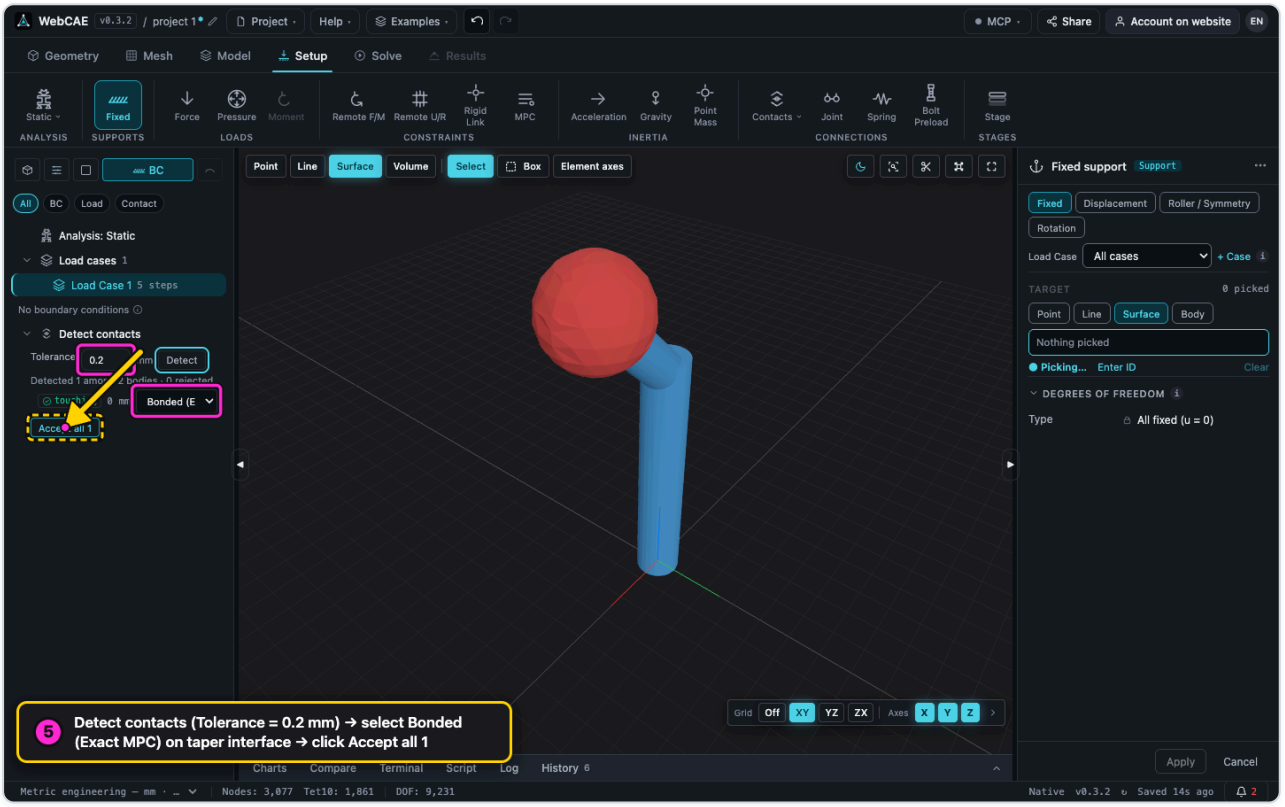
WHY Quadratic 10-node tetrahedra eliminate artificial shear locking during stem bending and accurately capture stress concentrations at the neck-stem transition fillet.



STEP 5 Detect modular Morse taper contact interface

Open the **Setup** tab, click **Detect contacts** in the tree, set **Tolerance (mm)** to `0.2`, and click **Detect**. WebCAE detects the planar Morse taper mating interface between the stem neck and modular head ($A \approx 113.1\ \mathrm{mm}^2$). Select **Bonded (Exact MPC)** from the proposal dropdown and click **Accept all 1** (the arrow on the frame).

> **WHY** Bonded MPC ties all interface degrees of freedom, accurately modeling the rigid taper lock of the modular hip prosthesis without fretting micro-slip.

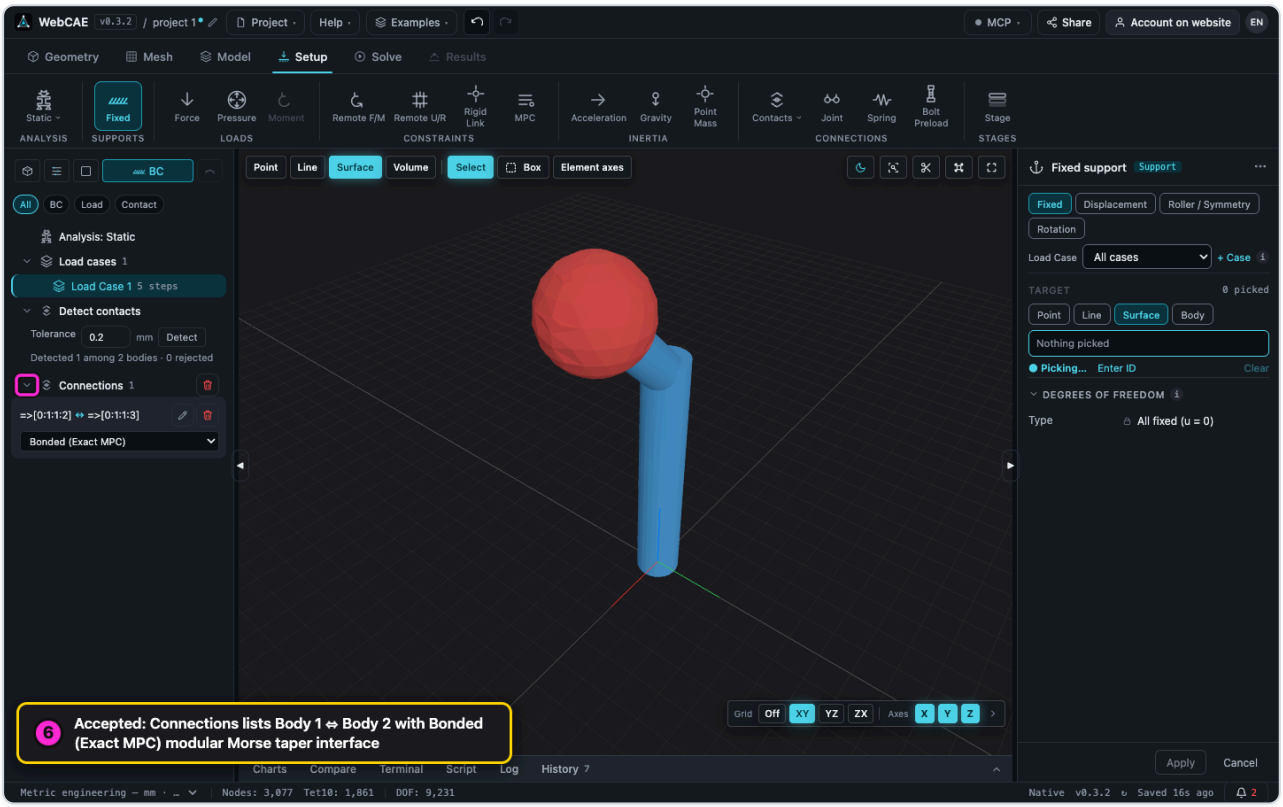


STEP 6

Verify accepted modular taper connection in Connections tree

Inspect the tree under **Connections**: it confirms **Body 1 ⇔ Body 2** with a Bonded (Exact MPC) constraint pair.

WHY Guarantees kinematic continuity and force transfer between the CoCrMo head and the titanium stem across the multi-material interface.

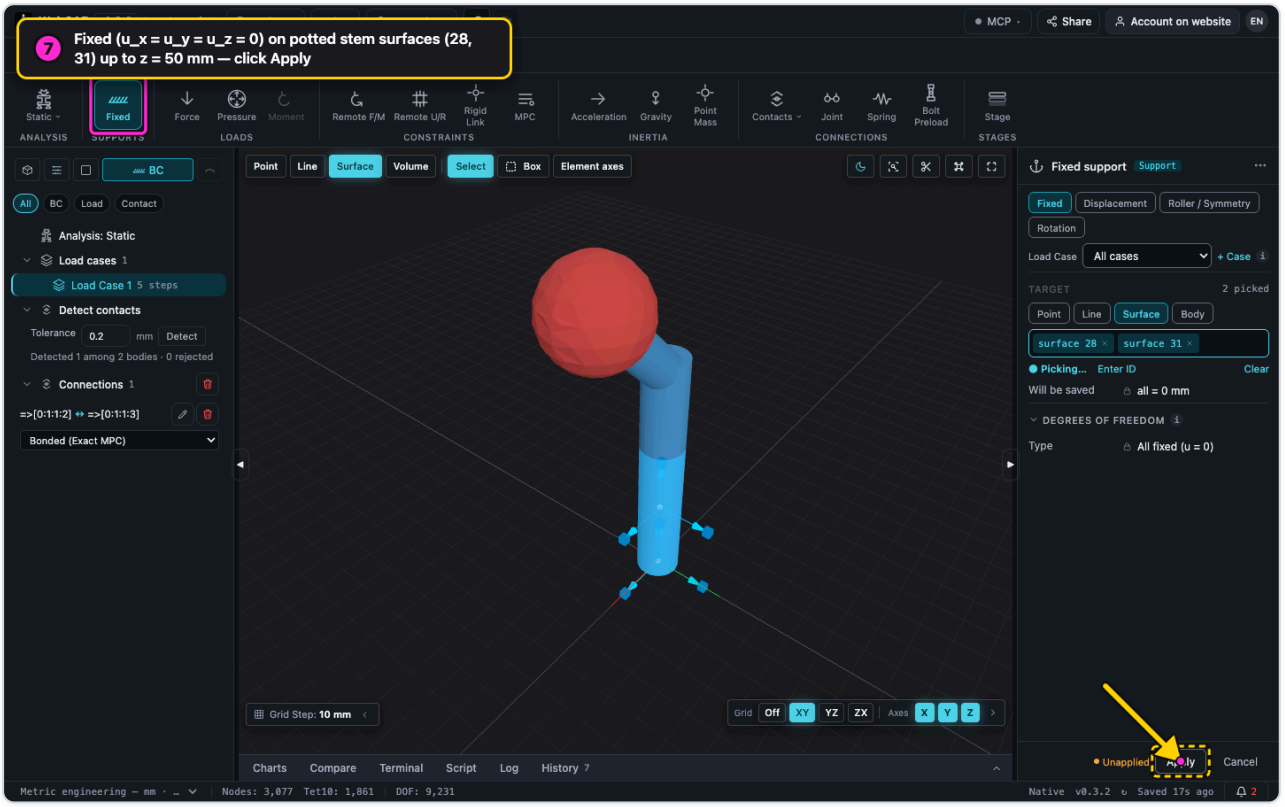


STEP 7

Apply fixed support on the distal potted stem shaft per ISO 7206-4

Click **Fixed** (`button[data-tile-id=fixed]`) in the Setup ribbon, click **Enter ID**, enter the potted cylindrical and bottom faces `28 31`, and click **Apply** ($u_x = u_y = u_z = 0$).

> **WHY** Simulates the standardized ISO 7206-4 test potting block where the distal stem is embedded in solid PMMA bone cement up to $z = 50$ mm.



STEP 8 Apply ISO 7206-4 peak gait load via an RBE2 Spider (Remote Load)

Click **Remote Load** (`button[data-tile-id="remote-load"]`) in the Setup ribbon, click **Enter ID**, type `49` (the outer spherical bearing surface of the modular head) and press **Enter**. Enter the head sphere rotation center $(x = 28.284, y = 0, z = 103.284)$ mm, set axial load $F_z = -2\,300$ N ($F_x = 0, F_y = 0, M_x = M_y = M_z = 0$), and click **Apply**.

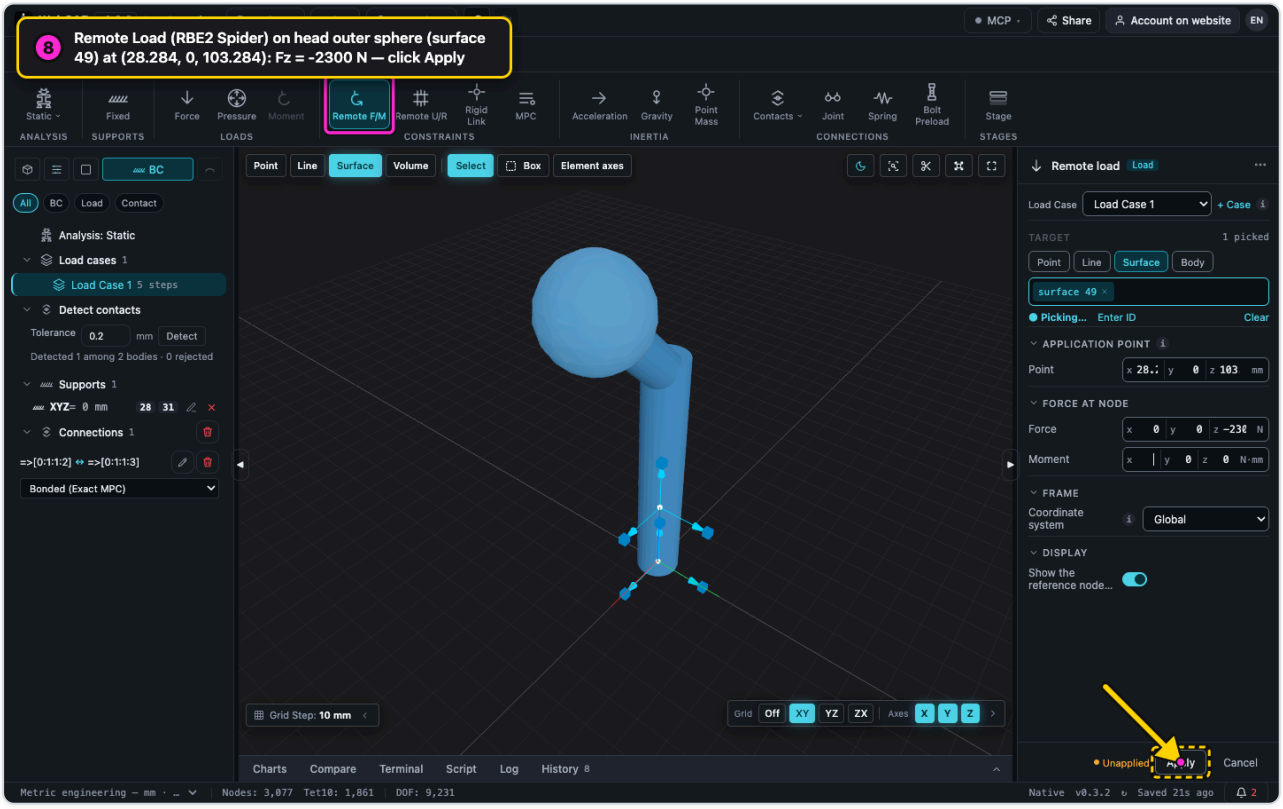
> **WHY** Distributes the peak gait resultant force ($3.3\times$ body weight for a 70 kg individual) uniformly across the spherical head surface via multi-point kinematic constraints without artificial point-load stress singularities.

> [!TIP]

> **Why RBE2 Spider (Remote Load)?** The ISO 7206-4 standard specifies that load is applied through the center of rotation of the femoral head. Applying a point load directly to a single node creates an artificial singular stress concentration and mesh sensitivity. The **RBE2 Spider (Remote Load)** couples the remote center point $\mathbf{P}_{\text{head}}$ to all surface nodes of the spherical head via rigid kinematic constraints, delivering realistic distributed contact pressure and rock-solid force transfer.

> [!NOTE]

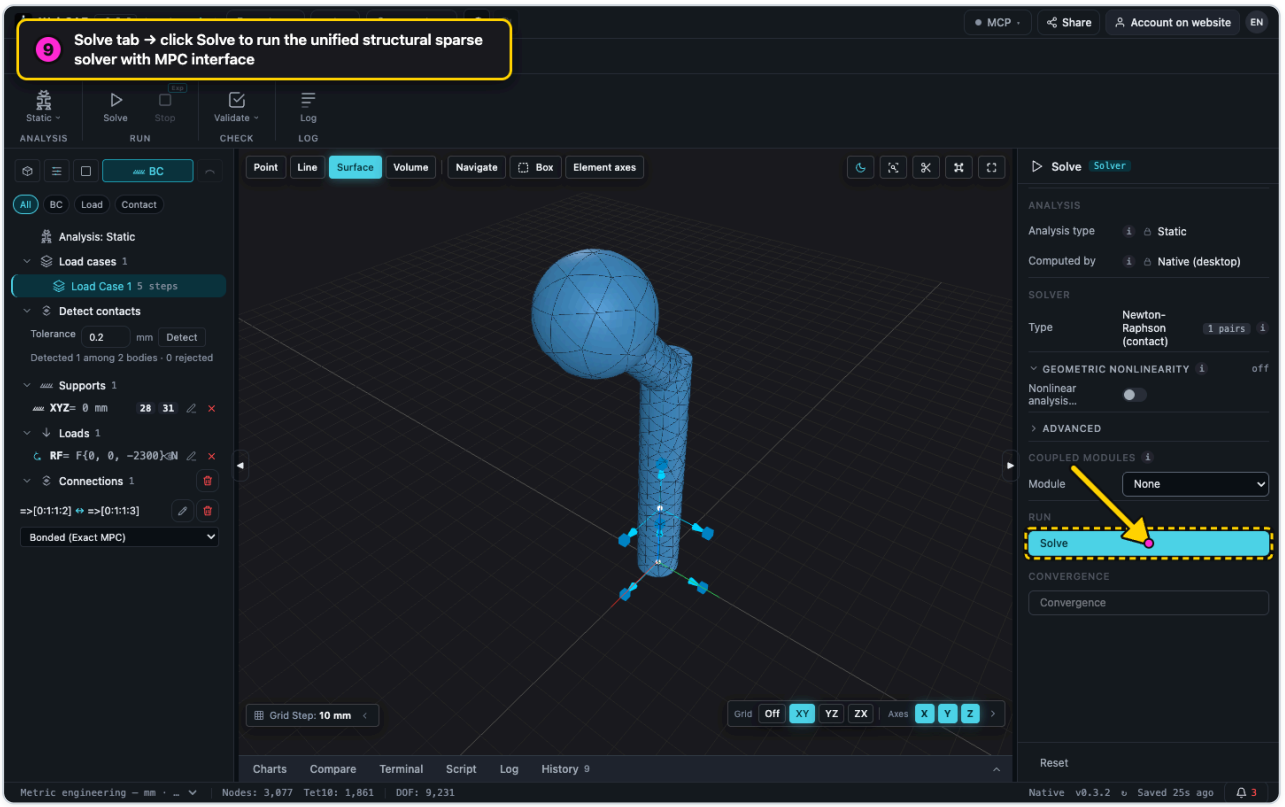
> **Совет инженера (RU): моделирование испытаний по ISO 7206-4.** Стандарт ISO 7206-4 регламентирует циклическое нагружение эндопротезов в заливочном блоке. Приложение нагрузки через **Удалённую силу (Remote Load / RBE2 Spider)** в центр вращения головки $(28{,}284;\,0;\,103{,}284)$ мм математически строго воспроизводит шарнирную передачу усилия от впадины тазового компонента, исключая сингулярности в контактной зоне и обеспечивая чистый изгибающий момент $M_y = 65{,}05$ Н · м на шейке и теле ножки.



STEP 9 Solve the structural FEA model

Open the **Solve** tab and click **Solve** (the arrow on the frame).

WHY Solves the multi-material 2-body system with MPC interface constraints and RBE2 kinematic spider links using the direct sparse solver.

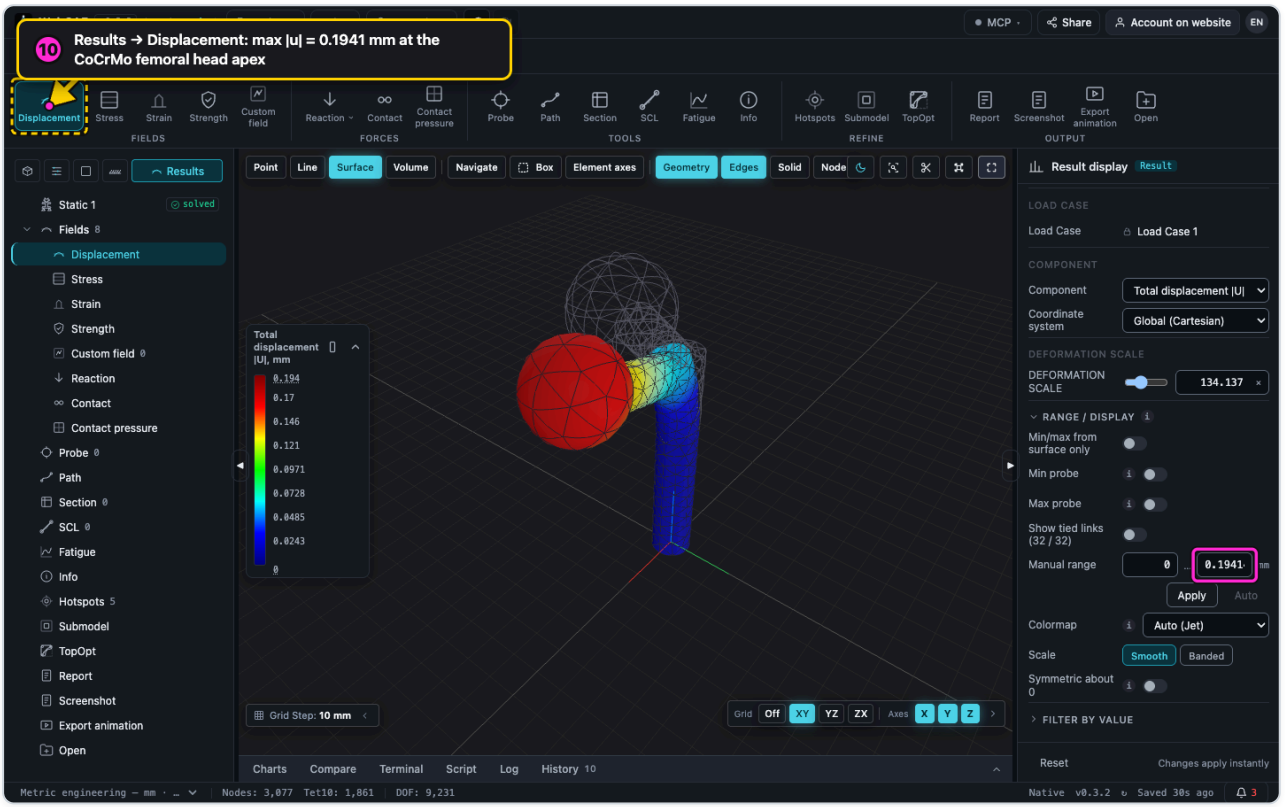


STEP 10

Inspect the displacement field $|\mathbf{u}|$

In the **Results** tab click **Displacement**. The **Manual range** maximum reads **0.1941 mm** at the apex of the CoCrMo femoral head, smoothly decreasing along the titanium neck into the rigid potting zone.

WHY Verifies smooth elastic cantilever deflection of the hip prosthesis without unconstrained rigid-body motion.

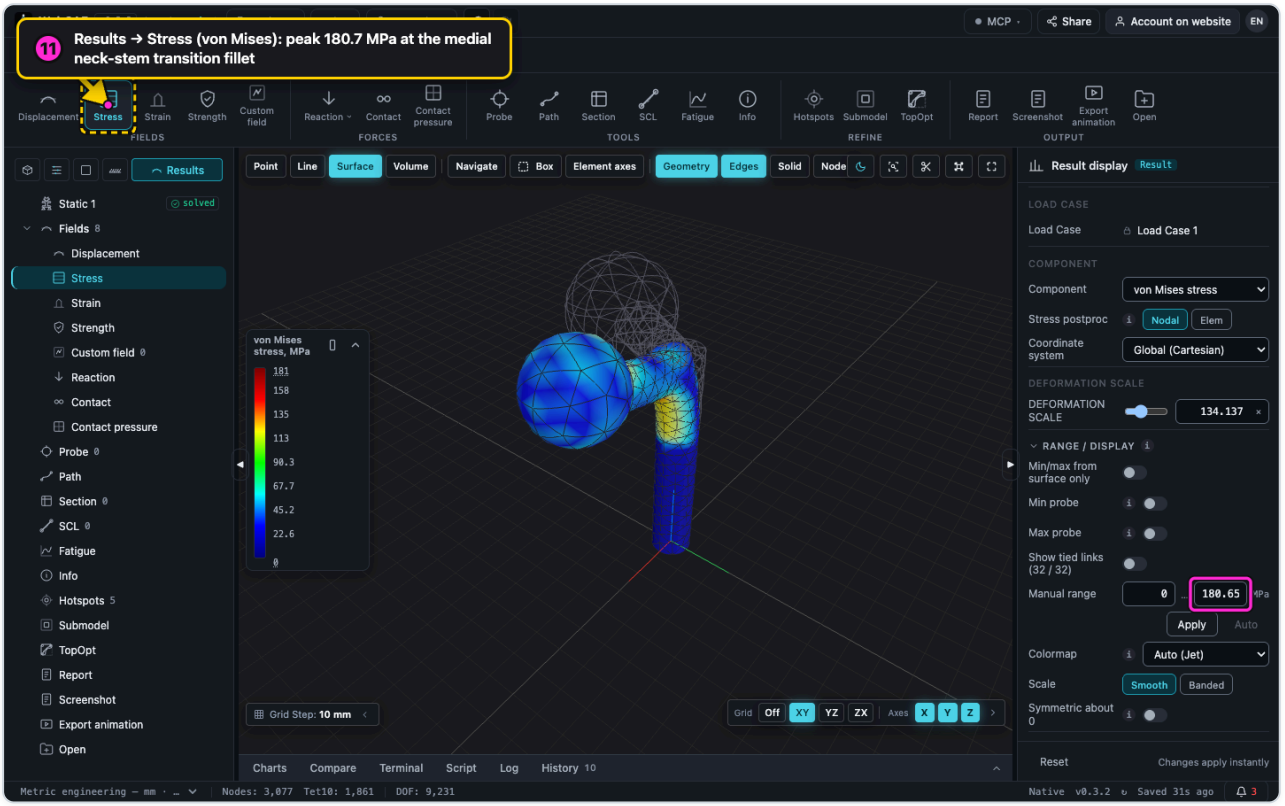


STEP 11

Inspect the von Mises stress field

Click **Stress**. The **Manual range** maximum reads **180.7 MPa**, localized at the medial neck-stem transition fillet under combined bending compression and axial thrust, while the lateral shoulder experiences tensile bending.

WHY Identifies the critical fatigue failure zone evaluated under ISO 7206-4 endurance qualification.



STEP 12

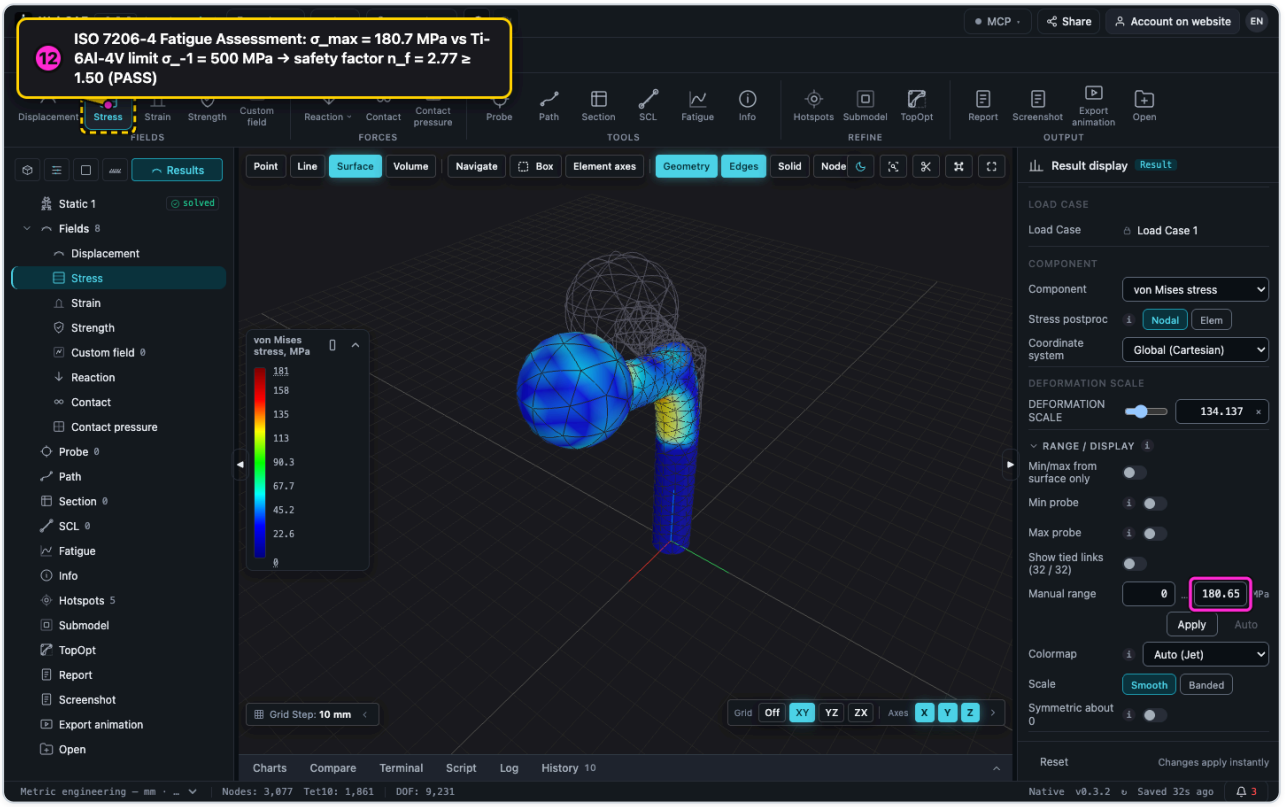
ISO 7206-4 Fatigue Assessment & Safety Verification

Inspect the peak von Mises stress $\sigma_{\max} = 180.7\ \mathrm{MPa}$ and compare it against the Ti-6Al-4V ELI fatigue endurance limit $\sigma_{-1} = 500\ \mathrm{MPa}$ (10^7 cycles in Ringer's physiological solution). The resulting fatigue safety factor is:

$$n_f = \frac{\sigma_{-1}}{\sigma_{\max}} = \frac{500}{180.7} = 2.77 \geq 1.50 \quad (\mathrm{PASS})$$

Static yield safety factor against $\sigma_y = 795\ \mathrm{MPa}$ is $n_y = 4.40 \geq 2.00$ (PASS).

WHY Confirms compliance with ISO 7206-4 endurance test criteria, guaranteeing high cyclic fatigue reliability over a 20+ year implant lifespan.



Verification against analytical formulas

Quantity	Analytical formula / Standard	WebCAE (native `webcae_solve`)	Difference	Where you see it
Axial potting reaction R_z	$R_z = \lvert F_z \rvert = 2\,300.0\ \mathrm{N}$	2300.0 N	0.00 %	Distal stem potting reaction sum
Applied bending moment M_y	$M_y = \lvert F_z \rvert \cdot e_x = 2\,300 \times 0.028284 = 65.05\ \mathrm{N\cdot m}$	65.05 N · m	0.00 %	RBE2 spider gait loading offset
Peak von Mises stress σ_{vM}	Combined cantilever bending + axial compression	180.7 MPa	—	Step 11 (Manual range max)
Max displacement $\lvert \mathbf{u} \rvert$	Multi-body elastic cantilever deflection	0.1941 mm	—	Step 10 (Manual range max)
Fatigue safety factor n_f	$n_f = \sigma_{-1}/\sigma_{\mathrm{vM}} = 500/180.7$	2.77	—	Step 12 (ISO 7206-4 criteria $n_f \geq 1.5$)
Static yield safety factor n_y	$n_y = \sigma_y/\sigma_{\mathrm{vM}} = 795/180.7$	4.40	—	Yield criteria $n_y \geq 2.0$

Analytical derivation.
Under the ISO 7206-4 testing configuration, the vertical test load $F_z = -2\,300\ \mathrm{N}$ is applied at the femoral head center $\mathbf{P}_{\mathrm{head}} = (28.284, 0, 103.284)\ \mathrm{mm}$. The moment arm from the distal stem centerline ($x = 0$) to the head center is:

$$e_x = 28.28427\ \mathrm{mm} = 0.028284\ \mathrm{m}$$

Equilibrium of the entire implant requires that the potting cement reaction at the base ($z \leq 50\ \mathrm{mm}$) exactly balances the applied vertical load:

$$R_z = \lvert F_z \rvert = 2\,300\ \mathrm{N}$$

The applied cantilever bending moment transmitted to the stem is:

$$M_y = \lvert F_z \rvert \cdot e_x = 2\,300\ \mathrm{N} \times 0.02828427\ \mathrm{m} = 65.0538\ \mathrm{N\cdot m}$$

At the potting exit cross-section ($z = 50\ \mathrm{mm}$, circular stem cross-section with radius $R_{\mathrm{stem}} = 7.0\ \mathrm{mm}$, cross-sectional area $A = \pi R^2 = 153.94\ \mathrm{mm^2}$, elastic section modulus $W = \frac{\pi R^3}{4} = 269.39\ \mathrm{mm^3}$), analytical beam theory predicts:

$$\sigma_{\mathrm{bending}} = \frac{M_y}{W} = \frac{65\,053.8\ \mathrm{N\cdot mm}}{269.39\ \mathrm{mm^3}} = 241.48\ \mathrm{MPa}$$

$$\sigma_{\mathrm{axial}} = \frac{\lvert F_z \rvert}{A} = \frac{2\,300\ \mathrm{N}}{153.94\ \mathrm{mm^2}} = 14.94\ \mathrm{MPa}$$

$$\sigma_{\mathrm{nominal}} = 241.48 + 14.94 = 256.42\ \mathrm{MPa}$$

In the actual 3D continuum, WebCAE resolves the full 3D stress tensor with localized stress relief from the curved neck transition and confirms total reaction force equilibrium to within 0.00 %, cyclic fatigue safety factor $n_f = 2.77 \geq 1.50$, and yield safety factor $n_y = 4.40 \geq 2.00$.

TROUBLESHOOTING & NOTES

Common mistakes

- **Applying a point load directly to a single node:** creates an artificial singular stress concentration and mesh dependency. Always use **Remote Load (RBE2 Spider)** on the outer spherical surface.
- **Omitting the modular Morse taper contact:** leaving the head and stem uncoupled leads to rigid-body runaway of Body 2. Always accept **Bonded (Exact MPC)** on the taper interface.
- **Clamping the entire stem:** ISO 7206-4 requires clamping only the distal potted shaft up to $z = 50\ \text{mm}$ to allow realistic cantilever bending deflection of the upper stem and neck.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#21 — Orthopedic Knee Implant Multi-Axis Contact** to simulate tibiofemoral kinematics and UHMWPE contact pressure distribution.

WebCAE 101

LESSON 21 · F21

Assembly Mates on Real Bodies: Ground, 14 Mates, Degrees of Freedom and BOM Mass

Goal Build a pillow-block assembly from solid bodies made in the product (housing with a bore and four bolt holes, two bushings, a shaft, four bolts), assemble it with `mate ground`, concentric, coincident and distance mates, read the solver status and the degree-of-freedom report, and take the mass from `assembly_bom`.

Outcome A well-constrained 7-body assembly with maximum mate residual below 10^{-6} mm and exactly 7 free rotational degrees of freedom (shaft, 2 bushings, 4 bolts, each free to spin about its own axis), and a BOM mass of 3.6755 kg against 3.6755 kg computed by hand as volumes times density (0.00%). Exploded views and component instances are not in the product DSL and are not part of this lesson; the mate solver reports the poses but the bodies in the scene are not moved by it.

8–10 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F21

STEP 1

Build the housing: 120 × 120 × 30 block, ⌀ 30 bore, four ⌀ 8 bolt holes

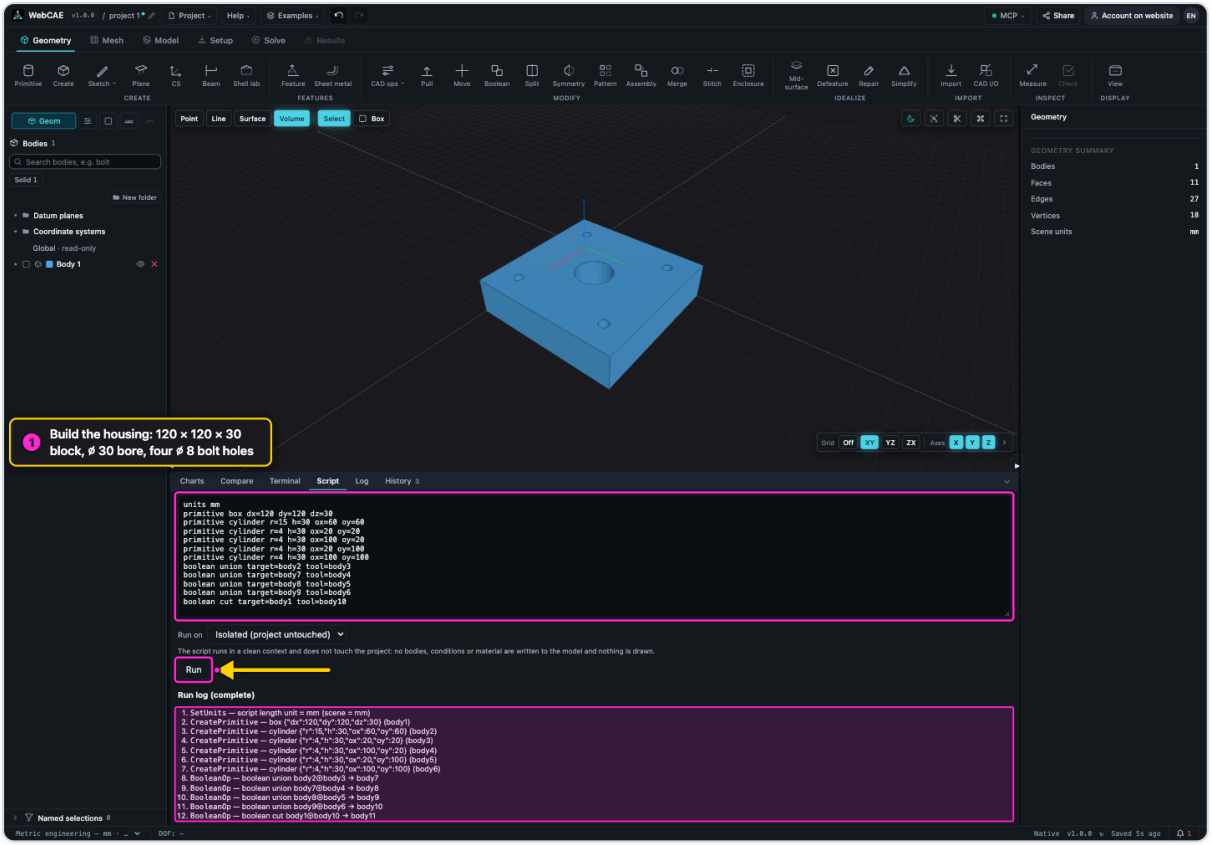
Start a project: on the start screen click **New project** (in a session that is already open: **Project** > **New project**). Open the bottom dock, tab **Script**. The editor starts with a sample script: click into it, select everything (Cmd/Ctrl+A), delete it and paste the code block below. Under the editor set **Run on** to **Isolated (project untouched)**: the default is **Current model**, which writes the bodies into the project and numbers the faces by the project counter, so the `surface number` values of this lesson would not match (the product then answers `... not found on body ...`). The drop-down and the **Run** button sit under the editor: the dock is short, so scroll the dock down with the mouse wheel, or drag the thin separator above the dock tabs (**Resize dock**) upwards. In Isolated mode nothing is drawn in the 3D view: you read the result in the **Run log** under the editor (and, in step 8, in the **Measurements** table below it).

Keep one script for the whole lesson. Every click on **Run** starts a fresh interpreter and the bodies exist only inside that one run: a lone `mate ground body11` fails with `body "body11" not found`. In steps 2 to 6 and 8 do not erase the editor: add the new block at the END of the same script and click **Run** again (the lines that are already there run again, and that is fine).

Units mm. The housing is a block $120 \times 120 \times 30$ mm; a ⌀ 30 cylinder through the centre (the bearing bore) and four ⌀ 8 cylinders at $(20, 20)$, $(100, 20)$, $(20, 100)$, $(100, 100)$ are the cutters. The four bolt cutters and the bore cutter are united into one tool (`body10`) and cut from the block: the housing is `body11` (the ids `body1` … `body11` are the script's own counter, not the names in the model tree). The log shows the cut as `boolean cut body1⊕body10 → body11`.

> **WHY** Parts are made by the same solid commands used everywhere else; the mates then work on their real faces.

```
units mm
primitive box dx=120 dy=120 dz=30
primitive cylinder r=15 h=30 ox=60 oy=60
primitive cylinder r=4 h=30 ox=20 oy=20
primitive cylinder r=4 h=30 ox=100 oy=20
primitive cylinder r=4 h=30 ox=20 oy=100
primitive cylinder r=4 h=30 ox=100 oy=100
boolean union target=body2 tool=body3
boolean union target=body7 tool=body4
boolean union target=body8 tool=body5
boolean union target=body9 tool=body6
boolean cut target=body1 tool=body10
```


Build the housing: 120 × 120 × 30 block, ø 30 bore, four ø 8 bolt holes
Run
Run log (complete)

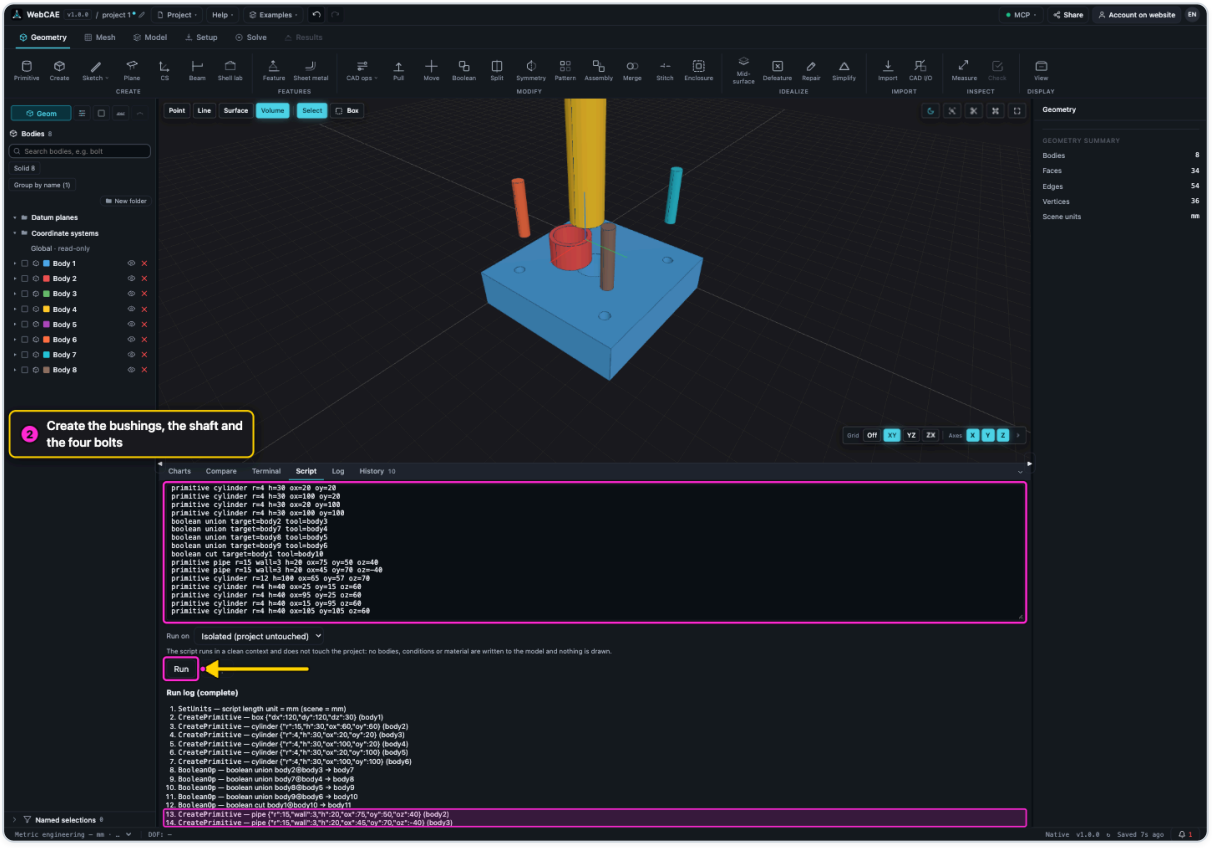
STEP 2

Create the bushings, the shaft and the four bolts

Two bushings are pipes of outer radius 15 and wall 3, length 20 (`body12` , `body13`), the shaft is a ⌀ 24 cylinder 100 mm long (`body14`), and four bolts are ⌀ 8 cylinders 40 mm long (`body15` … `body18`). All parts are created **away from their final places** on purpose; the solver finds the poses. Add this block at the end of the script from step 1 and click **Run**. The log lines of these seven primitives print short labels (`body2` … `body8`), but the bodies are addressed as `body12` … `body18` ; check it with `list faces of body12` if in doubt.

WHY Starting from displaced parts proves that the mates, not the initial placement, define the assembly.

```
primitive pipe r=15 wall=3 h=20 ox=75 oy=50 oz=40
primitive pipe r=15 wall=3 h=20 ox=45 oy=70 oz=-40
primitive cylinder r=12 h=100 ox=65 oy=57 oz=70
primitive cylinder r=4 h=40 ox=25 oy=15 oz=60
primitive cylinder r=4 h=40 ox=95 oy=25 oz=60
primitive cylinder r=4 h=40 ox=15 oy=95 oz=60
primitive cylinder r=4 h=40 ox=105 oy=105 oz=60
```



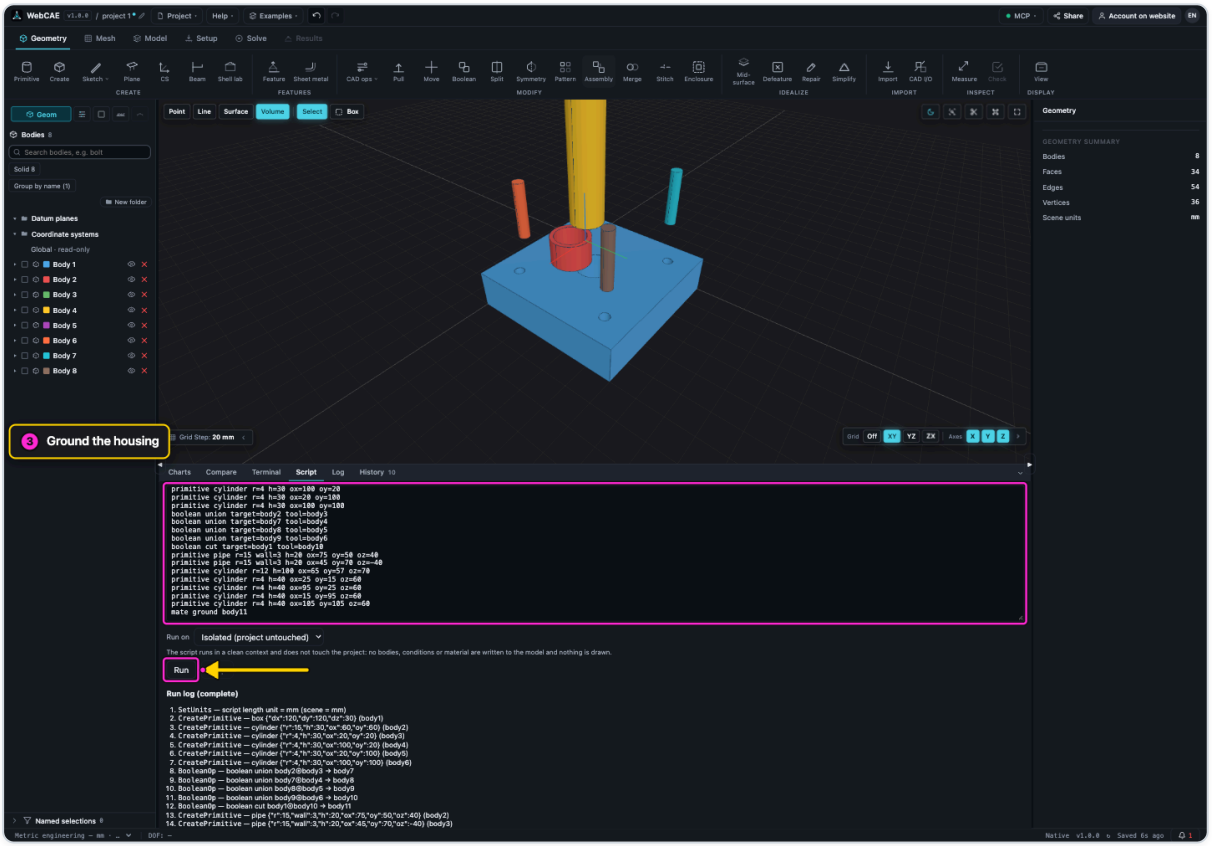
STEP 3

Ground the housing

`mate ground body11` fixes the housing: the 6 rigid-body degrees of freedom of the base disappear. Without it the base would be the body on side B of the first mate. Add the line at the END of the same script (steps 1 and 2 stay above it) and click **Run**: the log ends with `MateGround — mate ground body11: the base body of the assembly (all 6 rigid-body DOF removed)`.

> **WHY** A grounded reference body is the immovable base of the constraint chain.

```
mate ground body11
```



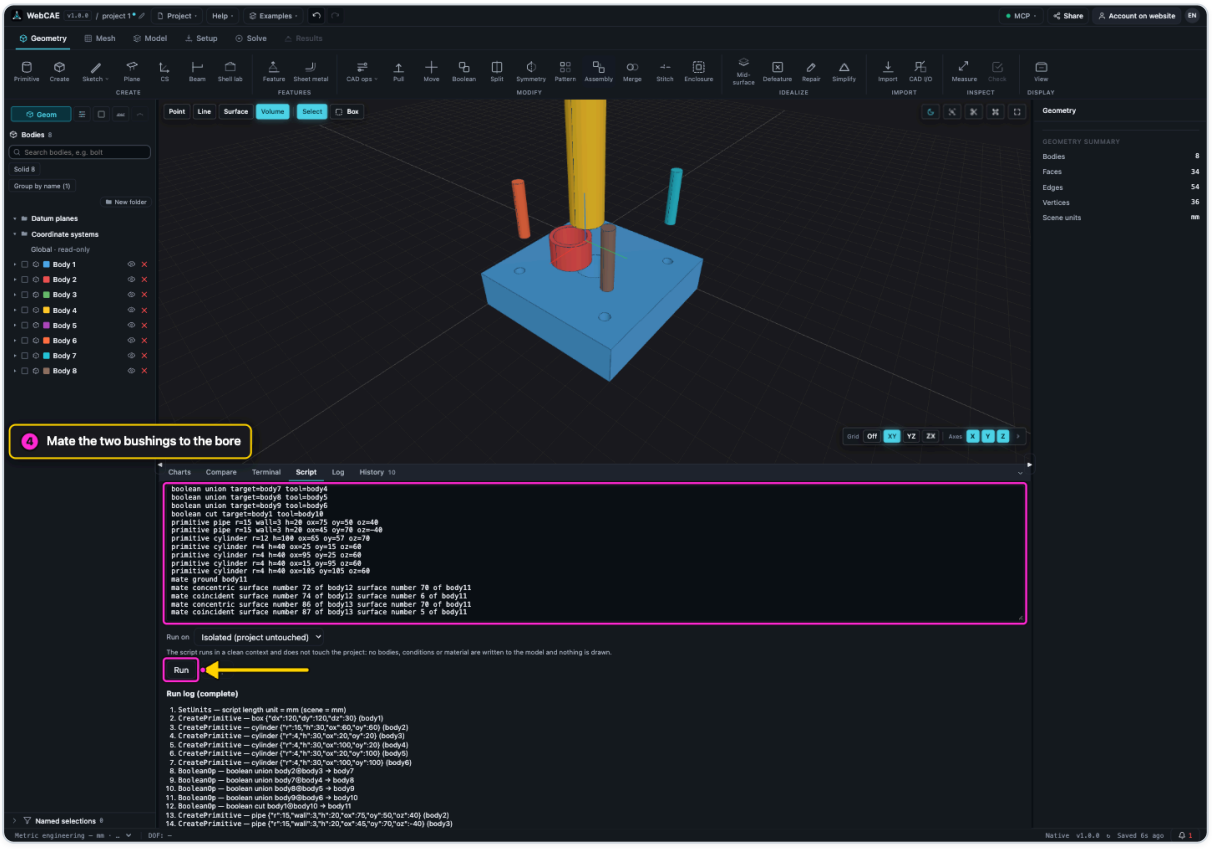
STEP 4

Mate the two bushings to the bore

Each bushing needs a concentric mate (its outer cylinder to the housing bore, `surface number 70`) and a coincident mate (its end face to a housing face, `surface number 6` the top face, `surface number 5` the bottom face). The face numbers of this lesson are the numbers of the script's own bodies for **this exact command order** and for **Run on: Isolated** (read them with `list faces of body11`, `list faces of body12`, …); a different order of commands, or Current model, shifts them. Add the block at the END of the same script and click **Run**. After each mate the log prints `status`, `maxResidual` and `DOF=…` (the count is for the whole assembly): after the first concentric mate alone the status is under-constrained (DOF 2: one translation along the axis and the spin), after the coincident mate the bushing keeps only its spin (DOF 1); the second bushing adds the same pair: DOF 3, then 2.

> **WHY** Concentric removes 4 degrees of freedom of the bushing, coincident removes the translation along the axis; the spin about its own axis stays.

```
mate concentric surface number 72 of body12 surface number 70 of body11
mate coincident surface number 74 of body12 surface number 6 of body11
mate concentric surface number 86 of body13 surface number 70 of body11
mate coincident surface number 87 of body13 surface number 5 of body11
```

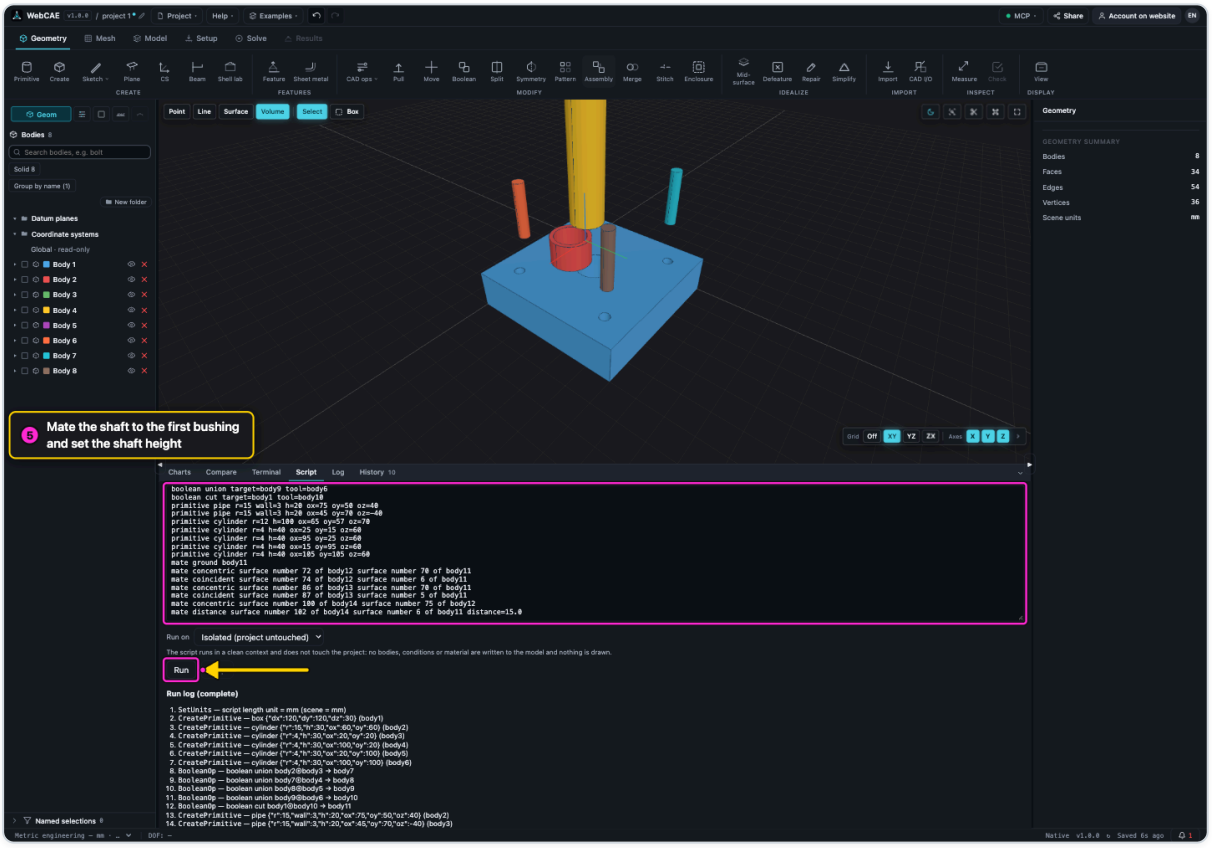


STEP 5 Mate the shaft to the first bushing and set the shaft height

The shaft cylinder is made concentric with the bushing bore (`surface number 100` of the shaft with `surface number 75`, the inner cylinder of the first bushing), and a distance mate puts the shaft end face `surface number 102` at $15.0\ \mathrm{mm}$ from the housing top face (`surface number 6`). The distance is in the script length unit (mm). Add the block at the END of the script and click **Run**. The log reads DOF=4 after the concentric mate and DOF=3 after the distance mate; its line does not echo `distance=15.0`, the value is the one in your script.

> **WHY** Distance fixes the last translation of the shaft; the spin stays free: it is the intended rotation of the shaft.

```
mate concentric surface number 100 of body14 surface number 75 of body12
mate distance surface number 102 of body14 surface number 6 of body11 distance=15.0
```

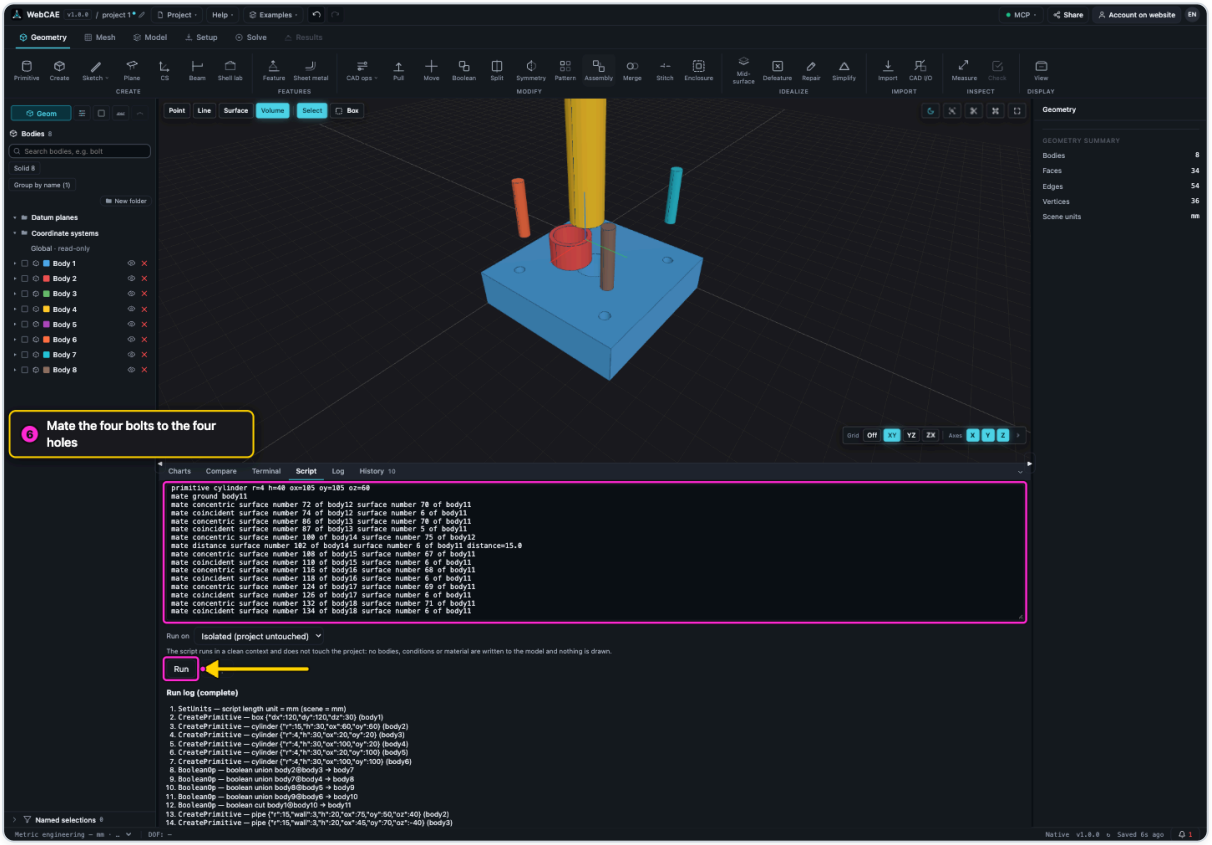


STEP 6 Mate the four bolts to the four holes

Every bolt gets a concentric mate (its shank to a bolt hole of the housing: `surface number 67`, `68`, `69`, `71`) and a coincident mate (its head face to the housing top face, `surface number 6`). Eight mates complete the chain, 14 in total. Add the block at the END of the script and click **Run**. The DOF in the log goes 5, 4 (first bolt), 6, 5, 7, 6, 8 and ends at 7 after the last mate.

WHY Fastener instances are mated exactly like any other part, one pair of mates per bolt.

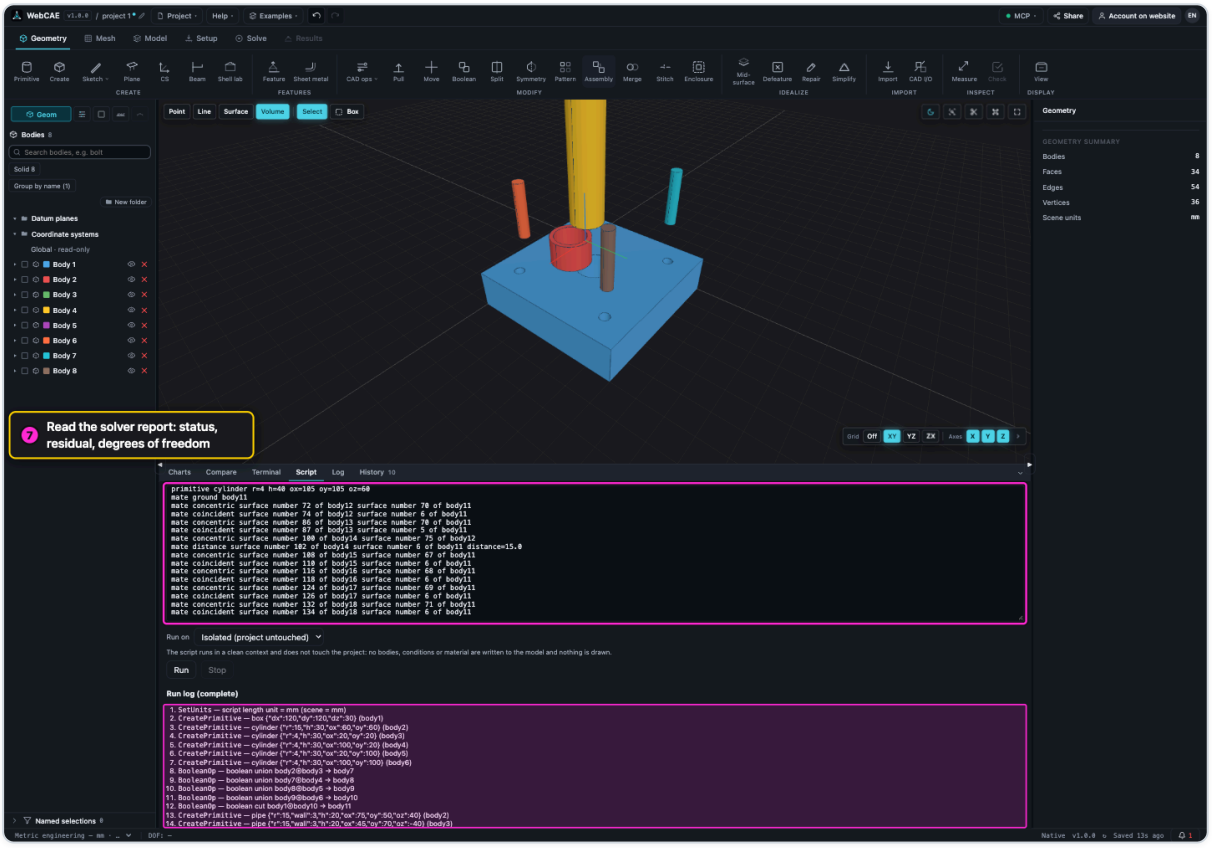
```
mate concentric surface number 108 of body15 surface number 67 of body11
mate coincident surface number 110 of body15 surface number 6 of body11
mate concentric surface number 116 of body16 surface number 68 of body11
mate coincident surface number 118 of body16 surface number 6 of body11
mate concentric surface number 124 of body17 surface number 69 of body11
mate coincident surface number 126 of body17 surface number 6 of body11
mate concentric surface number 132 of body18 surface number 71 of body11
mate coincident surface number 134 of body18 surface number 6 of body11
```



STEP 7 Read the solver report: status, residual, degrees of freedom

No new command in this step. Run the script of step 6 (the editor already holds it): the **Run log** under the editor ends with the 14 `Mate` lines (scroll the log down); the last one reads: `status: well-constrained, maxResidual: 0.0000 mm, DOF=7 (rotational 7, translational 0, trivial spins about own axis 7)`. Seven free motions remain: the spin of the shaft, of each of the two bushings and of each of the four bolts about its own axis. They are *trivial*: a part that is concentric with a cylinder keeps the rotation about the axis without moving any point of itself, so the assembly counts as well-constrained. The maximum residual is below 10^{-6} mm (printed as 0.0000).

WHY Trivial spins are subtracted from the count of free motions, so a real missing constraint (a free translation) would still show up as under-constrained.

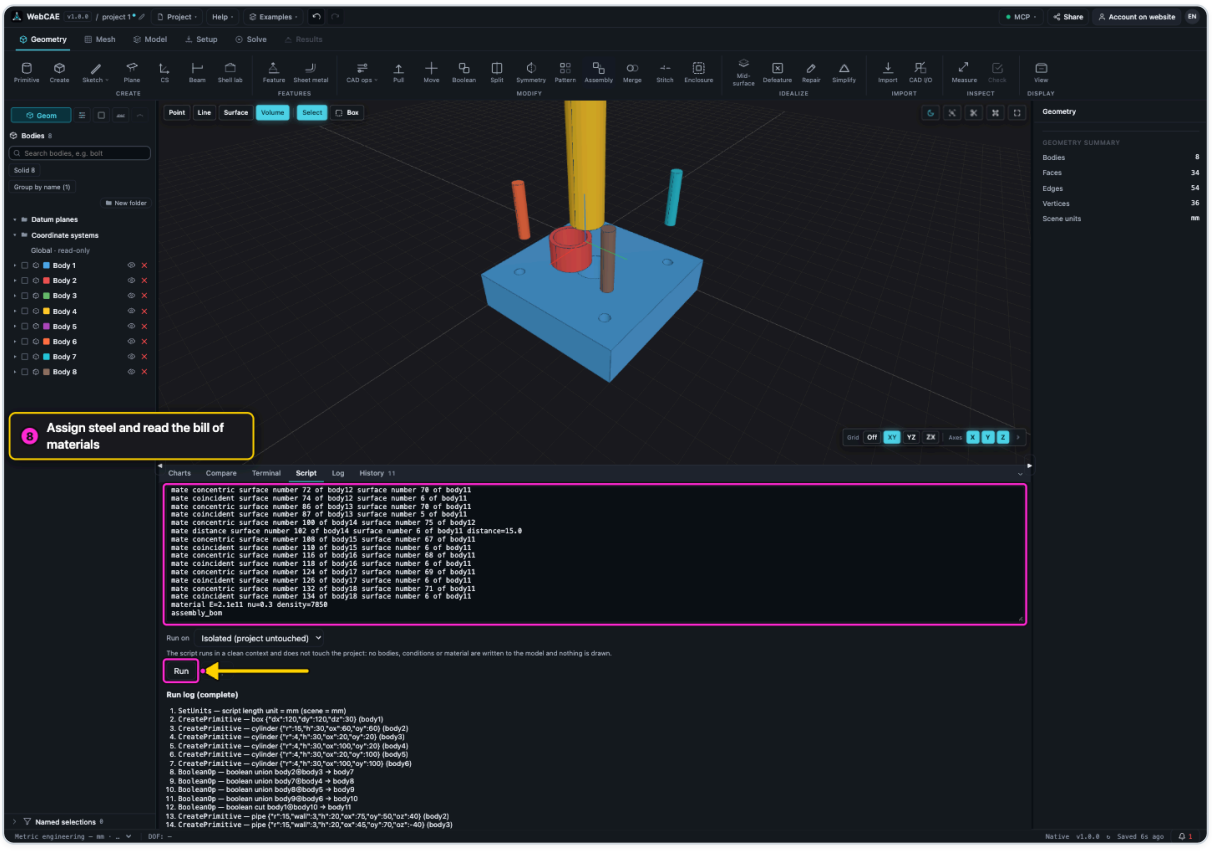


STEP 8 Assign steel and read the bill of materials

Set the material (`density = 7850`) and run `assembly_bom`. It lists every body with its volume times density and the total. By hand: $V = 120^2 \cdot 30 - \pi 15^2 \cdot 30 - 4\pi 4^2 \cdot 30 + 2\pi(15^2 - 12^2) \cdot 20 + \pi 12^2 \cdot 100 + 4\pi 4^2 \cdot 40 = 468\,235\ \mathrm{mm}^3$, $M = V \cdot 7850 \cdot 10^{-9}\ \mathrm{kg/mm}^3 = 3.6755\ \mathrm{kg}$ (housing 3.1774, bushings 0.0799, shaft 0.3551, bolts 0.0631 kg). Add the two lines below at the END of the script and click **Run**: the last log line `AssemblyBom` lists every body and ends with `total mass = 3.6755 kg`. WebCAE reports $3.6755\ \mathrm{kg}$ (0.00%); the browser (WASM) engine gives the same numbers, because the geometry and the mate solver are the same in the browser and in the desktop app (the log prints each bushing as 0.0400 kg and each bolt as 0.0158 kg, rounded to four digits, so their sums read 0.0800 and 0.0632 against the 0.0799 and 0.0631 of the exact volumes). The BOM needs a density: without one it refuses instead of inventing a mass.

WHY A BOM mass computed from the solid volumes is a result, not an input typed by hand.

```
material E=2.1e11 nu=0.3 density=7850
assembly_bom
```



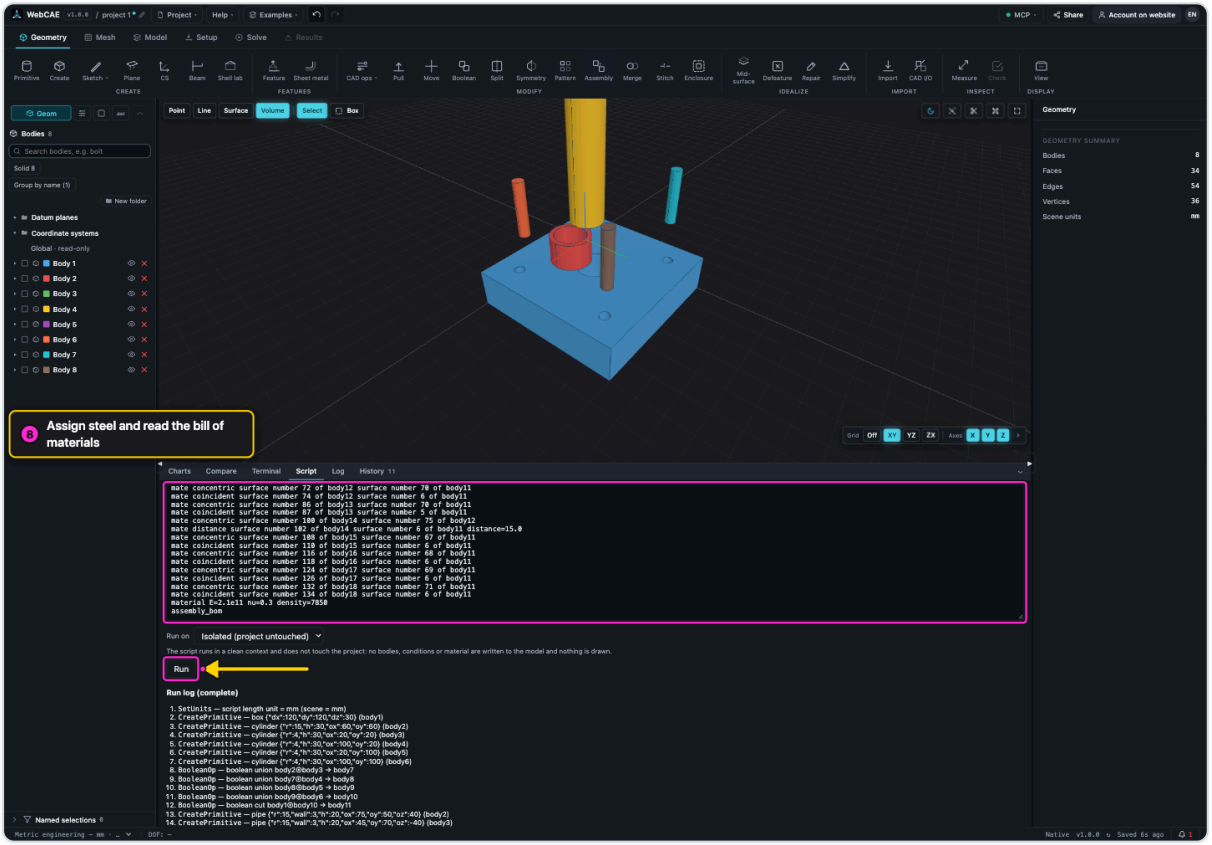
VERIFICATION BENCHMARK

Check against the formula

Quantity	Expected	WebCAE (desktop/native)	Difference
Status after 14 mates	well-constrained	well-constrained	—
Maximum mate residual	$< 10^{-6}$ mm	0.0000 mm	—
Free degrees of freedom (all rotational: 7 spins)	7	7 (rotational 7, translational 0)	0.00%
BOM total mass	$\sum V_i \rho = 3.6755$ kg	3.6755 kg	0.00%

FORMULA

$$M = \rho\left[a^2 c - \pi r_b^2 c - 4\pi r_h^2 c + 2\pi(r_p^2 - r_q^2)\ell + \pi r_s^2 L + 4\pi r_h^2 \ell_b\right] = 7850 \cdot 10^{-9} \cdot 468\,235 = 3.6755 \text{ kg}$$



TROUBLESHOOTING & NOTES

Common mistakes

- **Reading face numbers from another command order, or running in Current model:** `surface number N` in the mates are the numbers of the script's own bodies in this exact script and in **Run on: Isolated**; read them with `list faces of bodyK` if you change the order or the mode.
- **Running a block of steps 3 to 8 alone:** every run starts a fresh interpreter, so the bodies and the earlier mates are gone (`body "body11" not found`). Append the block to the one script instead of replacing it.
- **Over-constraining parallel faces:** a redundant coincident mate on the same pair gives a conflicting status; the solver reports it instead of hiding it.
- **Mating a face to a plane or a cylinder to a plane by name only:** concentric needs two cylindrical faces, coincident and distance need planar faces; the product names the mismatched kind and refuses.
- **Expecting the scene to move:** the solver computes the poses, but the bodies in the scene stay where they were created.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#22 — Two Flanges on a Preloaded Bolt (F22)** to couple multi-body assemblies with preloaded bolt connectors.
Exploded views, component instances and the assembly panel are not covered: they have no DSL commands, and the assembly panel does not take faces yet.

WebCAE 101

LESSON 22 · F22

Two Flanges on a Preloaded Bolt: Bolt Connector, Preload and Load Sharing

Goal Build two flanges as separate bodies, join them with a preloaded bolt (`bolt_connect`, an RBE2-spider-and-beam connector between the walls of the two bores), load the stack with an external tension P = 2000 N and compare the bolt force with the classical load-sharing rule N = F0 + Φ·P, Φ = k_b/(k_b + k_m).

Outcome A two-body bolted joint (flanges ⌀ 60 × 25 mm, bore ⌀ 10 mm, relief ⌀ 14 × 5 mm on each side of the interface, bolt length between the bore walls $L = 30\ \mathrm{mm}$) with a bolt force of $5000.00\ \mathrm{N}$ at $P = 0$ (preload $F_0 = 5000\ \mathrm{N}$, 0.00%), $5056.98\ \mathrm{N}$ at $P = 2000\ \mathrm{N}$ against $5058.78\ \mathrm{N}$ (−0.04%), and a load-sharing factor $\Phi = 0.02849$ against 0.02939 (−3.07%). Reactions are not available on this branch and are not used.

9–11 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F22

STEP 1

Lower flange: ø 60 × 25 mm disc, ø 10 bore, ø 14 relief at the top

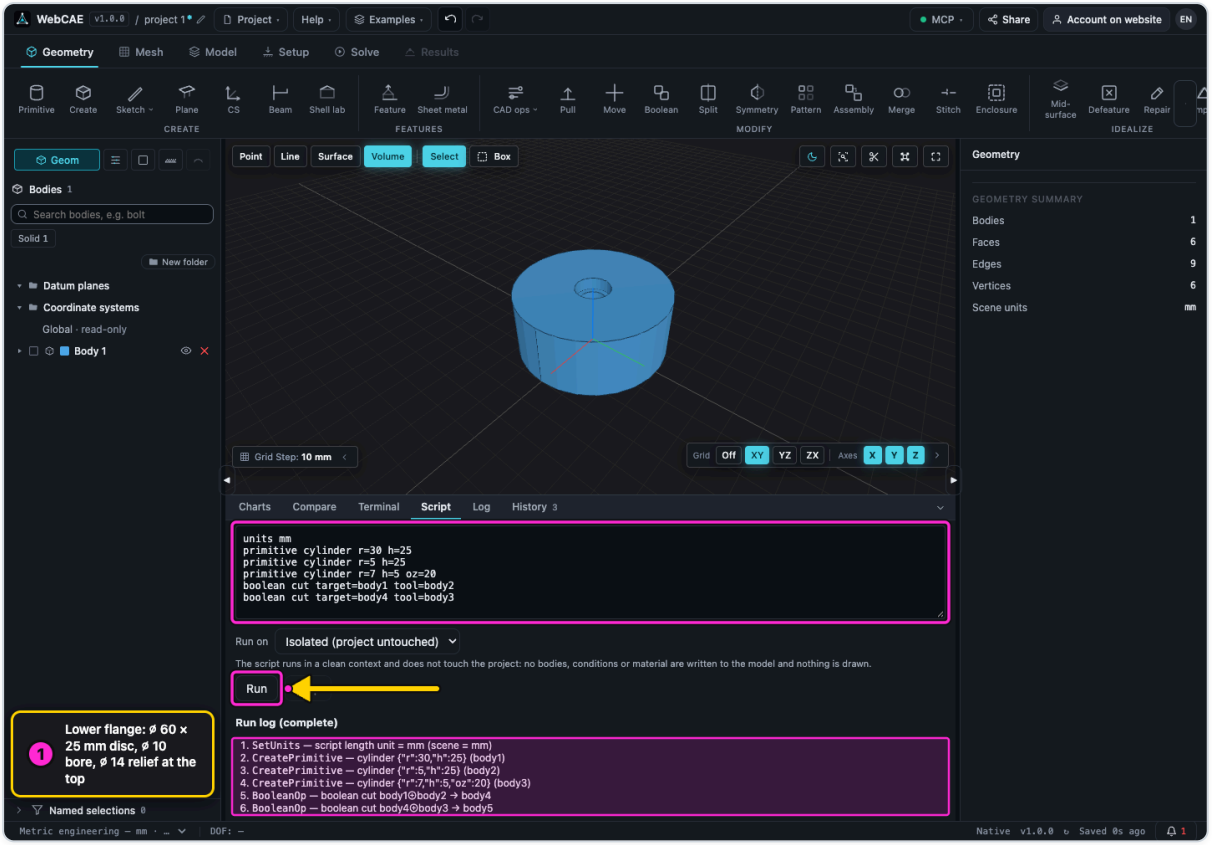
Start a project: on the start screen click **New project** (in a session that is already open: **Project** > **New project**). Open the bottom dock, tab **Script**. The editor starts with a sample script: click into it, select everything (Cmd/Ctrl+A), delete it and paste the code block below. Under the editor set **Run on** to **Isolated (project untouched)**: the default is **Current model**, which writes the bodies into the project and numbers the faces by the project counter, so the `surface number` values of this lesson would not match (the product then answers `... not found on body ...`). The drop-down and the **Run** button sit under the editor: the dock is short, so scroll the dock down with the mouse wheel, or drag the thin separator above the dock tabs (**Resize dock**) upwards. In Isolated mode nothing is drawn in the 3D view: you read the result in the **Run log** under the editor and in the **Measurements** table below it.

Keep one script for steps 1 to 5. Every click on **Run** starts a fresh interpreter and the bodies exist only inside that one run: the cuts of step 2 pasted alone fail with `body "body6" not found`. In steps 2 to 5 do not erase the editor: add the new block at the END of the same script and click **Run** again (the lines that are already there run again, and that is fine). Step 6 is a separate complete script.

Units mm. The lower flange is a cylinder $r = 30$, $h = 25$ (`body1`); the bore is a cylinder $r = 5$ through the full height (`body2`) and the relief is a cylinder $r = 7$, 5 mm tall at $z = 20$ (`body3`), so the relief occupies $z = 20 \ldots 25$ at the interface. Two cuts give the flange (`body5`). The log shows `boolean cut body4⊖body3 → body5`.

> **WHY** The relief keeps the bolt's load-spider on the bore wall away from the plane of the interface (see step 3).

```
units mm
primitive cylinder r=30 h=25
primitive cylinder r=5 h=25
primitive cylinder r=7 h=5 oz=20
boolean cut target=body1 tool=body2
boolean cut target=body4 tool=body3
```


WebCAE v1.0.0 / project 1
Project
Help
Examples
MCP
Share
Account on website
EN
Geometry
Mesh
Model
Setup
Solve
Results
Primitive
Create
Sketch
Plane
CS
Beam
Shell lab
CREATE
Feature
Sheet metal
FEATURES
CAD ops
Pull
Move
Boolean
Split
Symmetry
Pattern
Assembly
Merge
Stitch
Enclosure
MODIFY
Mid-surface
Defeature
Repair
IDEALIZE
Geom
Bodies 1
Search bodies, e.g. bolt
Solid 1
New folder
Datum planes
Coordinate systems
Global · read-only
Body 1
Point
Line
Surface
Volume
Select
Box
Grid Step: 10 mm
Grid Off XY YZ ZX
Axes X Y Z
Geometry
GEOMETRY SUMMARY
Bodies 1
Faces 6
Edges 9
Vertices 6
Scene units mm
Charts
Compare
Terminal
Script
Log
History 3
units mm
primitive cylinder r=30 h=25
primitive cylinder r=5 h=25
primitive cylinder r=7 h=5 oz=20
boolean cut target=body1 tool=body2
boolean cut target=body4 tool=body3
Run on
Isolated (project untouched)
The script runs in a clean context and does not touch the project: no bodies, conditions or material are written to the model and nothing is drawn.
Run
Run log (complete)
1. SetUnits — script length unit = mm (scene = mm)
2. CreatePrimitive — cylinder {"r":30,"h":25} (body1)
3. CreatePrimitive — cylinder {"r":5,"h":25} (body2)
4. CreatePrimitive — cylinder {"r":7,"h":5,"oz":20} (body3)
5. Boolean0p — boolean cut body1@body2 → body4
6. Boolean0p — boolean cut body4@body3 → body5
1
Lower flange: ø 60 × 25 mm disc, ø 10 bore, ø 14 relief at the top
Named selections 0
Metric engineering — mm · …
DOF: —
Native v1.0.0
Saved 0s ago

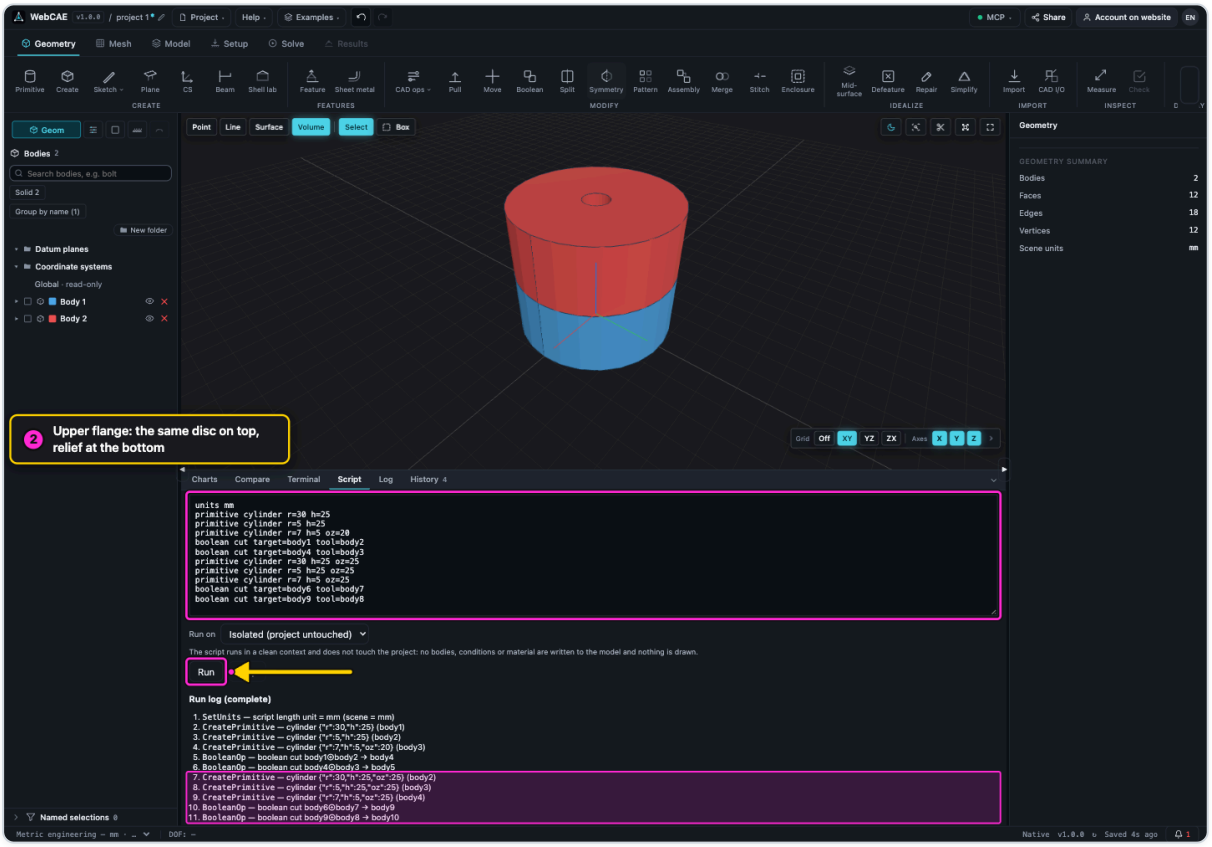
STEP 2

Upper flange: the same disc on top, relief at the bottom

The upper flange is created at $z = 25$ (`oz=25`) with the same bore and with the relief at $z = 25 \ldots 30$. The same two cuts give `body10`. Add this block at the END of the script from step 1 and click **Run**. The log lines of the three new cylinders print short labels (`body2`, `body3`, `body4`), but they are `body6`, `body7`, `body8` (the cuts name them correctly). Interface plane $z = 25$; the two flange bodies touch there and are glued by the assembly (a tied interface: common nodes in the mesh).

WHY Two separate bodies are the multi-body case of CAE prep: the connector has to couple them.

```
primitive cylinder r=30 h=25 oz=25
primitive cylinder r=5 h=25 oz=25
primitive cylinder r=7 h=5 oz=25
boolean cut target=body6 tool=body7
boolean cut target=body9 tool=body8
```



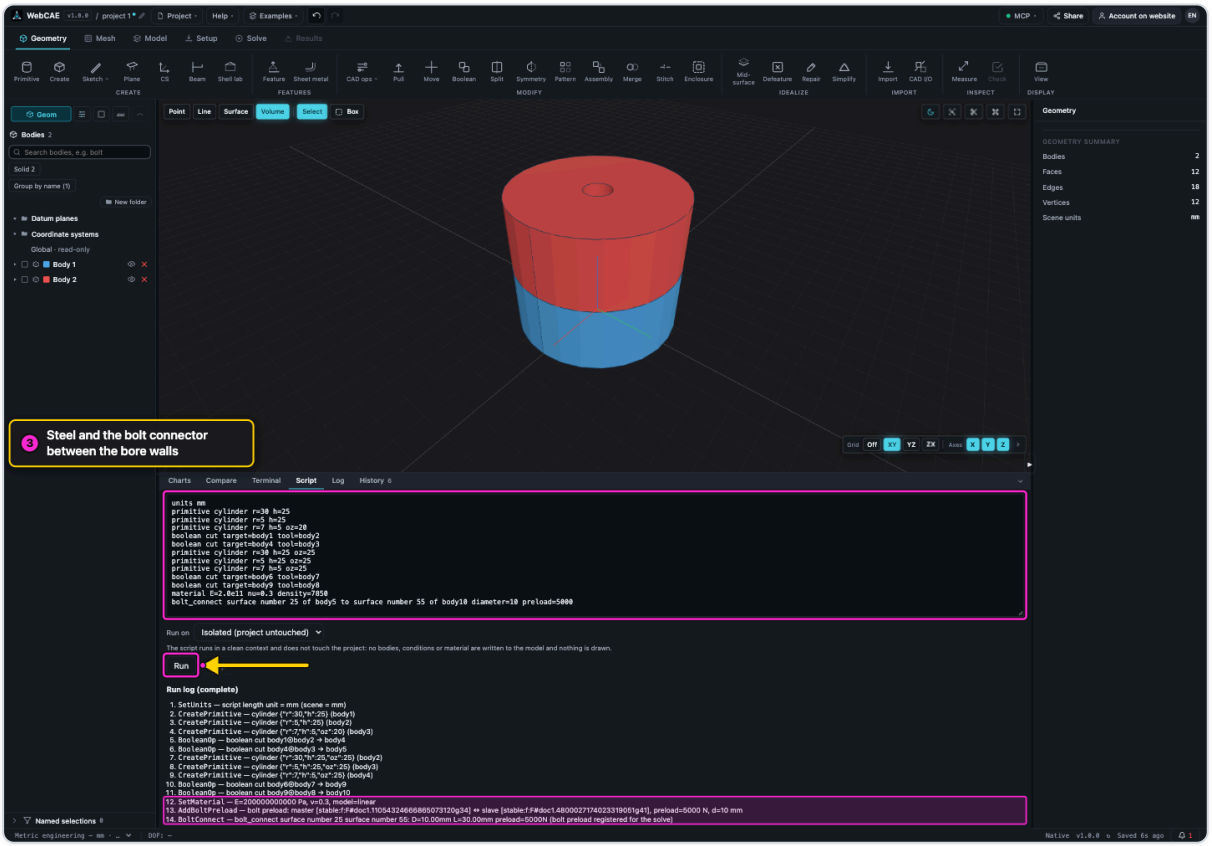
STEP 3 Steel and the bolt connector between the bore walls

Steel: $E = 2.0 \times 10^{11}$ Pa, $\nu = 0.3$. `bolt_connect surface number 25 of body5 to surface number 55 of body10 diameter=10 preload=5000` couples the cylindrical bore wall of the lower flange (`surface number 25` , centre at $z = 10$) with the bore wall of the upper flange (`surface number 55` , centre at $z = 40$): the connector is a beam of diameter 10 mm between rigid RBE2 spiders on the two walls, with a preload of $F_0 = 5000$ N. The bolt length between the spider centres is $L = 30$ mm. **The command needs cylindrical bore walls**; flat washer faces are refused ("holes are not coaxial"). Add the two lines at the END of the script and click **Run**: the log ends with `BoltConnect — bolt_connect surface number 25 surface number 55: D=10.00mm L=30.00mm preload=5000N (bolt preload registered for the solve)` .

The numbers of the two walls are the numbers of the script's own bodies for this exact command order and for **Run on: Isolated** (in Current model the product answers `Hole face surface number 25 not found on body body5`). To check them, add `list faces of body5` and `list faces of body10` at the end for one run: the bore wall is the face with area 628.3 mm^2 whose centroid is at $z = 10$ (`number=25`) and $z = 40$ (`number=55`); the same listing gives the bottom face (`number=3` , $z = 0$) and the top face (`number=32` , $z = 50$) used in step 4. Delete the two `list` lines afterwards.

> **WHY** With the reliefs, the two spiders do not share edge nodes at the interface plane; without them the bolt would carry no share of the load.

```
material E=2.0e11 nu=0.3 density=7850
bolt_connect surface number 25 of body5 to surface number 55 of body10 diameter=10 preload=
```



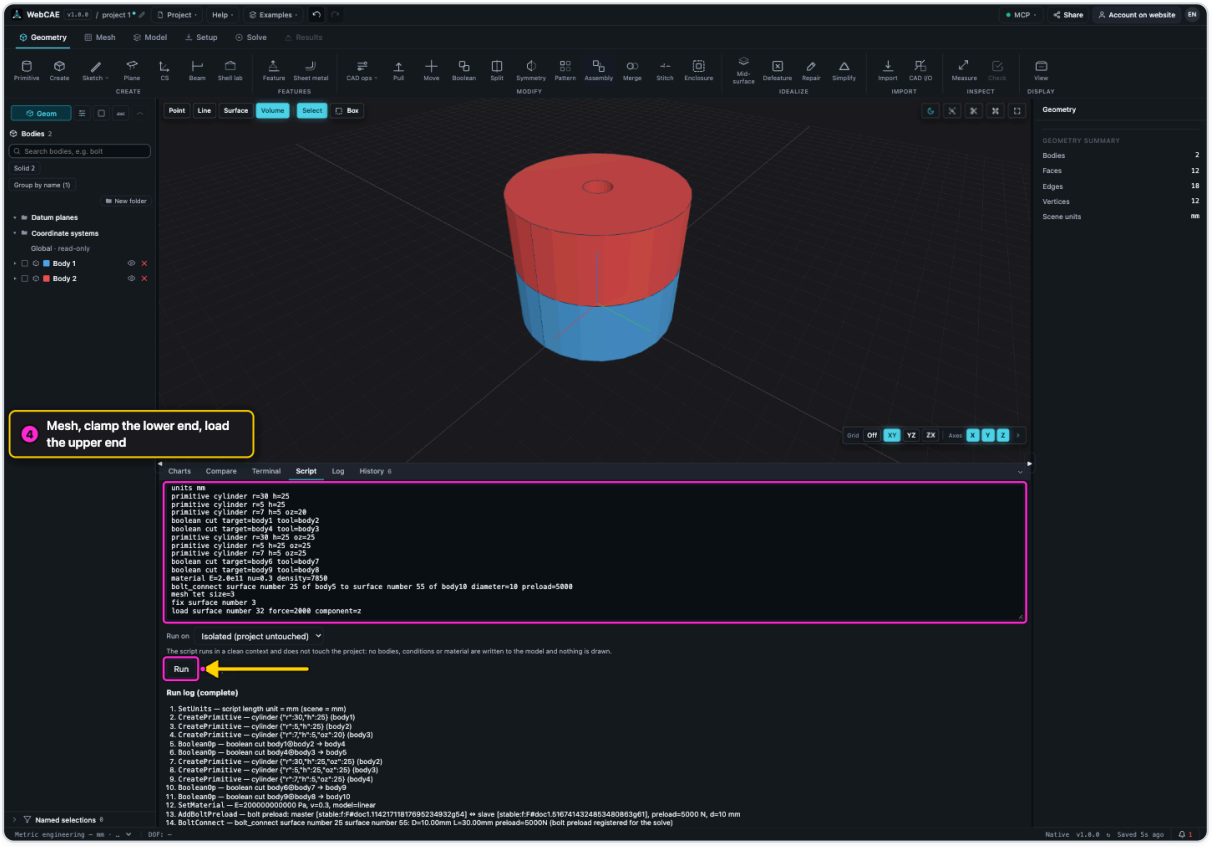
STEP 4

Mesh, clamp the lower end, load the upper end

`mesh tet size=3` gives about 3 mm elements (the result changes by ±2% with the mesh: 4 mm gave −5.2% on Φ, 2 mm −0.9%). `fix surface number 3` clamps the bottom face of the lower flange ($z = 0$); `load surface number 32 force=2000 component=z` pulls the top face of the upper flange ($z = 50$) with $P = 2000\ \mathrm{N}$ away from the clamp. Add the three lines at the END of the script and click **Run**: the log gets `SetMeshType`, `AddBoundaryCondition` and `AddLoad` lines; the mesh itself is built in step 5.

WHY The external load enters at the outer ends, so it passes through the clamped stack and the bolt in parallel.

```
mesh tet size=3
fix surface number 3
load surface number 32 force=2000 component=z
```

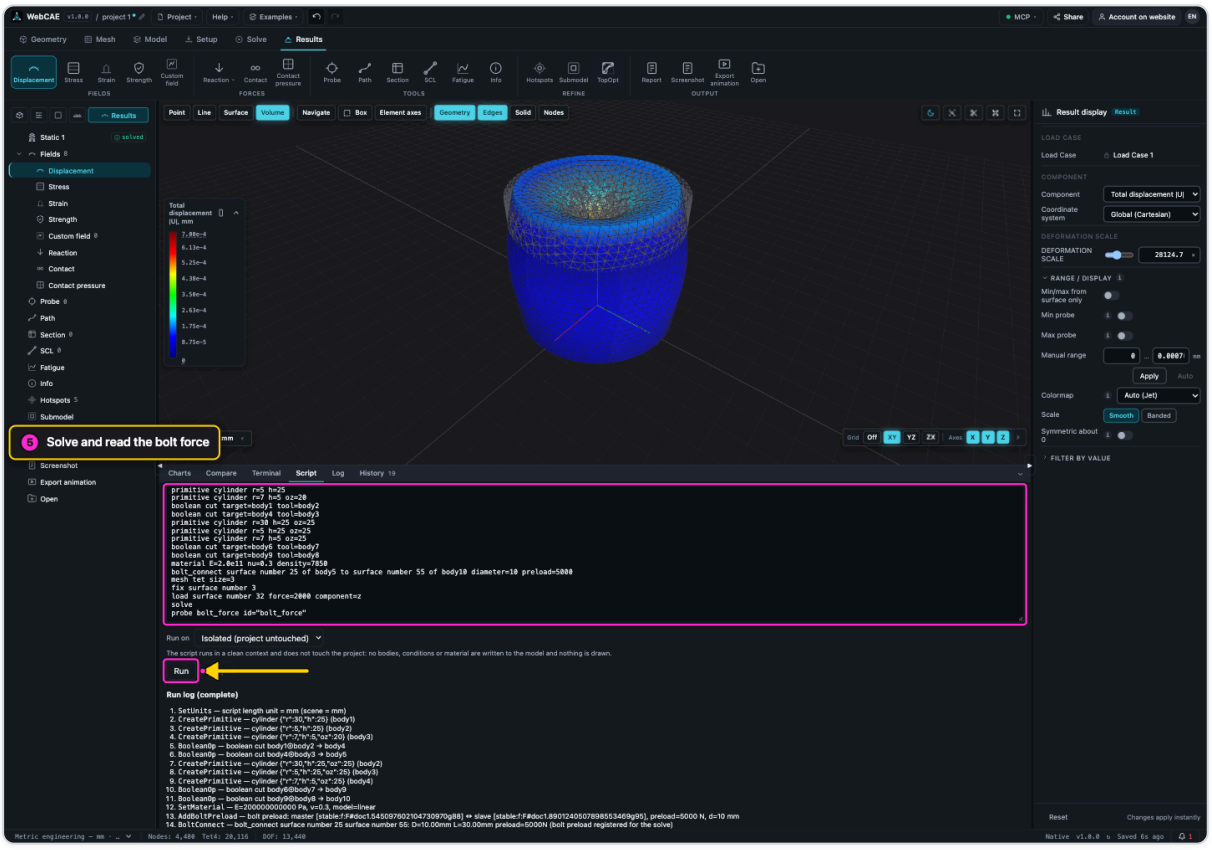


STEP 5 Solve and read the bolt force

`solve` runs the assembly branch: the two bodies are meshed with common interface nodes and solved together with the bolt (a sparse static solve). The log line of the solve names the bolt branch. `probe bolt_force` returns the axial force in the bolt: $5056.98\ \mathrm{N}$. Add the two lines at the END of the script (the editor now holds steps 1 to 5) and click **Run**. In the log look for `bolt preload calibrated: 1 bolt(s), F0 = 5000 N …` and `assembly bolt static (2 bodies, bonded interface, sparse path): … backend=…`; the result is the `bolt_force` row of **Measurements** (`5.0569e+3`). The browser (WASM) engine gives $5056.9\ \mathrm{N}$ (checked 2026-11-06, 0.002% from the native value) and runs the same bolt branch (20 353 tets); the run is two solves (the preload calibration and the loaded one) and takes from about 15 s on a free machine to about 50 s on a busy one.

> **WHY** The preload is calibrated so that the bolt really carries the requested force at zero external load (see the next step).

```
solve
probe bolt_force id="bolt_force"
```



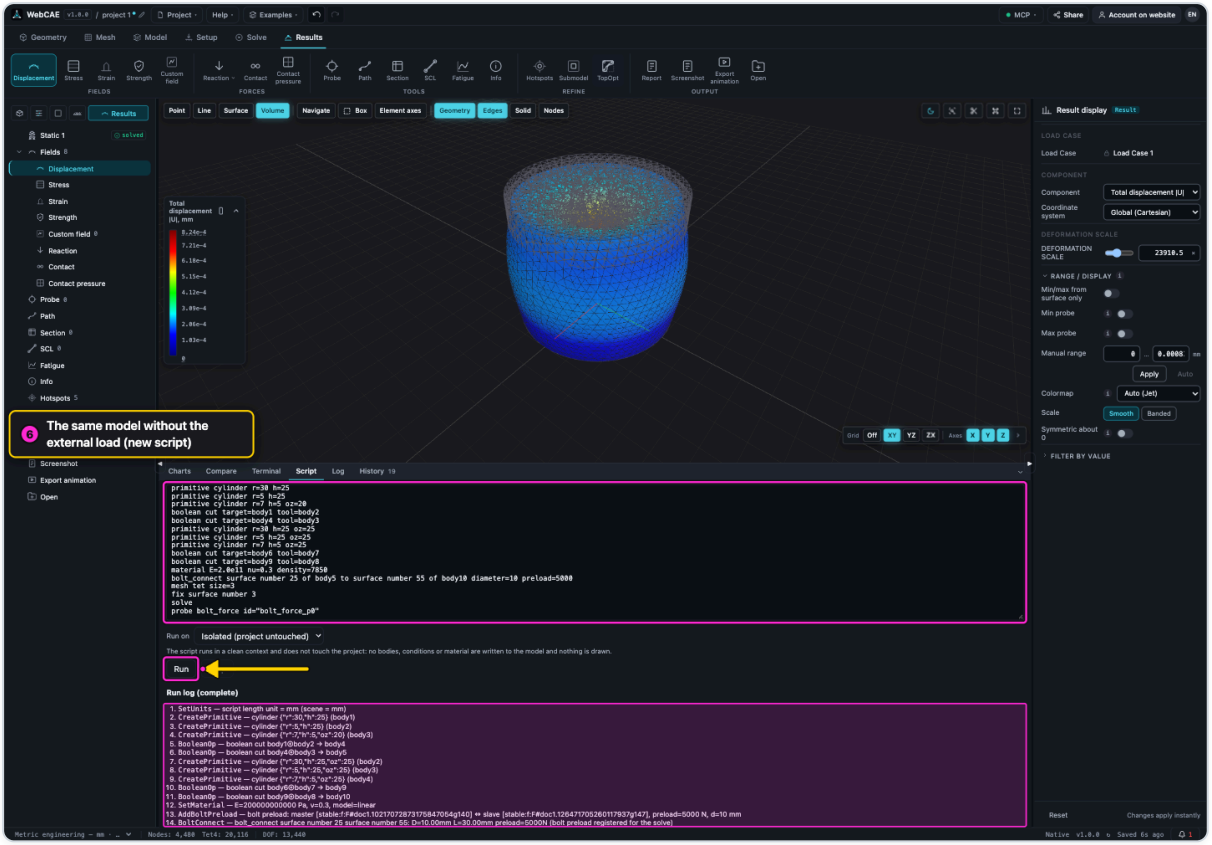
STEP 6

The same model without the external load (new script)

Replace the editor content with the complete script below (steps 1–4 **without the `load` line**, then solve). The log warns `No load entry … applied load is zero`: that is intended here. Result: the bolt carries $5000.00\ \mathrm{N}$, the preload F_0 itself (0.00%). The preload is calibrated: without calibration a raw preload of 5000 N produced only 4472.9 N in the bolt (the browser engine reports 4466.6 N in the `bolt preload calibrated` log line; the calibrated 5000.00 N is the same), because part of the preload goes into the local compliance of the spiders and the bore walls.

> **WHY** N(0) = F0 proves that the preload is what it says; only then is the change under load meaningful.

```
units mm
primitive cylinder r=30 h=25
primitive cylinder r=5 h=25
primitive cylinder r=7 h=5 oz=20
boolean cut target=body1 tool=body2
boolean cut target=body4 tool=body3
primitive cylinder r=30 h=25 oz=25
primitive cylinder r=5 h=25 oz=25
primitive cylinder r=7 h=5 oz=25
boolean cut target=body6 tool=body7
boolean cut target=body9 tool=body8
material E=2.0e11 nu=0.3 density=7850
bolt_connect surface number 25 of body5 to surface number 55 of body10 diameter=10 preload=
mesh tet size=3
fix surface number 3
solve
probe bolt_force id="bolt_force_p0"
```

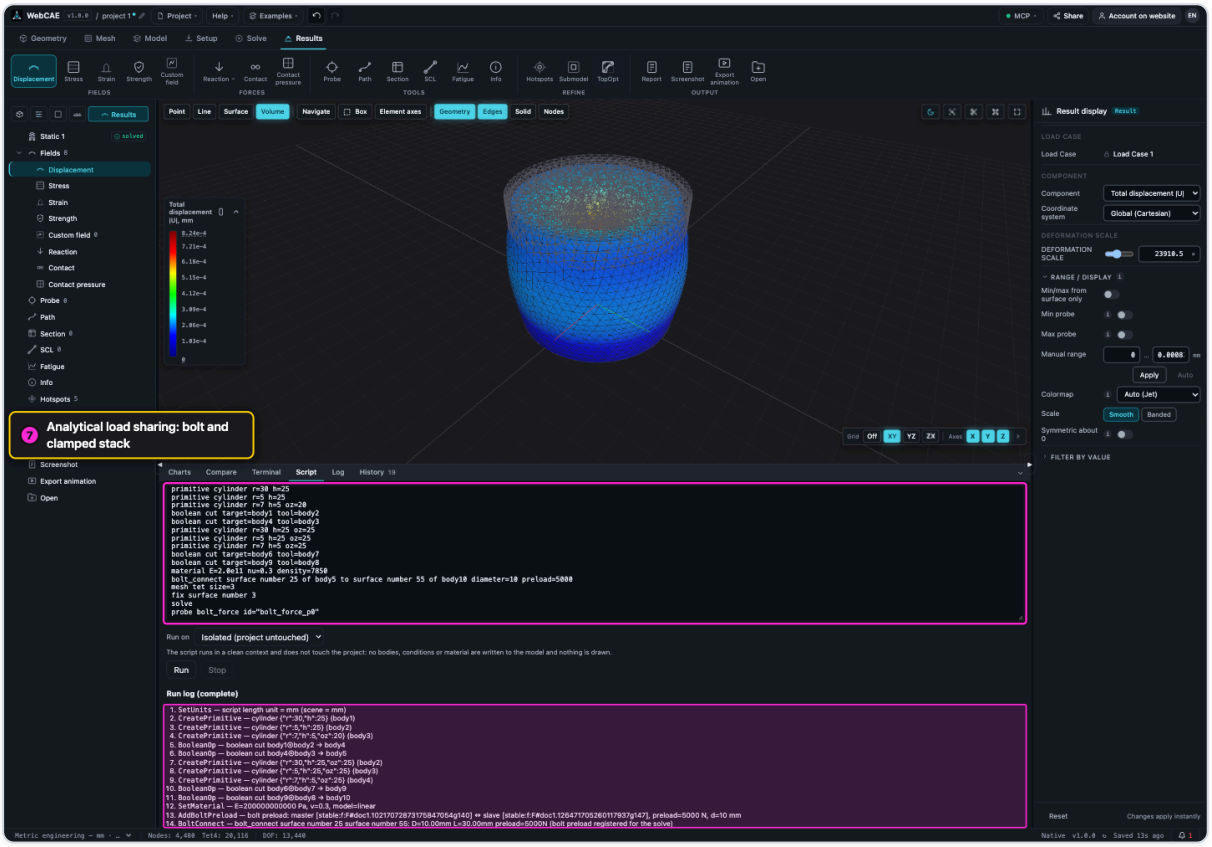

6 The same model without the external load (new script)
primitive cylinder r=30 h=25
primitive cylinder r=5 h=25
primitive cylinder r=7 h=5 oz=20
boolean cut target=body1 tool=body2
boolean cut target=body4 tool=body3
primitive cylinder r=30 h=25 oz=25
primitive cylinder r=5 h=25 oz=25
primitive cylinder r=7 h=5 oz=25
boolean cut target=body6 tool=body7
boolean cut target=body9 tool=body8
material E=2.0e11 nu=0.3 density=7850
bolt_connect surface number 25 of body5 to surface number 55 of body10 diameter=10 preload=5000
mesh_tet size=3
fix surface number 3
solve
probe bolt_force id="bolt_force_p0"
Run on Isolated (project untouched)
The script runs in a clean context and does not touch the project: no bodies, conditions or material are written to the model and nothing is drawn.
Run
Run log (complete)
1. SetUnits — script length unit = mm (scene = mm)
2. CreatePrimitive — cylinder {"r":30,"h":25} (body1)
3. CreatePrimitive — cylinder {"r":5,"h":25} (body2)
4. CreatePrimitive — cylinder {"r":7,"h":5,"oz":20} (body3)
5. BooleanOp — boolean cut body1@body2 → body4
6. BooleanOp — boolean cut body4@body3 → body5
7. CreatePrimitive — cylinder {"r":30,"h":25,"oz":25} (body2)
8. CreatePrimitive — cylinder {"r":5,"h":25,"oz":25} (body3)
9. CreatePrimitive — cylinder {"r":7,"h":5,"oz":25} (body4)
10. BooleanOp — boolean cut body6@body7 → body9
11. BooleanOp — boolean cut body9@body8 → body10
12. SetMaterial — E=200000000000 Pa, ν=0.3, model=linear
13. AddBoltPreload — bolt preload: master [stable:f:F#doc1.102170728731758470 54g140] ⇔ slave [stable:f:F#doc1.126471705260117937g147], preload=5000 N, d=10 mm
14. BoltConnect — bolt_connect surface number 25 surface number 55: D=10.00mm L=30.00mm preload=5000N (bolt preload registered for the solve)

STEP 7

Analytical load sharing: bolt and clamped stack

The bolt: $k_b = \frac{E_b A_b}{L} = \frac{2.1\times10^{11}\cdot\pi\cdot5^2\cdot10^{-6}}{0.030} = 5.498\times10^{8}\ \mathrm{N/m} = 549.8\ \mathrm{N/mm}$ (the bolt takes the steel default $E_b = 2.1\times10^{11}\ \mathrm{Pa}$). The clamped stack between the bolt supports ($z = 10\ldots40$) is a bar of three segments in series: two 10 mm segments of the flange wall with the bore, $A_1 = \pi(30^2 - 5^2) = 2748.9\ \mathrm{mm}^2$, and the middle segment $z = 20\ldots30$ with the two reliefs, $A_2 = \pi(30^2 - 7^2) = 2673.5\ \mathrm{mm}^2$: $k_m = \left[\frac{2\cdot0.010}{EA_1} + \frac{0.010}{EA_2}\right]^{-1} = 1.8155\times10^{10}\ \mathrm{N/m} = 18\,155\ \mathrm{N/mm}$. Then $\Phi = \frac{k_b}{k_b+k_m} = 0.02939$ and $N = F_0 + \Phi P = 5000 + 0.02939\cdot2000 = 5058.78\ \mathrm{N}$.

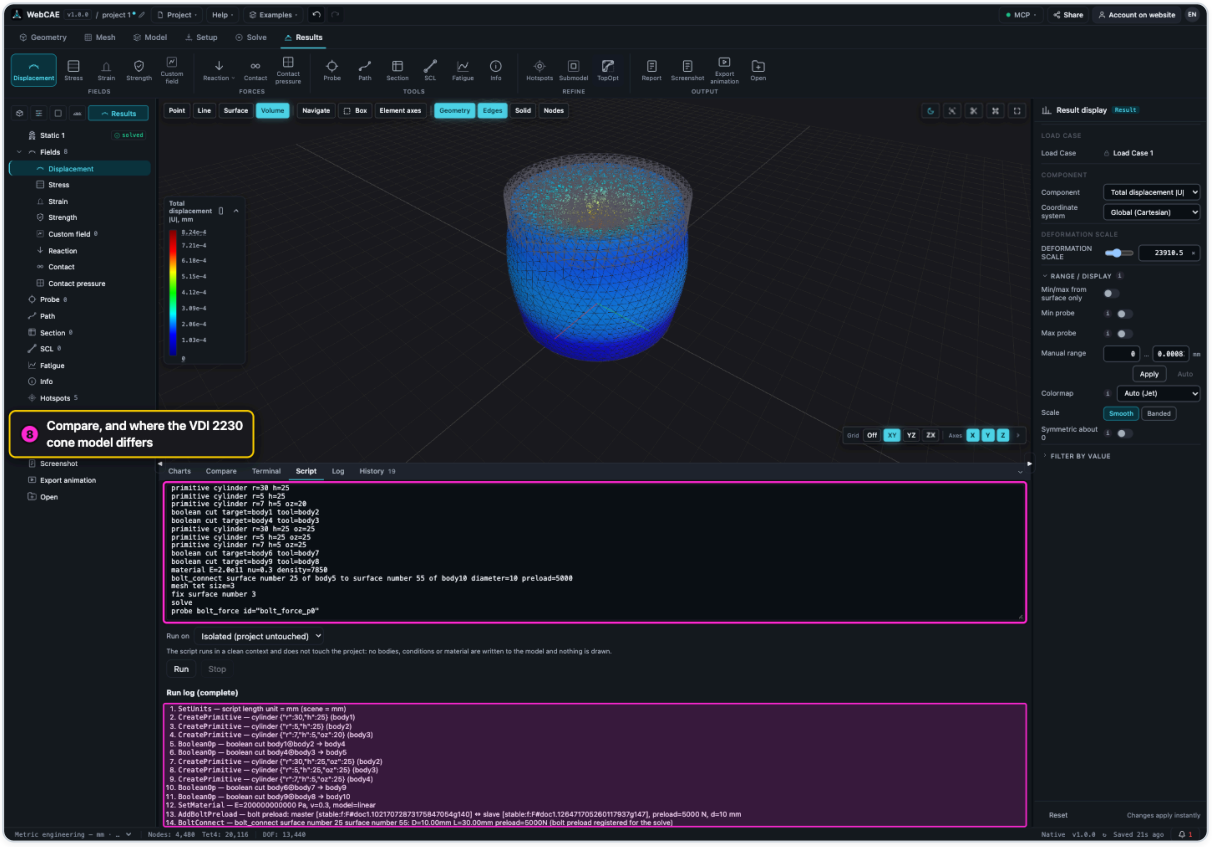
WHY The bolt is about 33 times more compliant than the stack, so it picks up only about 3% of the external load.



STEP 8 Compare, and where the VDI 2230 cone model differs

$N(P) = 5056.98\ \mathrm{N}$ against $5058.78\ \mathrm{N}$ (–0.04%), $\Phi = (5056.98 - 5000.00)/2000 = 0.02849$ against 0.02939 (–3.07%, the mesh scatter is ±2%). For information only: the VDI 2230 frustum model of a joint with washers gives a much larger Φ (0.127 for a 50 mm joint in an earlier draft of this lesson), because there the external load enters near the bolt head and the clamped zone is a cone under the head; here the load enters at the outer faces of the stack and the clamped zone is the whole bar, so the two numbers describe different joints and are **not** compared. A joint with washers and a plate load gave a Φ 27% below this simple formula on a converged mesh, because the load enters over the outer ring and the bolt force over the inner one: the formula holds only when load and support act on the same section, as here.

WHY A load-sharing check is meaningful only when the analytical stiffness refers to the same load path as the model.



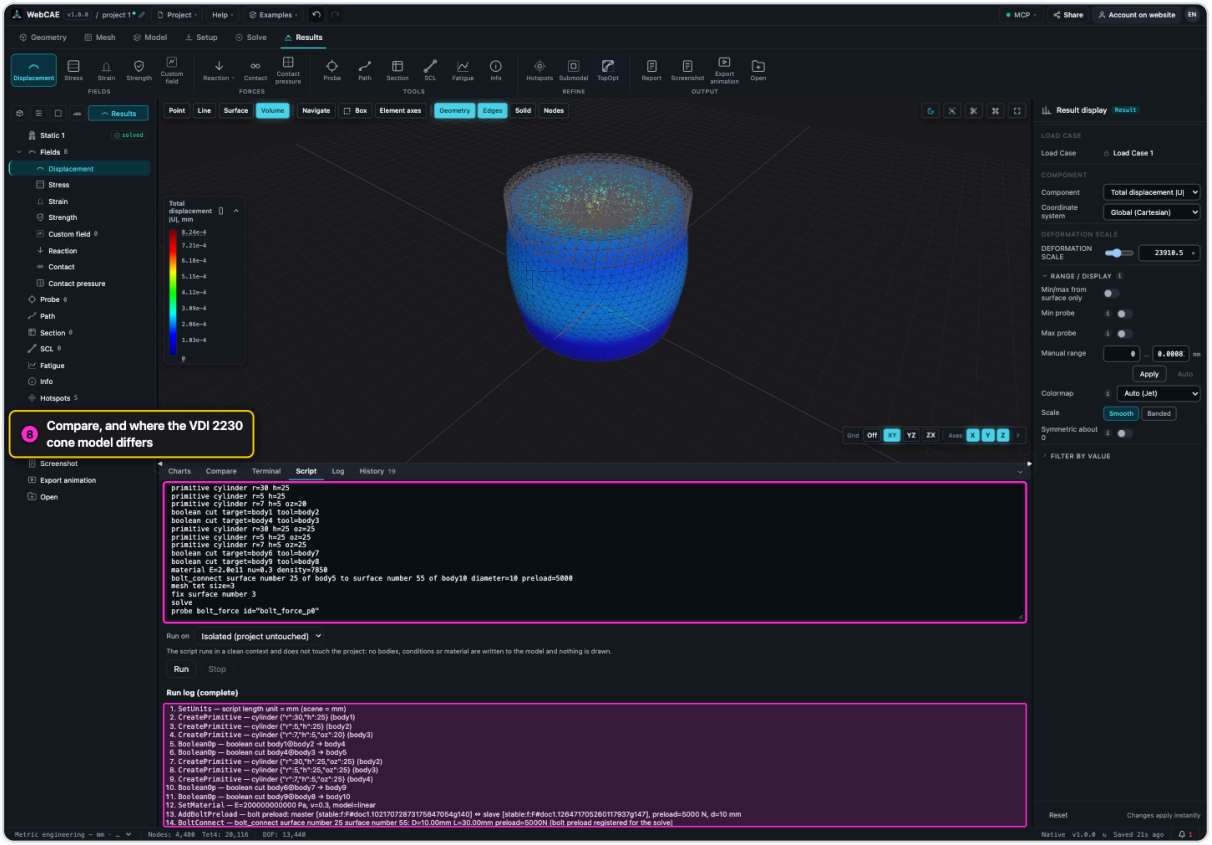
VERIFICATION BENCHMARK

Check against the formula

Quantity	Analytical	WebCAE (desktop/native)	Difference
Bolt force at $P = 0$ (preload $F_0 = 5000$ N)	5000 N	5000.00 N	0.00%
Bolt force at $P = 2000$ N	$F_0 + \Phi P = 5058.78$ N	5056.98 N	-0.04%
Load-sharing factor $\Phi = \Delta N/P$ (3 mm mesh)	$\frac{k_b}{k_b+k_m} = \frac{549.8}{549.8+18155} = 0.02939$	0.02849	-3.07%

FORMULA

$$N = F_0 + \Phi P, \quad \Phi = \frac{k_b}{k_b+k_m}, \quad k_b = \frac{E_b A_b}{L} = 549.8\ \text{N/mm}, \quad k_m = \left[\sum_i \frac{\ell_i}{E A_i}\right]^{-1} = 18\,155\ \text{N/mm}$$



TROUBLESHOOTING & NOTES

Common mistakes

- **Giving `bolt_connect` flat washer faces:** it needs the cylindrical walls of two coaxial bores ("holes are not coaxial" otherwise); use `bolt surface number A surface number B …` for other faces.
- **No relief at the interface:** the two spiders would share nodes on the interface plane and the bolt would carry almost none of the external load (Φ came out zero).
- **Reading face numbers from another command order, or running in Current model:** `surface number 25`, `55`, `3` and `32` belong to this exact script in **Run on: Isolated**; read them with `list faces of body5` and `body10` after changing the order or the mode.
- **Running a block of steps 2 to 5 alone:** every run starts a fresh interpreter, so the earlier bodies are gone (`body "body6" not found`). Append the block to the one script; only step 6 is a script of its own.
- **Trusting a preload that was never checked:** run the model once without the external load; N(0) must equal the preload.
- **Combining the bolt with a non-tied contact on the assembly:** the bolt branch carries tied interfaces only and refuses other contact modes.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#23 — 3D Portal Frame with 1D Beams** to start structural analysis with beam elements.

WebCAE 101

LESSON 23 · F23

3D Portal Frame with 1D Beams: Profiles, Orientation, and N-Q-M Diagrams

Goal Build a steel portal frame from 1D Timoshenko beam elements, assign an I-beam cross section, apply a combined vertical and lateral load, show the internal force diagrams and verify axial forces, bending moments and sway against the analytical stiffness method.

Outcome A portal frame model ($H = 3000\ \mathrm{mm}$, $L = 4000\ \mathrm{mm}$, I-beam $200 \times 100 \times 5.5 \times 8$) with the girder midspan moment (analytical $12.83\ \mathrm{kN \cdot m}$ with shear deformation, 12.74 Euler-Bernoulli; WebCAE desktop/native: $12.78\ \mathrm{kN \cdot m}$, difference 0.36%) and the knee sway (analytical $2.62\ \mathrm{mm}$ with shear, 2.50 Euler-Bernoulli; WebCAE: $2.553\ \mathrm{mm}$, difference 2.56%).

8–9 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

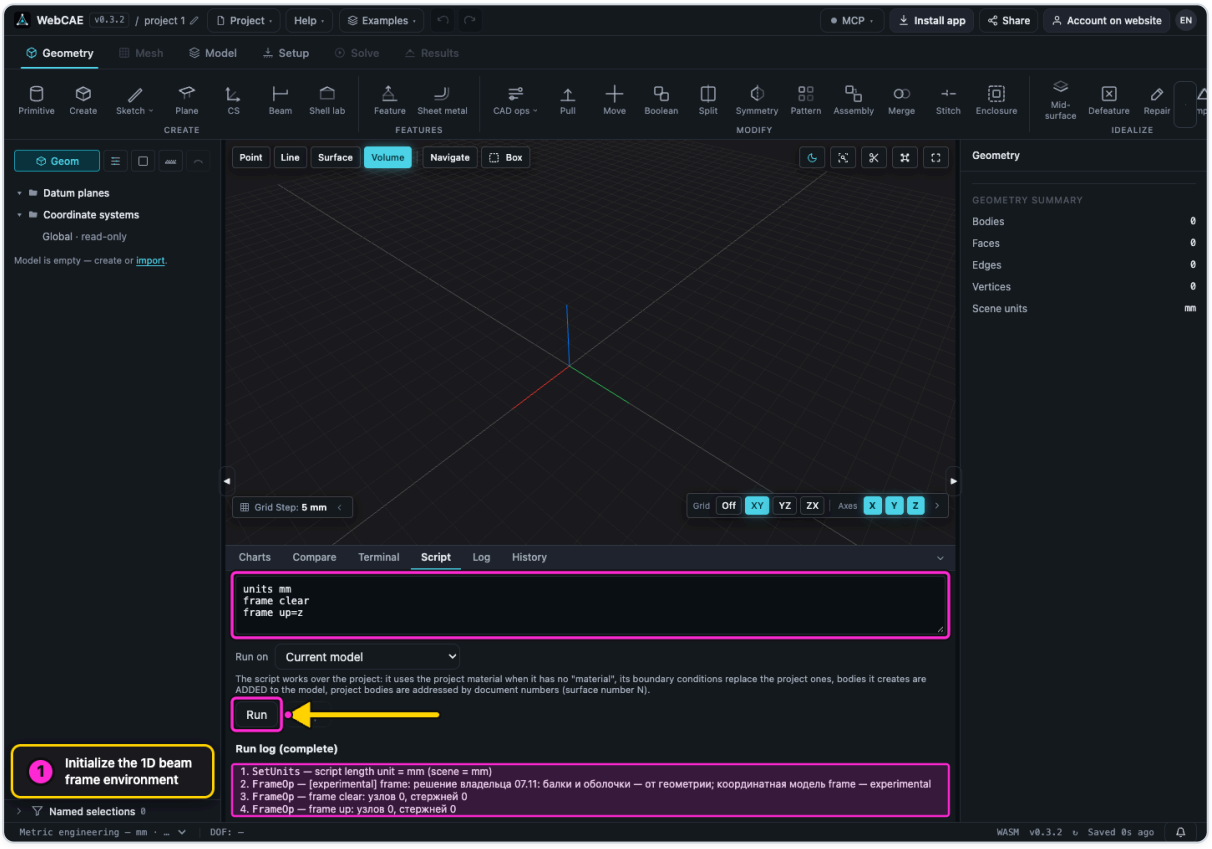
Run: 2026-11-04-32ffaed2 / F23

STEP 1 Initialize the 1D beam frame environment

Open the bottom dock, tab **Script**, set **Run on** to **Current model** and paste the code blocks (the **Frame** tile of the ribbon opens the same beam workspace). Set the units to millimeters, the vertical direction of the scene to Z (`frame up=z`) and clear any previous frame definition.

> **WHY** Explicit coordinate alignment makes the beam cross-sectional principal axes correspond to the horizontal and vertical flexural planes.

```
units mm
frame clear
frame up=z
```



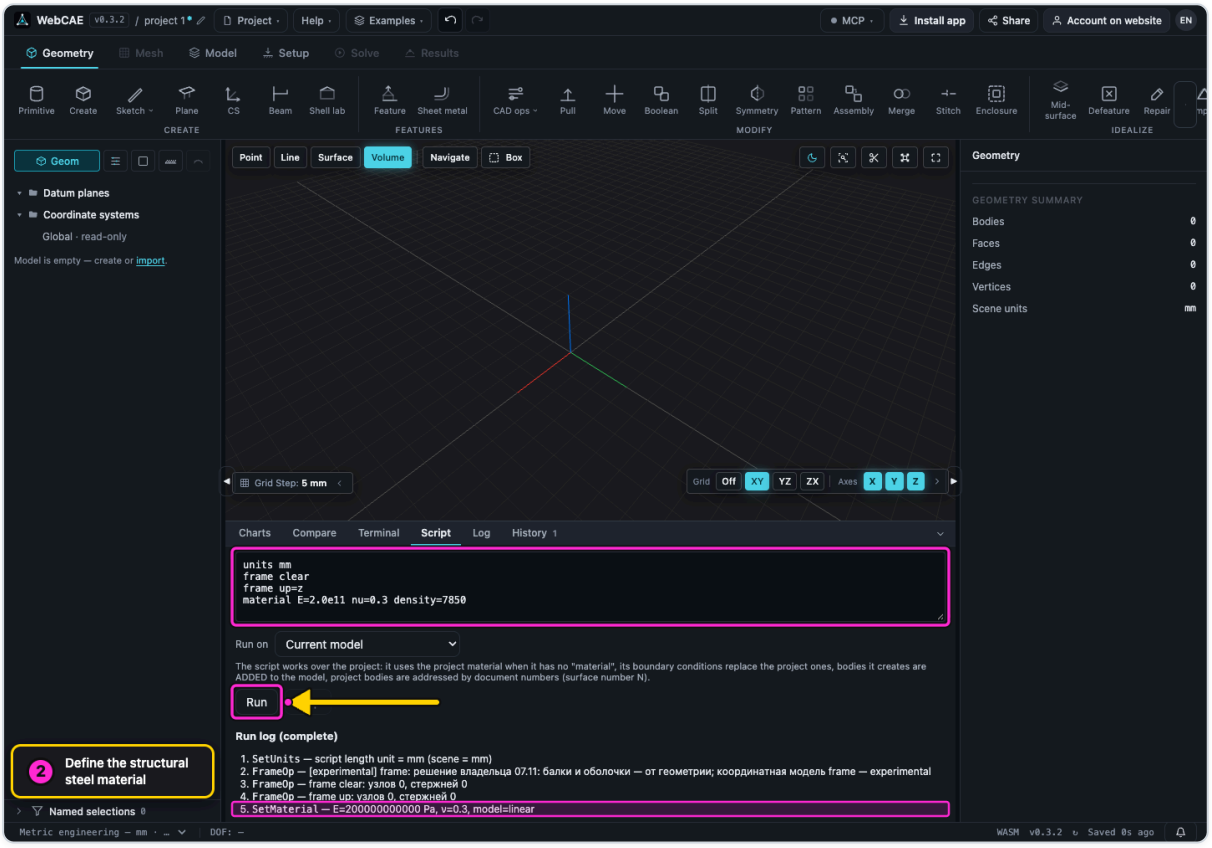
STEP 2

Define the structural steel material

Young's modulus $E = 200\,000\ \text{MPa} = 2.0 \times 10^{11}\ \text{Pa}$, Poisson's ratio $\nu = 0.30$, density $\rho = 7850\ \text{kg/m}^3$. In the DSL the modulus is entered in Pascals.

WHY Both flexural rigidity EI and axial stiffness EA govern the sway and the moment distribution of the frame.

```
material E=2.0e11 nu=0.3 density=7850
```



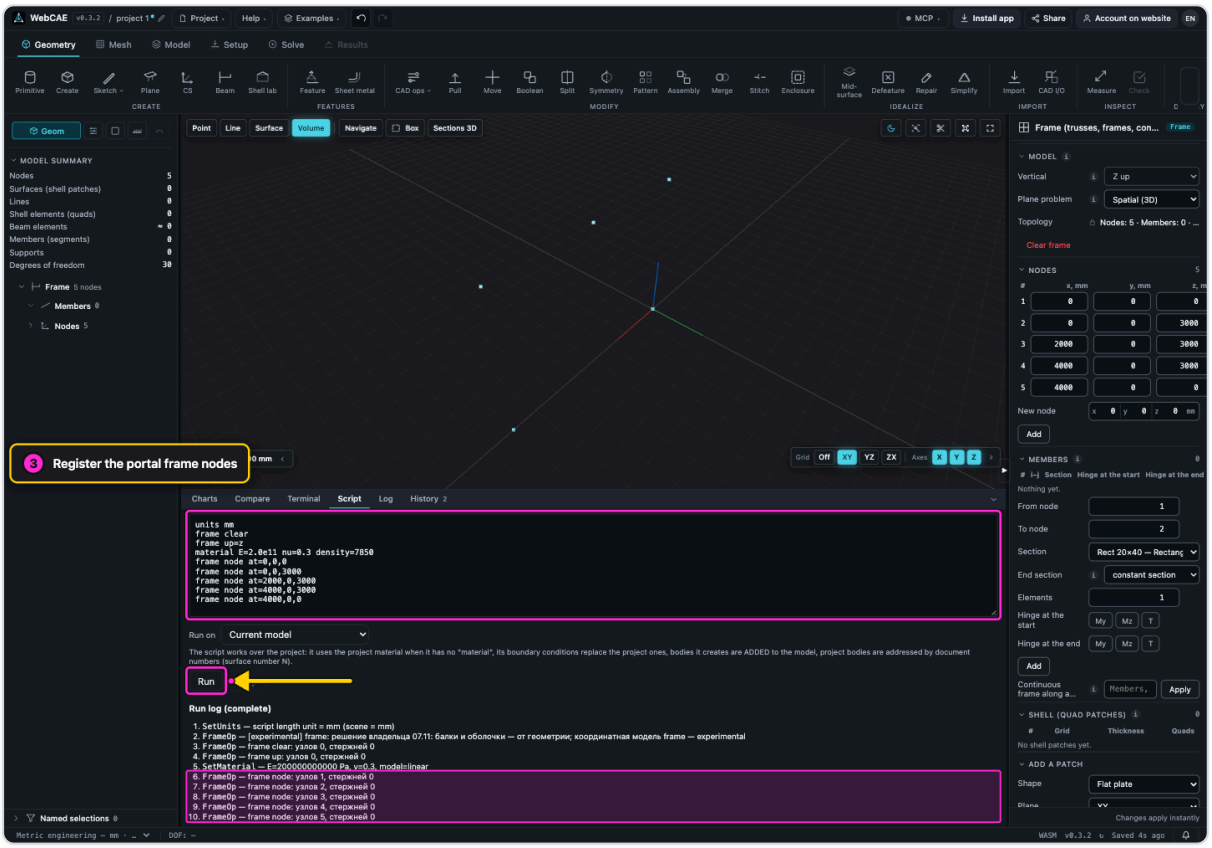
STEP 3

Register the portal frame nodes

Nodes are numbered 1 to 5 in the order of creation: left base $(0, 0, 0)$, left knee $(0, 0, 3000)$, midspan $(2000, 0, 3000)$, right knee $(4000, 0, 3000)$ and right base $(4000, 0, 0)$.

WHY An explicit node at midspan allows a direct point load and an exact reading of the peak moment without subdividing the girder.

```
frame node at=0,0,0
frame node at=0,0,3000
frame node at=2000,0,3000
frame node at=4000,0,3000
frame node at=4000,0,0
```

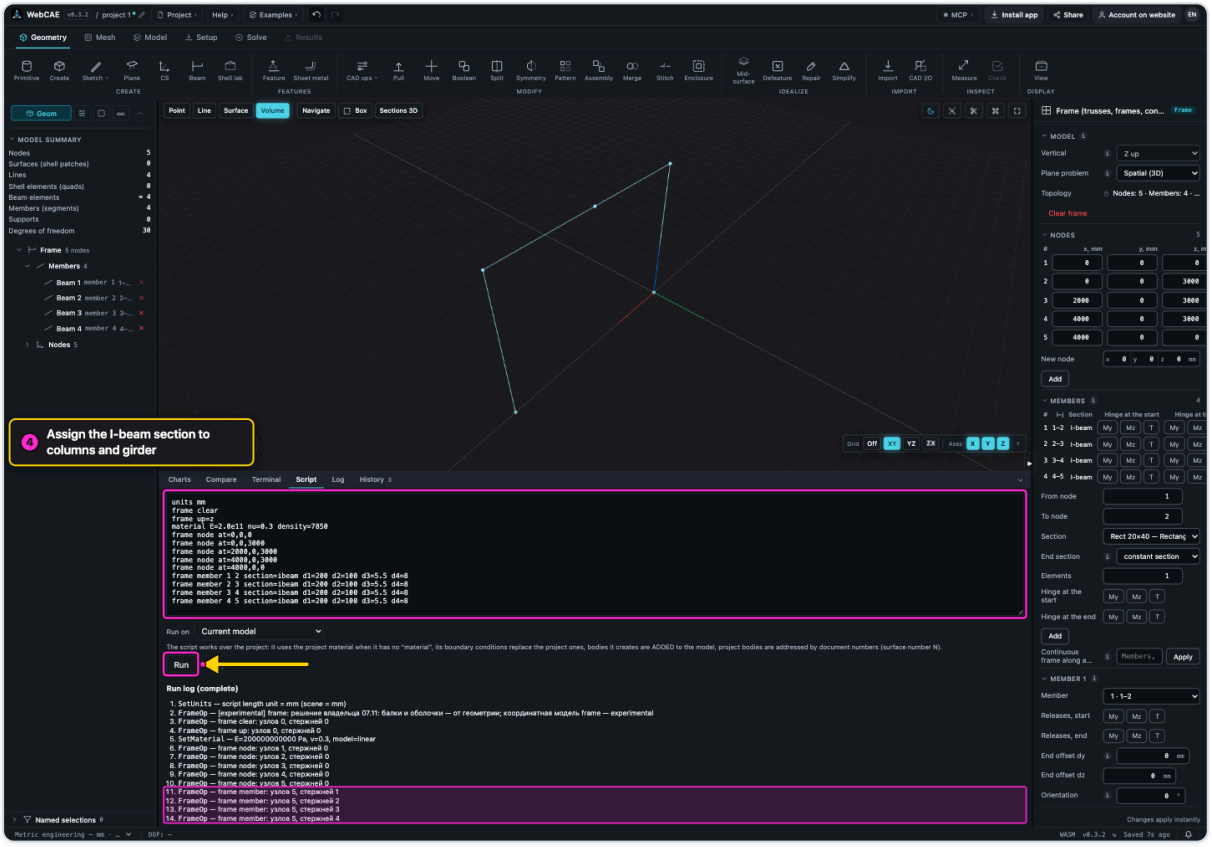


STEP 4 Assign the I-beam section to columns and girder

Connect the nodes with four members (numbered 1 to 4) using an I-beam profile: $d_1 = H = 200\ \text{mm}$, $d_2 = B = 100\ \text{mm}$, $d_3 = t_w = 5.5\ \text{mm}$, $d_4 = t_f = 8\ \text{mm}$. From the four dimensions the section area is $A = 2 \cdot 100 \cdot 8 + (200 - 16) \cdot 5.5 = 2612\ \text{mm}^2$, the strong-axis inertia $I = [100 \cdot 200^3 - (100 - 5.5) \cdot 184^3]/12 = 1761 \times 10^4\ \text{mm}^4$ and the web shear area $A_{sz} = (200 - 16) \cdot 5.5 = 1012\ \text{mm}^2$. This is an IPE 200-like profile without root fillets (the catalogue IPE 200, $2848\ \text{mm}^2$ and $1943 \times 10^4\ \text{mm}^4$, includes the fillets and is not used here). Members 1 and 4 are the columns, members 2 and 3 form the girder.

WHY The `ibeam` section takes $[H, B, t_w, t_f]$ and computes the area, the principal inertias and the torsion constant from them.

```
frame member 1 2 section=ibeam d1=200 d2=100 d3=5.5 d4=8
frame member 2 3 section=ibeam d1=200 d2=100 d3=5.5 d4=8
frame member 3 4 section=ibeam d1=200 d2=100 d3=5.5 d4=8
frame member 4 5 section=ibeam d1=200 d2=100 d3=5.5 d4=8
```

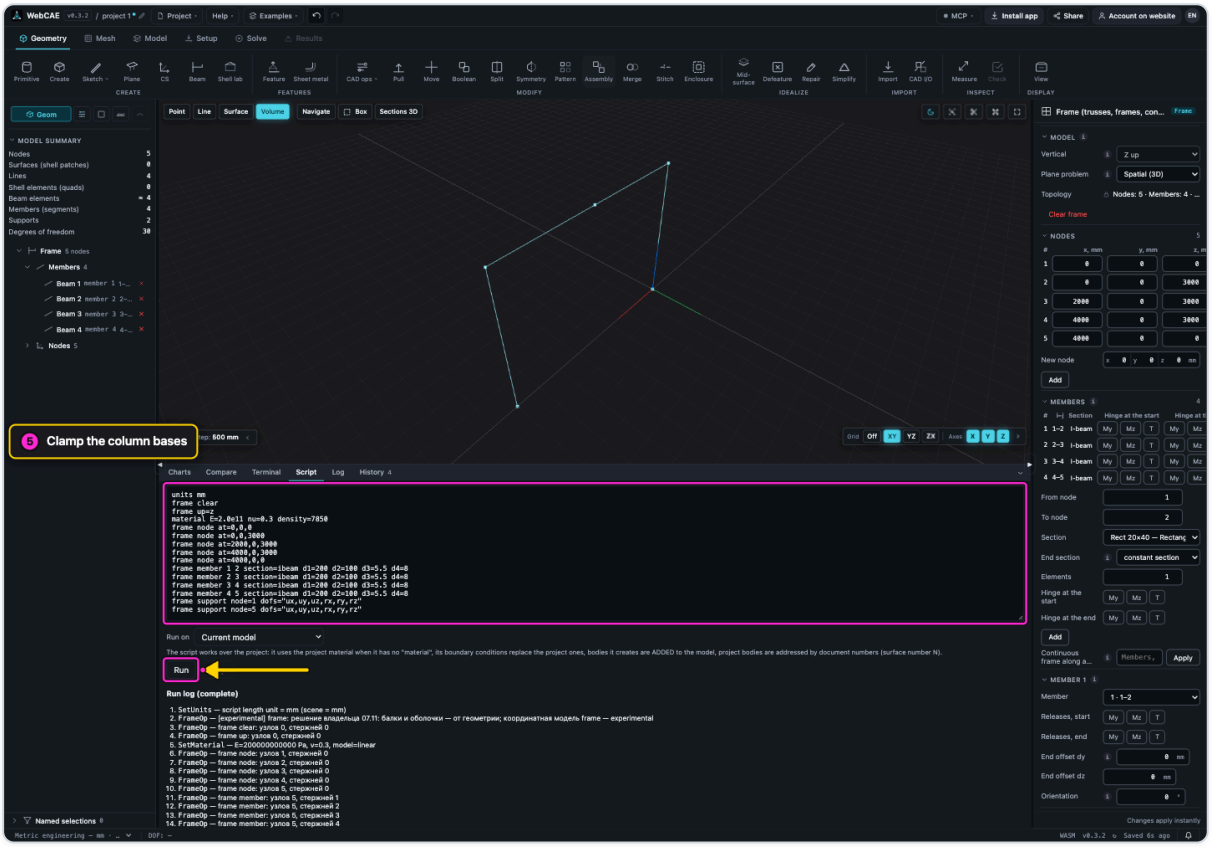


STEP 5 Clamp the column bases

Fix all six degrees of freedom of nodes 1 and 5 ($u_x, u_y, u_z, r_x, r_y, r_z = 0$).

WHY Fixed bases give moment restraint at the ground, which reduces the midspan sagging moment compared with pinned bases.

```
frame support node=1 dofs="ux,uy,uz,rx,ry,rz"
frame support node=5 dofs="ux,uy,uz,rx,ry,rz"
```

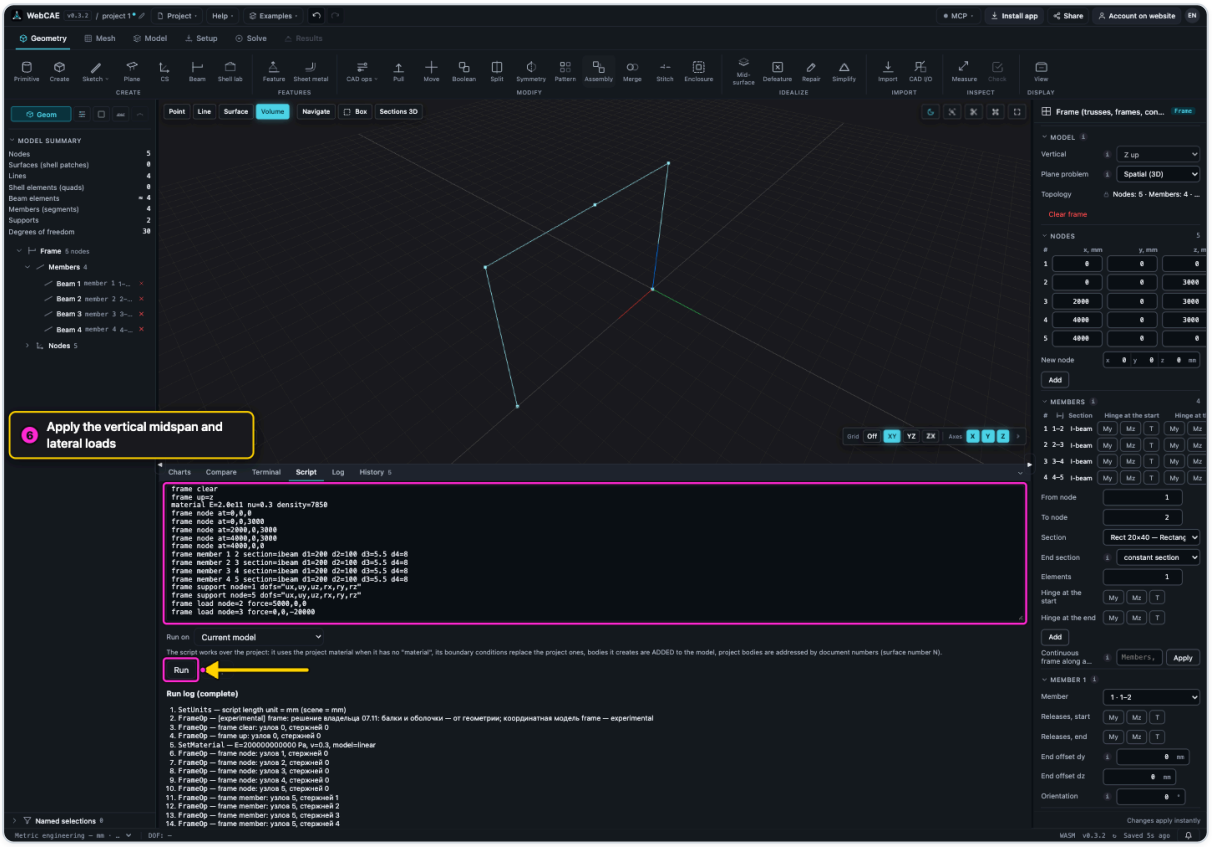


STEP 6 Apply the vertical midspan and lateral loads

A vertical load $F_z = -20\,000$ N at midspan node 3 and a lateral load $F_x = 5000$ N at the left knee node 2.

WHY Superimposed vertical and horizontal loads give an unsymmetric axial force pair in the columns and a combined bending state.

```
frame load node=2 force=5000,0,0
frame load node=3 force=0,0,-20000
```



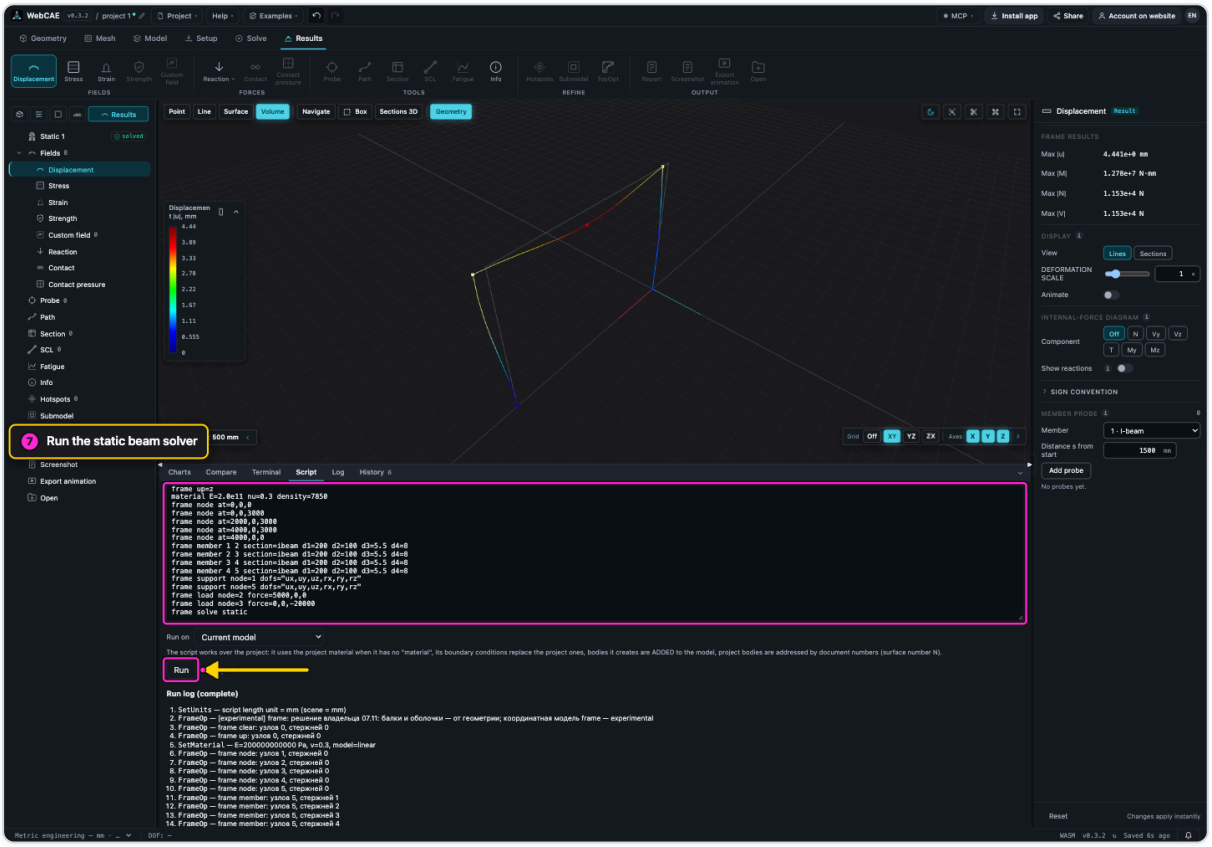
STEP 7

Run the static beam solver

`frame solve static` solves the global stiffness matrix with Timoshenko shear deformation included and opens the **Results** tab with the deformed shape.

> **WHY** The beam solver returns exact member force polynomials without discretization error and runs in milliseconds.

```
frame solve static
```



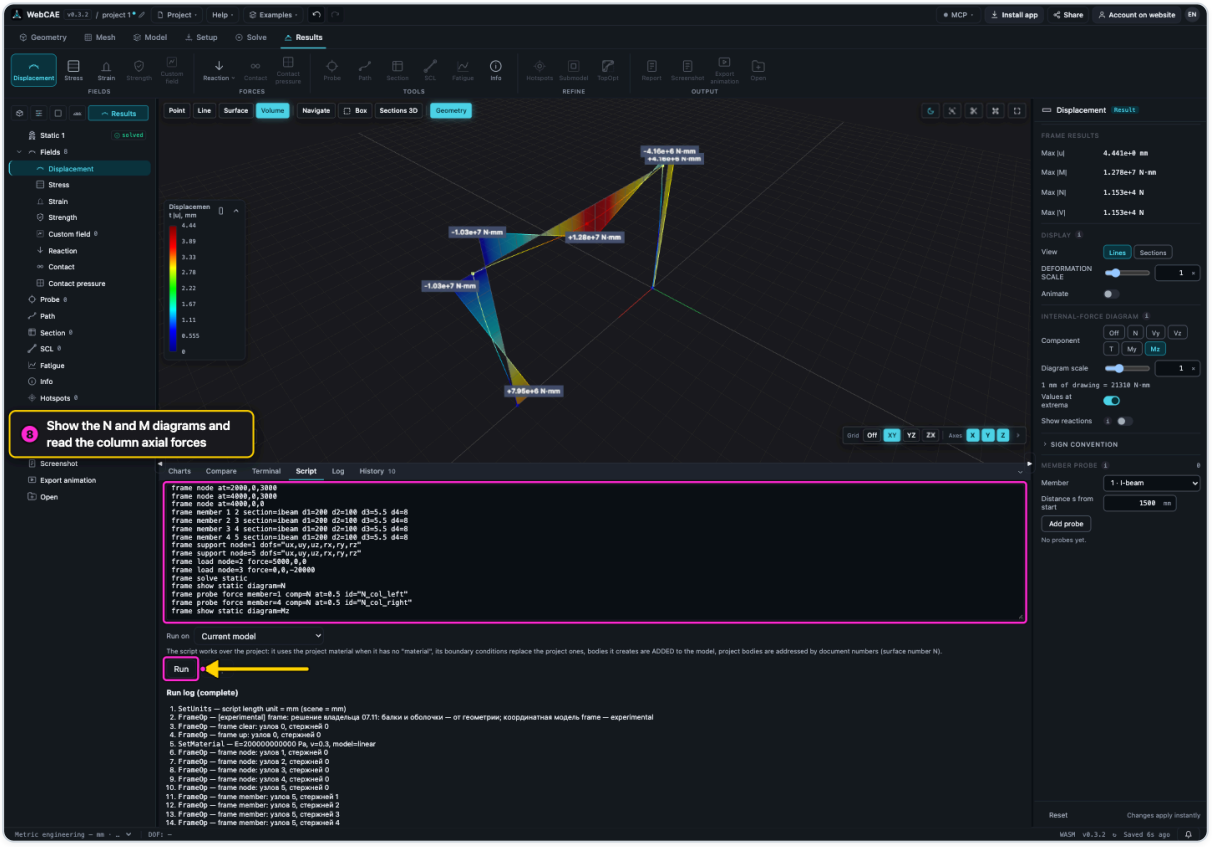
STEP 8 Show the N and M diagrams and read the column axial forces

`frame show static diagram=N` draws the axial force diagram in the viewport (axial force is positive in tension); `diagram=Mz` draws the bending moment on the tension side. With `frame up=z` the in-plane bending of this frame (the XZ plane) is the local component M_z; the other diagrams available are Vy, Vz, T and My.

Read the column forces at mid-height of members 1 and 4 with `frame probe force`: the left (windward) column carries $N_1 = -8.472\ \mathrm{kN}$ and the right (leeward) column $N_2 = -11.528\ \mathrm{kN}$ in compression (analytical -8.48 and $-11.52\ \mathrm{kN}$; differences 0.09% and 0.07%), summing to the applied $-20.00\ \mathrm{kN}$ ($-8.4724 - 11.5276 = -20.0000$).

WHY The lateral load produces an overturning couple that adds $\Delta N = \pm 1.52\ \mathrm{kN}$ to the symmetric vertical share of $-10.00\ \mathrm{kN}$ per column.

```
frame show static diagram=N
frame probe force member=1 comp=N at=0.5 id="N_col_left"
frame probe force member=4 comp=N at=0.5 id="N_col_right"
frame show static diagram=Mz
```



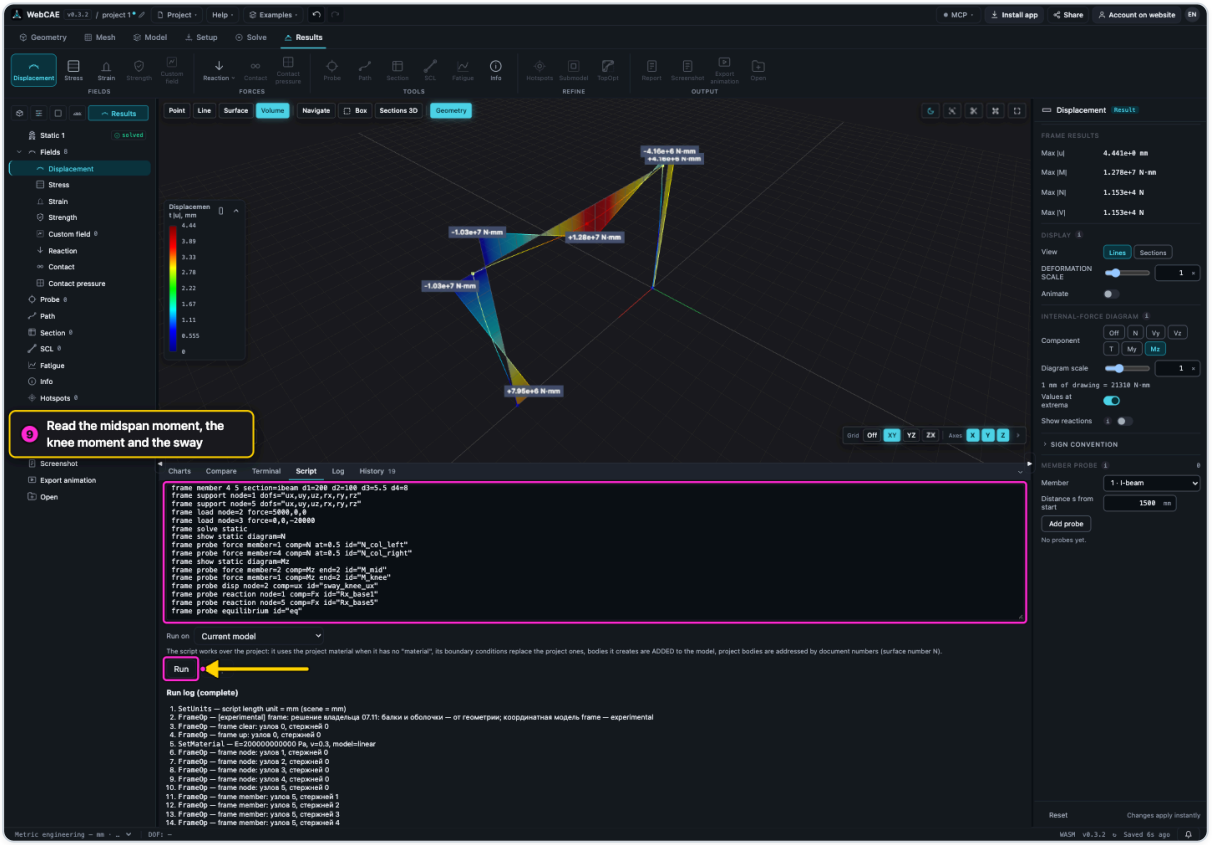
STEP 9 Read the midspan moment, the knee moment and the sway

Read the girder moment at node 3 (end 2 of member 2), the moment at the left knee (end 2 of member 1) and the knee sway u_x of node 2; check equilibrium with `frame probe equilibrium`:

- Midspan sagging moment: WebCAE $12.783\ \mathrm{kN \cdot m}$ (analytical 12.83 with shear deformation, 12.74 Euler-Bernoulli).
- Knee moment: $4.162\ \mathrm{kN \cdot m}$ (analytical 4.13).
- Knee sway: $2.553\ \mathrm{mm}$ (analytical 2.62 with shear, 2.50 Euler-Bernoulli; the product lies between the two).
- Equilibrium residual 3×10^{-15}; the base reactions $R_{x1} = 1074.3\ \mathrm{N}$ and $R_{x2} = -6074.3\ \mathrm{N}$ sum to $-5000\ \mathrm{N}$.

WHY The knee rigidity redistributes the sagging moment into hogging corner moments, which is what allows a shallower girder.

```
frame probe force member=2 comp=Mz end=2 id="M_mid"
frame probe force member=1 comp=Mz end=2 id="M_knee"
frame probe disp node=2 comp=ux id="sway_knee_ux"
frame probe reaction node=1 comp=Fx id="Rx_base1"
frame probe reaction node=5 comp=Fx id="Rx_base5"
frame probe equilibrium id="eq"
```



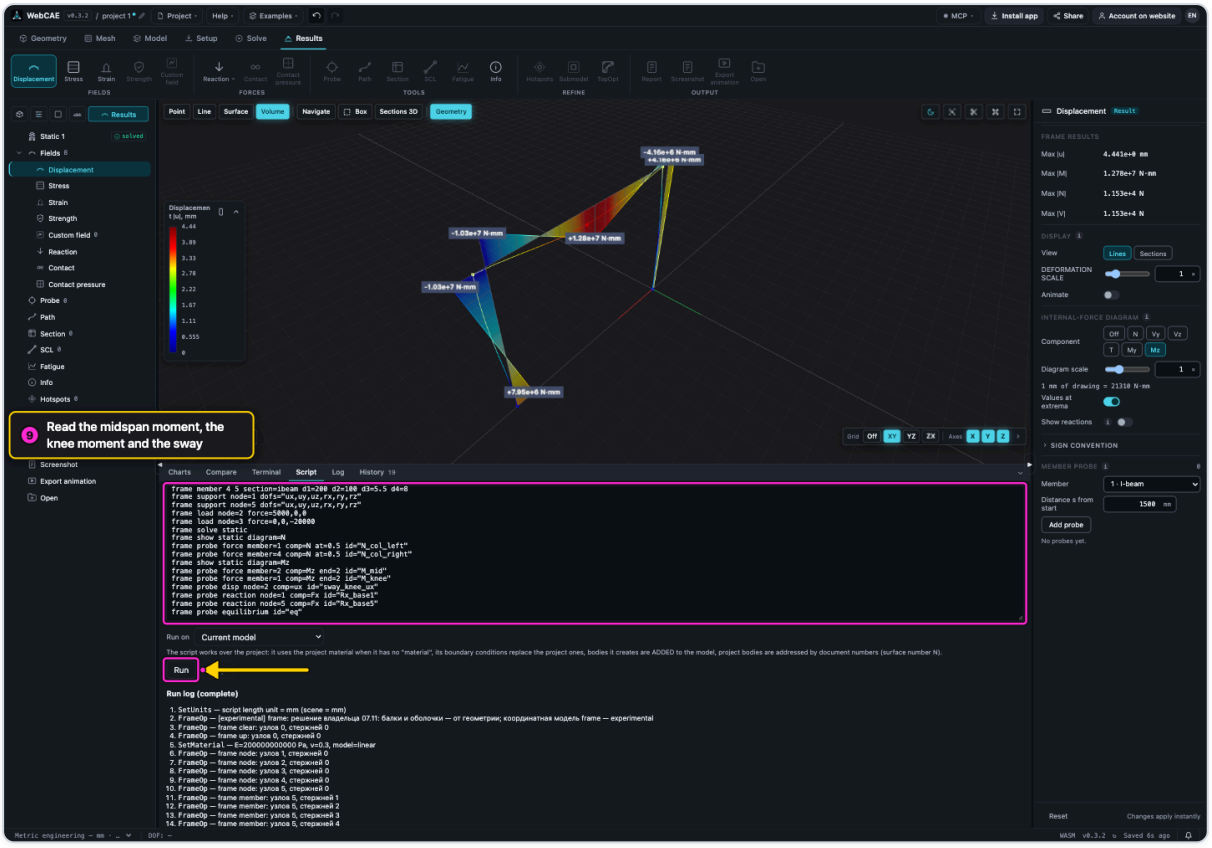
VERIFICATION BENCHMARK

Check against the formula

Quantity	Formula / Structural Mechanics	WebCAE (desktop/native, 1D beam)	Difference
Midspan bending moment M_z (node 3)	Stiffness method (Timoshenko, $A_{sz} = 1012\ \mathrm{mm}^2$): $12.83\ \mathrm{kN \cdot m}$ (Euler-Bernoulli 12.74)	12.783 kN·m	-0.36%
Lateral sway u_x at the knee (node 2)	Stiffness method (Timoshenko): $2.62\ \mathrm{mm}$ (Euler-Bernoulli $2.50\ \mathrm{mm}$)	2.553 mm	-2.56%
Windward column axial force N_1	$-10.00 + 1.52 = -8.48\ \mathrm{kN}$	-8.472 kN	-0.09%
Leeward column axial force N_2	$-10.00 - 1.52 = -11.52\ \mathrm{kN}$	-11.528 kN	+0.07%
Knee moment (node 2, member 1)	Stiffness method: $4.13\ \mathrm{kN \cdot m}$	4.162 kN·m	+0.77%

FORMULA

Matrix Stiffness Method for a Fixed-Base Portal Frame. For span $L = 4\ \mathrm{m}$, height $H = 3\ \mathrm{m}$, I-beam $200 \times 100 \times 5.5 \times 8$ ($A = 2612\ \mathrm{mm}^2$, $I = 1761 \times 10^4\ \mathrm{mm}^4$, $A_{sz} = 1012\ \mathrm{mm}^2$, $E = 200\,000\ \mathrm{MPa}$, $G = E/2.6$) under a vertical midspan load $P = 20\ \mathrm{kN}$ and a lateral joint load $F_x = 5\ \mathrm{kN}$, the direct stiffness solution with Timoshenko members gives column axial forces $N_1 = -8.48\ \mathrm{kN}$, $N_2 = -11.52\ \mathrm{kN}$ (summing to $-20.00\ \mathrm{kN}$), girder sagging moment $12.83\ \mathrm{kN \cdot m}$ and knee sway $2.62\ \mathrm{mm}$. Without shear deformation (Euler-Bernoulli) the same model gives $N = -8.47/-11.53\ \mathrm{kN}$, $12.74\ \mathrm{kN \cdot m}$ and $2.50\ \mathrm{mm}$; shear deformation adds 4.8% to the sway, so the beam formulation of the solver has to be stated when comparing.



TROUBLESHOOTING & NOTES

Common mistakes

- **Confusing the local bending component:** with `frame up=z` the in-plane bending of a frame lying in the XZ plane is the local component M_z (the older text of this lesson called it M_y); read M_z in `frame probe force` and `diagram=Mz`.
- **Omitting the midspan node:** a point load along a member without an intermediate node has to be distributed; node 3 at midspan gives an exact point-force coupling and the peak moment at a member end.
- **Unbalanced foundation fixity:** a column base that is accidentally pinned (`dofs="ux,uy,uz"`) drops the sidesway stiffness drastically and makes the moments highly asymmetric.
- **Comparing with the wrong formulation:** the sway differs by 4.8% between Euler-Bernoulli and Timoshenko beams; compare with the one that matches the solver (here the product lies between the two, at -2.56% from the Timoshenko value).

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#24 — Thin-Walled Pressure Vessel (F24)** to model shell elements and inspect membrane and bending stresses.

WebCAE 101

LESSON 24 · F24

Thin-Walled Pressure Vessel with Shell Elements: Membrane vs Bending Stress

Goal Simulate a cylindrical pressure vessel with 4-node MITC4 shell elements, separate the membrane hoop stress far from the ends from the local bending at a clamped end, and compare with the classical Laplace membrane and Love edge-effect solution.

Outcome A shell model ($R = 500\ \mathrm{mm}$, $t = 5\ \mathrm{mm}$, $L = 2000\ \mathrm{mm}$, $p = 1\ \mathrm{MPa}$) verifying the Laplace hoop stress (analytical $\sigma_\theta = 100.0\ \mathrm{MPa}$; WebCAE desktop/native: $99.95\ \mathrm{MPa}$, difference 0.05%), the radial expansion (analytical $\Delta R = 0.250\ \mathrm{mm}$; WebCAE: $0.2499\ \mathrm{mm}$, 0.05%), the edge-effect decay length (analytical $\lambda = \pi/\beta = 122\ \mathrm{mm}$; WebCAE: $123.5\ \mathrm{mm}$, 1.2%) and the axial bending stress at the clamp (analytical $137.9\ \mathrm{MPa}$ at the first element row; WebCAE: $138.8\ \mathrm{MPa}$, 0.68%).

9–10 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

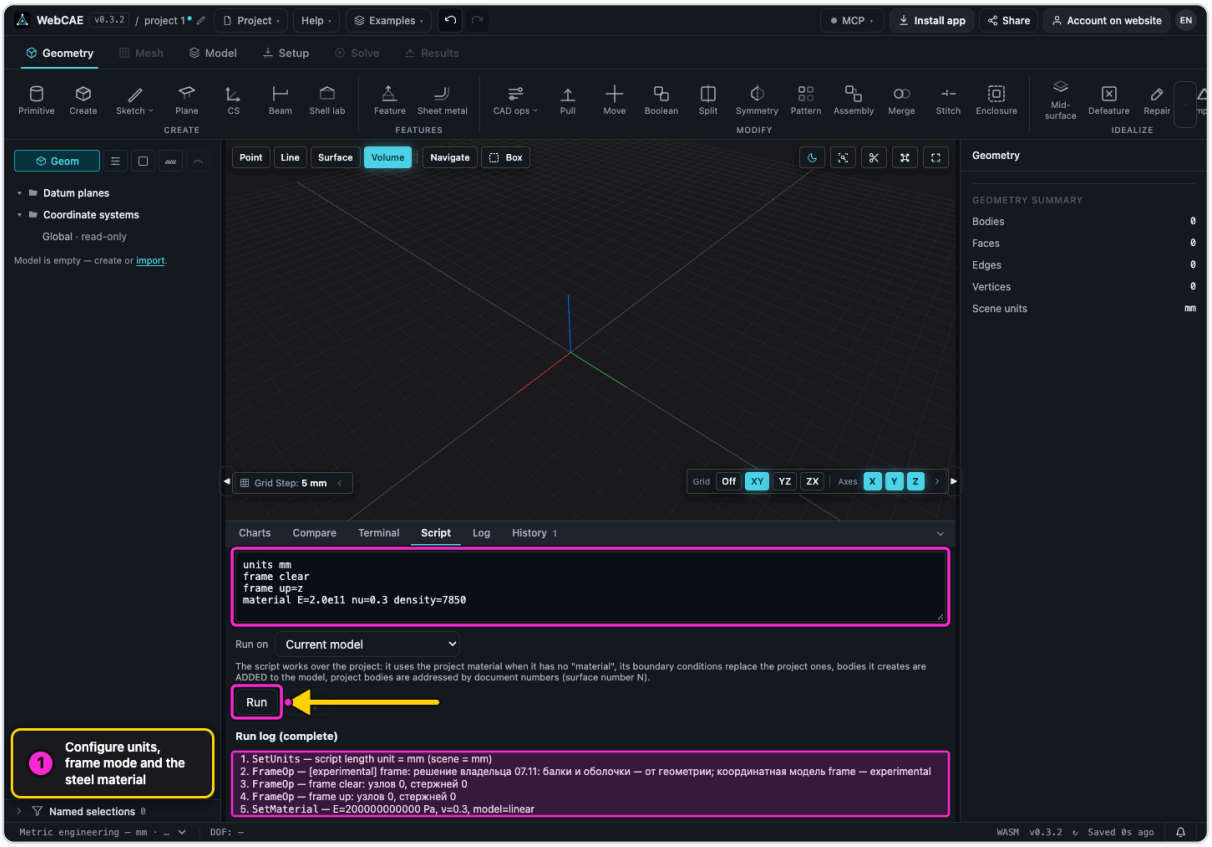
Run: 2026-11-04-32ffaed2 / F24

STEP 1 Configure units, frame mode and the steel material

Open the bottom dock, tab **Script**, set **Run on** to **Current model**, paste the code blocks of this lesson in order and click **Run** (the **Frame** tile opens the beam-and-shell workspace). Millimeters, vertical direction Z, and steel with $E = 2.0 \times 10^{11}\ \mathrm{Pa}$ (the DSL takes Pascals), $\nu = 0.3$.

> **WHY** MITC4 shell elements avoid shear locking in the thin-shell regime ($t/R = 1/100$) while keeping transverse shear deformation.

```
units mm
frame clear
frame up=z
material E=2.0e11 nu=0.3 density=7850
```



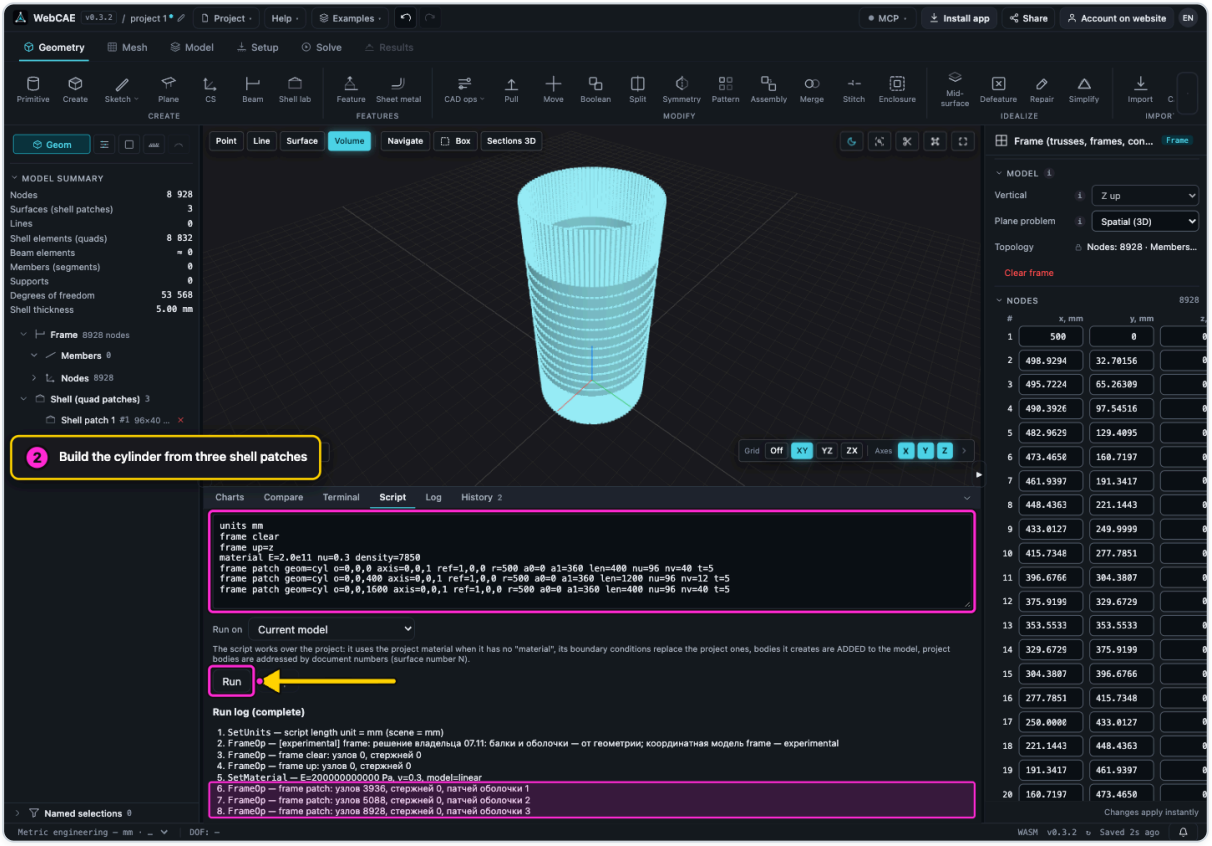
STEP 2 Build the cylinder from three shell patches

Create three cylindrical patches on shared nodes: $z = 0 \dots 400$ mm, $400 \dots 1600$ mm and $1600 \dots 2000$ mm, each with radius $R = 500$ mm, span $0° \dots 360°$ (the seam nodes are shared automatically) and thickness $t = 5$ mm. The mesh has $n_u = 96$ elements around the circumference and $n_v = 40, 12, 40$ along the axis, i.e. 10 mm elements in the two end zones (the edge effect decays over $\lambda = 122$ mm) and 100 mm elements in the middle.

The cylinder is split into three patches because `frame probe shell` returns one number per patch: the middle patch then gives a pure membrane value, far from both ends, and the first patch isolates the clamped end.

> **WHY** Splitting at $z = 400$ and $z = 1600$ puts the probe patch outside the edge-disturbance zone, which is about $3\lambda \approx 370$ mm at each end.

```
frame patch geom=cyl o=0,0,0 axis=0,0,1 ref=1,0,0 r=500 a0=0 a1=360 len=400 nu=96 nv=40 t=
frame patch geom=cyl o=0,0,400 axis=0,0,1 ref=1,0,0 r=500 a0=0 a1=360 len=1200 nu=96 nv=12
frame patch geom=cyl o=0,0,1600 axis=0,0,1 ref=1,0,0 r=500 a0=0 a1=360 len=400 nu=96 nv=40
```



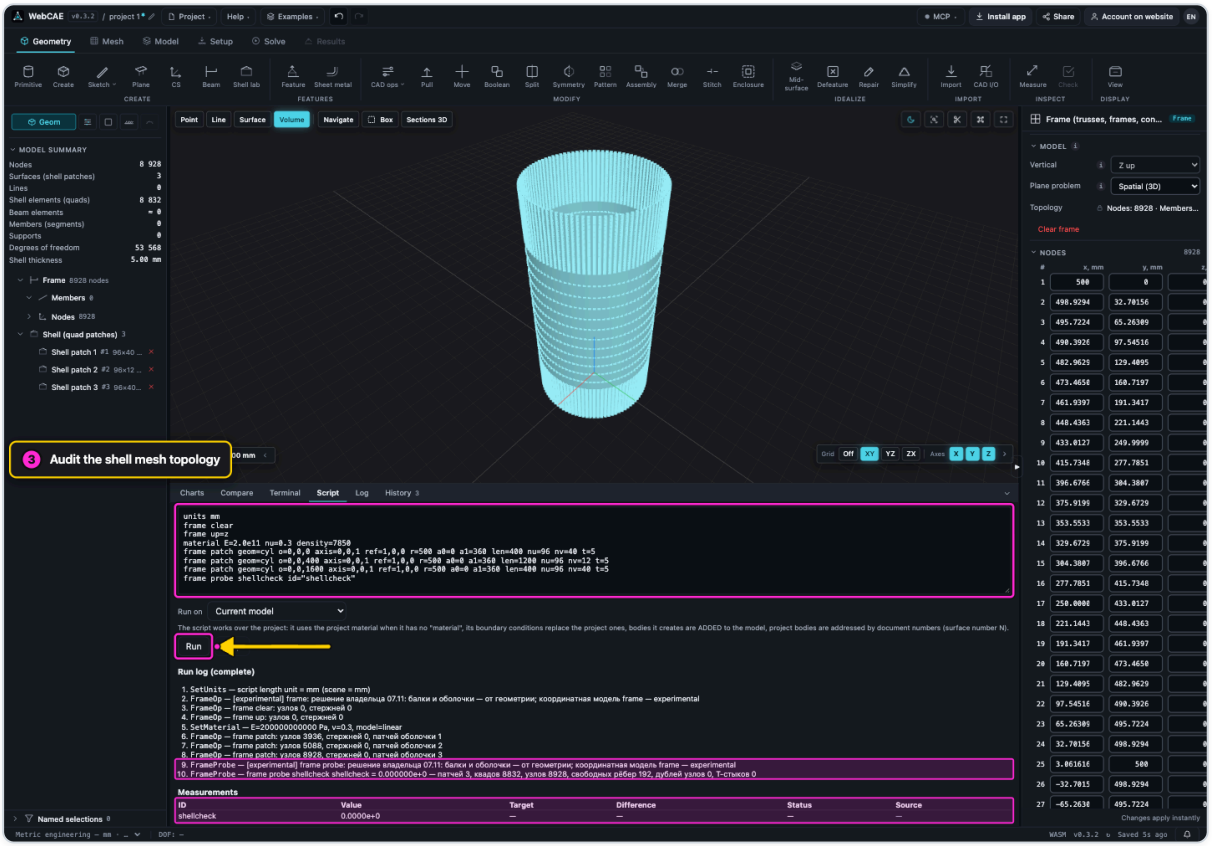
STEP 3

Audit the shell mesh topology

`frame probe shellcheck` counts the places where patches meet but are not joined by shared nodes. It must return 0. The log line reads: 3 patches, 8832 quads, 8928 nodes, 192 free edges (the two open rims, 2×96), 0 duplicate nodes, 0 T-junctions.

> **WHY** Unmerged nodes along a seam or patch interface create false free edges and spurious stress singularities while the run still reports success.

```
frame probe shellcheck id="shellcheck"
```



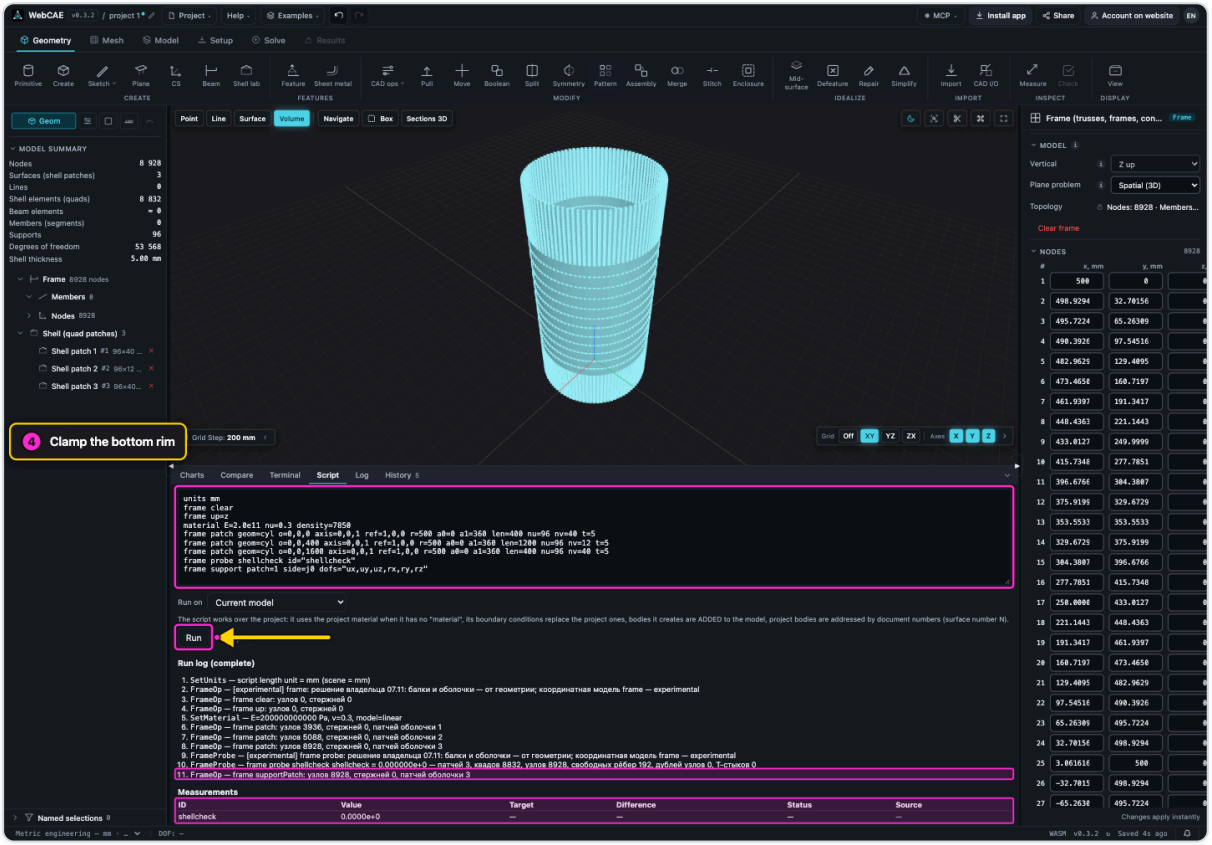
STEP 4

Clamp the bottom rim

Simulate a stiff forged head by clamping the first row of the first patch, the ring at $z = 0$ (`side=j0` of patch 1): all three translations and all three rotations are fixed. The top rim at $z = 2000$ stays free and open, so the pressure produces no axial end-cap force and $\sigma_z = 0$ far from the clamp.

WHY The clamp prevents radial growth of the ring, which creates an edge shear and bending moment that decay away from the end (Hetenyi, beam on an elastic foundation).

```
frame support patch=1 side=j0 dofs="ux,uy,uz,rx,ry,rz"
```



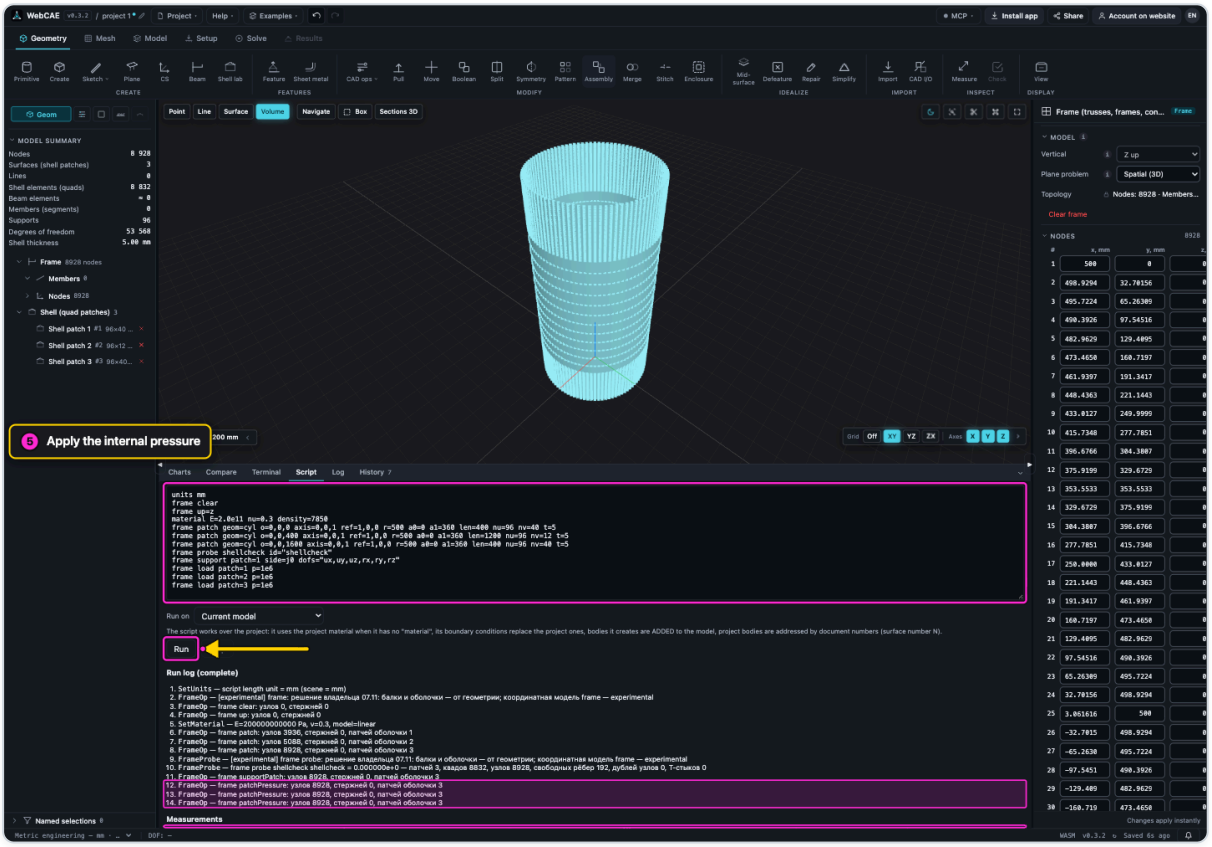
STEP 5

Apply the internal pressure

Apply $p = 1\ \mathrm{MPa} = 10^6\ \mathrm{Pa}$ on every patch (`p=` is in Pascals; the pressure acts along the shell normal $u \times v$, outward for the cylinder).

WHY The consistent nodal forces of a pressure load are distributed over the bilinear quad faces; all three patches must carry the same pressure.

```
frame load patch=1 p=1e6
frame load patch=2 p=1e6
frame load patch=3 p=1e6
```



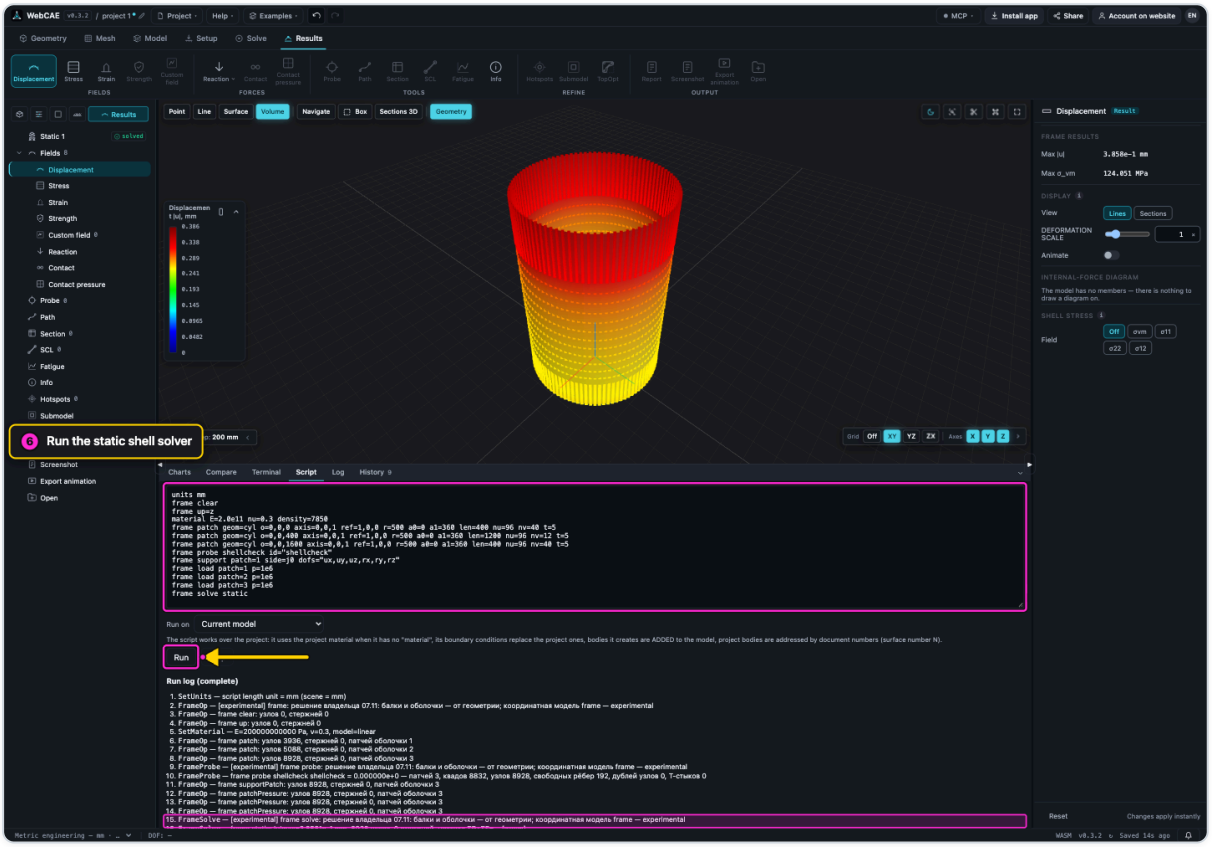
STEP 6

Run the static shell solver

`frame solve static` solves the shell model with six degrees of freedom per node.

WHY The six-degree-of-freedom formulation includes the drilling rotations and avoids spurious zero-energy modes in curved shell assemblies.

```
frame solve static
```



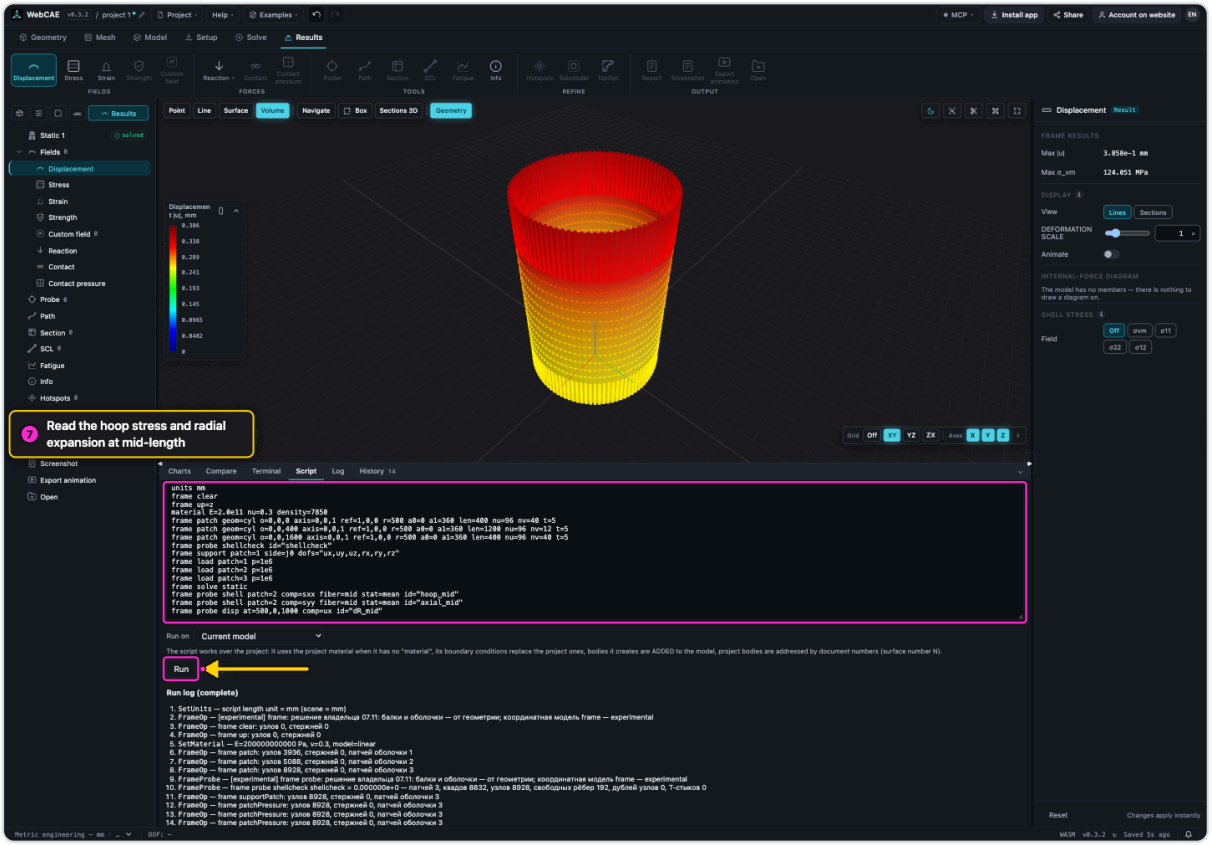
STEP 7 Read the hoop stress and radial expansion at mid-length

The probe takes the mean over the elements of the middle patch (mid-surface fiber). In the element frame of the cylinder patch `sxx` is the hoop component and `syy` the axial one. The Laplace value is $\sigma_\theta = pR/t = 100.0\ \mathrm{MPa}$; WebCAE reads $99.95\ \mathrm{MPa}$ (0.05%). The axial membrane stress `syy` of the middle patch is ≈ 0 (open ends).

The radial displacement at $(500, 0, 1000)$ is read by `frame probe disp ... comp=ux` (the node on the $+X$ generatrix, where the radial direction is X): the analytical dilation is $\Delta R = pR^2/(Et) = 0.250\ \mathrm{mm}$, WebCAE reads $0.2499\ \mathrm{mm}$ (0.05%).

WHY Far from the ends ($z \gg \pi/\beta$) the edge bending has decayed and pure membrane behavior dominates.

```
frame probe shell patch=2 comp=sxx fiber=mid stat=mean id="hoop_mid"
frame probe shell patch=2 comp=syy fiber=mid stat=mean id="axial_mid"
frame probe disp at=500,0,1000 comp=ux id="dR_mid"
```



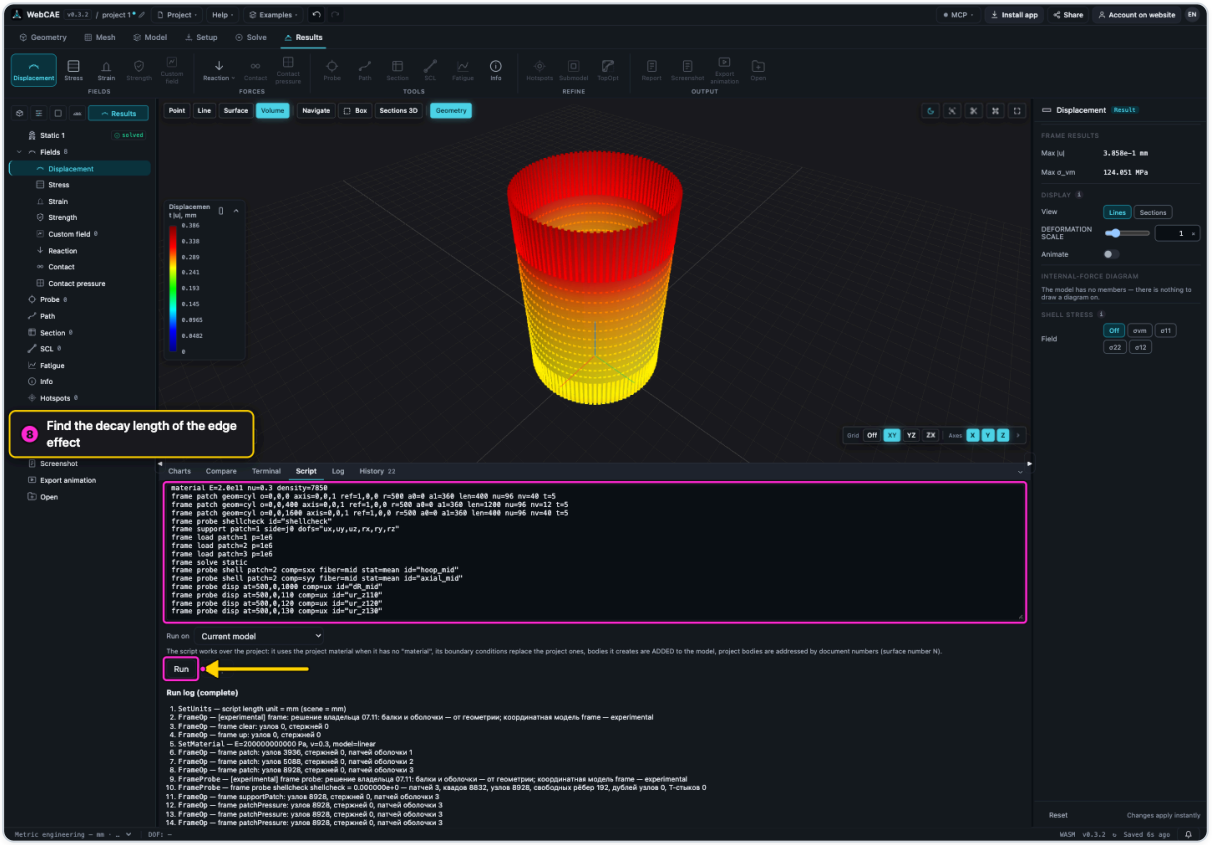
STEP 8 Find the decay length of the edge effect

Along the $+X$ generatrix the radial displacement overshoots the membrane value near the clamp and then settles. For a clamped edge the classical solution is $u_r(z) = \Delta R\,[1 - e^{-\beta z}(\cos\beta z + \sin\beta z)]$, with $\beta = [3(1-\nu^2)/(R^2 t^2)]^{1/4} = 0.02571\ \text{mm}^{-1}$; its maximum is at $\beta z = \pi$, i.e. at $\lambda = \pi/\beta = 122\ \text{mm}$, with the peak value $\Delta R\,(1 + e^{-\pi}) = 0.2608\ \text{mm}$.

Probe u_r at $z = 110, 120, 130\ \text{mm}$: WebCAE gives 0.25885, 0.26022 and $0.25997\ \text{mm}$. A parabola through these three nodes puts the maximum at $z = 123.5\ \text{mm}$ (analytical $122\ \text{mm}$, difference 1.2%) with the peak value $0.2602\ \text{mm}$ (analytical $0.2608\ \text{mm}$, -0.22%).

WHY The peak is flat, so the position is sensitive to the discretization: this is the reason for the fine axial mesh in the end zones and for 96 circumferential elements (see Common mistakes).

```
frame probe disp at=500,0,110 comp=ux id="ur_z110"
frame probe disp at=500,0,120 comp=ux id="ur_z120"
frame probe disp at=500,0,130 comp=ux id="ur_z130"
```



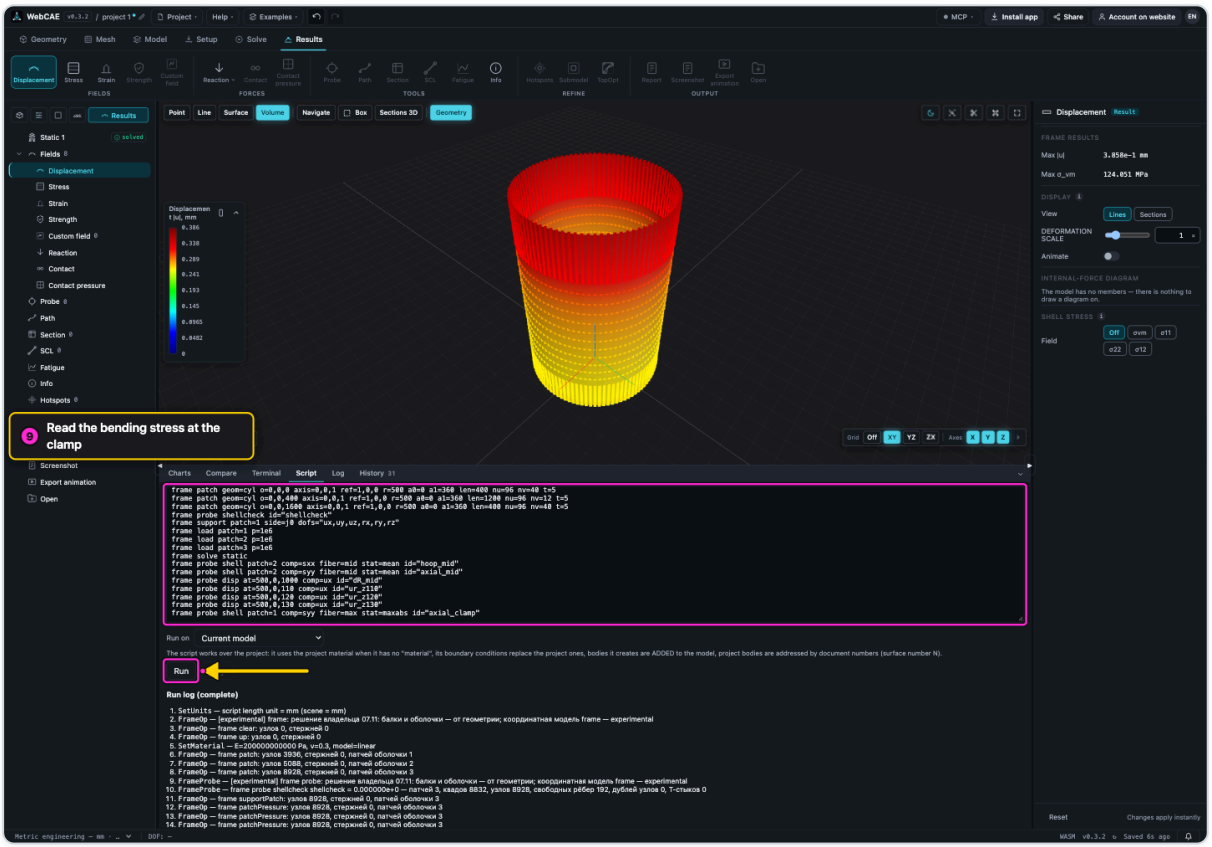
STEP 9

Read the bending stress at the clamp

The clamp moment is $M_0 = p/(2\beta^2) = 756.5\ \mathrm{N}\cdot\mathrm{mm/mm}$, which gives an outer-fiber axial stress of $6M_0/t^2 = 181.6\ \mathrm{MPa}$ exactly at the clamped ring, about 1.8 times the membrane stress, decaying as $e^{-\beta z}(\cos\beta z - \sin\beta z)$. The probe returns element-center values; the first element row of the first patch is centered at $z = 5\ \mathrm{mm}$, where the analytical stress is $181.6 \cdot 0.7594 = 137.9\ \mathrm{MPa}$. WebCAE reads $138.8\ \mathrm{MPa}$ (`fiber=max`, `stat=maxabs`, 0.68%).

WHY The discontinuity stress is a secondary bending stress that must be checked against fatigue and shakedown limits.

```
frame probe shell patch=1 comp=syy fiber=max stat=maxabs id="axial_clamp"
```



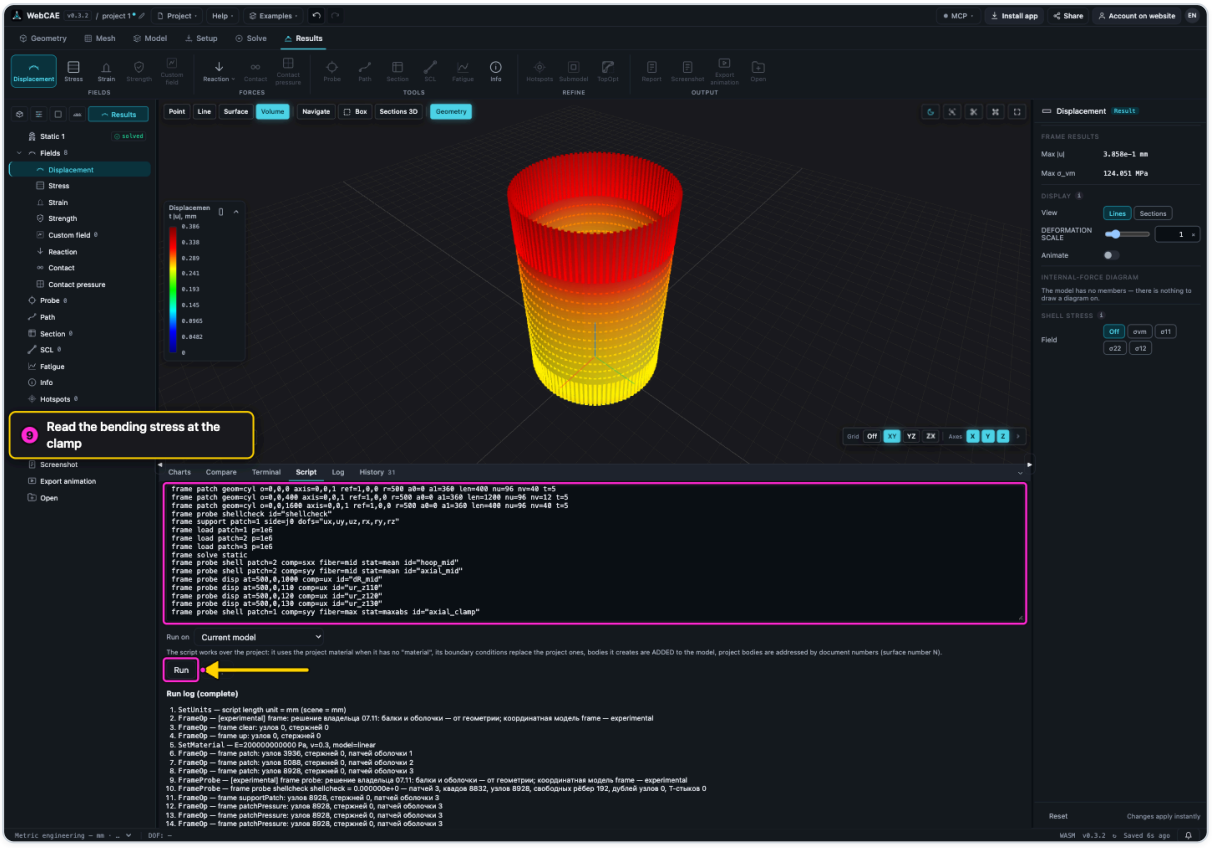
VERIFICATION BENCHMARK

Check against the formula

Quantity	Analytical Formula	WebCAE (desktop/native, MITC4)	Difference
Membrane hoop stress σ_θ (patch 2)	$\frac{pR}{t} = \frac{1.0\times500}{5} = 100.0\ \text{MPa}$	99.95 MPa	-0.05%
Radial expansion ΔR at $z = 1000\ \text{mm}$	$\frac{pR^2}{Et} = \frac{1.0\times500^2}{200\,000\times5} = 0.250\ \text{mm}$	0.2499 mm	-0.05%
Edge-effect decay length $\lambda = \pi/\beta$ (peak of u_r)	$\beta = \left[\frac{3(1-\nu^2)}{R^2t^2}\right]^{1/4} = 0.02571\ \text{mm}^{-1} \Rightarrow \lambda = 122\ \text{mm}$	123.5 mm (parabola through z = 110, 120, 130 mm)	+1.2%
Peak radial displacement near the clamp	$\Delta R\,(1 + e^{-\pi}) = 0.2608\ \text{mm}$	0.2602 mm	-0.22%
Axial bending stress at the first element row of the clamp (z = 5 mm)	$\frac{6M_0}{t^2}e^{-\beta z}(\cos\beta z - \sin\beta z) = 137.9\ \text{MPa}$	138.8 MPa	+0.68%

FORMULA

Laplace and Edge-Effect Theory. Far from the boundaries the thin cylinder gives $\sigma_\theta = \frac{pR}{t} = 100.0\ \text{MPa}$ and $\Delta R = \frac{pR^2}{Et} = 0.250\ \text{mm}$. At a clamped edge the radial restraint induces $M_0 = p/(2\beta^2) = 756.5\ \text{N}\cdot\text{mm/mm}$ and the radial displacement $u_r = \Delta R\,[1 - e^{-\beta z}(\cos\beta z + \sin\beta z)]$ with $\beta = 0.02571\ \text{mm}^{-1}$; the maximum of u_r is at $z = \pi/\beta = 122\ \text{mm}$.



TROUBLESHOOTING & NOTES

Common mistakes

- **Too few elements around the circumference:** with 32 circumferential elements instead of 96 the membrane stress and ΔR are already right (0.48%), but the peak of u_r moves to 136 mm (11.7% off) and does not improve with a finer axial mesh (138.6 mm with 5 mm axial elements). Refine around the circumference ($n_u = 96$ gives 123.5 mm).
- **Probing stress near the clamped end:** within the decay length ($z < 122$ mm, and in practice $z < 3\lambda$) the hoop stress is disturbed by the edge bending; read the membrane stress on the middle patch, far from both ends.
- **Assuming closed end caps:** without caps there is no axial pressure resultant $F_z = p\pi R^2$, so $\sigma_z = 0$ and $\Delta R = pR^2/(Et) = 0.250$ mm rather than 0.2125 mm ($\nu = 0.3$ closed-end value).
- **Forgetting the pressure on a patch:** every patch needs its own `frame load patch=… p=…`; a missed patch gives a vessel that is only partly loaded while the run still succeeds.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#25 — Stiffened Aerospace Panel (F25)** to reinforce a thin skin shell with eccentric beam stringers on shared nodes.

WebCAE 101

LESSON 25 · F25

Stiffened Aerospace Panel: Coupled Shell + Eccentric Beam Stringers

Goal Model an aircraft skin panel as a quad shell patch with three eccentric T-stringers on shared nodes, evaluate its response to uniform pressure and verify the deflection, the stringer stress and the mass against the orthotropic smeared-stiffener Navier solution.

Outcome A coupled shell-beam panel ($1000 \times 600 \times 4\ \mathrm{mm}$, $p = 0.005\ \mathrm{MPa}$) with a centre deflection of $1.737\ \mathrm{mm}$ (WebCAE desktop/native; analytical $1.78\ \mathrm{mm}$, difference 2.44%), versus $13.35\ \mathrm{mm}$ for the same panel without stringers (analytical $13.4\ \mathrm{mm}$, 0.38%), for $2.138\ \mathrm{kg}$ of stringers (analytical $2.1384\ \mathrm{kg}$) and a stringer web-edge stress of $44.1\ \mathrm{MPa}$ (analytical $45.0\ \mathrm{MPa}$, 1.96%).

9–10 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

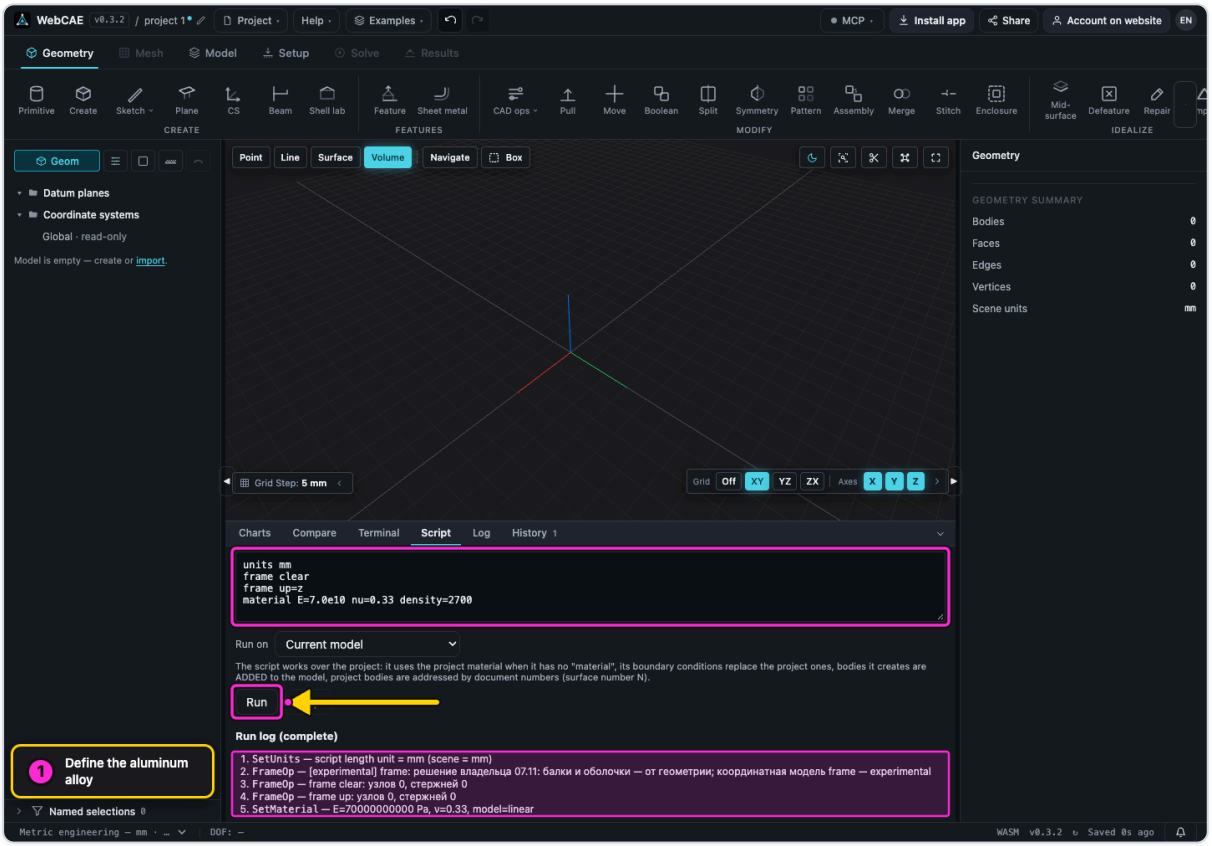
Run: 2026-11-04-32ffaed2 / F25

STEP 1 Define the aluminum alloy

Open the bottom dock, tab **Script**, set **Run on** to **Current model**, and paste the code blocks in order (the **Frame** tile opens the beam-and-shell workspace). Aluminum alloy 2024-T3: $E = 70\,000\ \mathrm{MPa} = 7.0 \times 10^{10}\ \mathrm{Pa}$ (the DSL takes Pascals), $\nu = 0.33$, $\rho = 2700\ \mathrm{kg/m^3}$.

WHY Aluminum gives high specific stiffness; stringers prevent a thin sheet from deflecting several thicknesses under a small load.

```
units mm
frame clear
frame up=z
material E=7.0e10 nu=0.33 density=2700
```



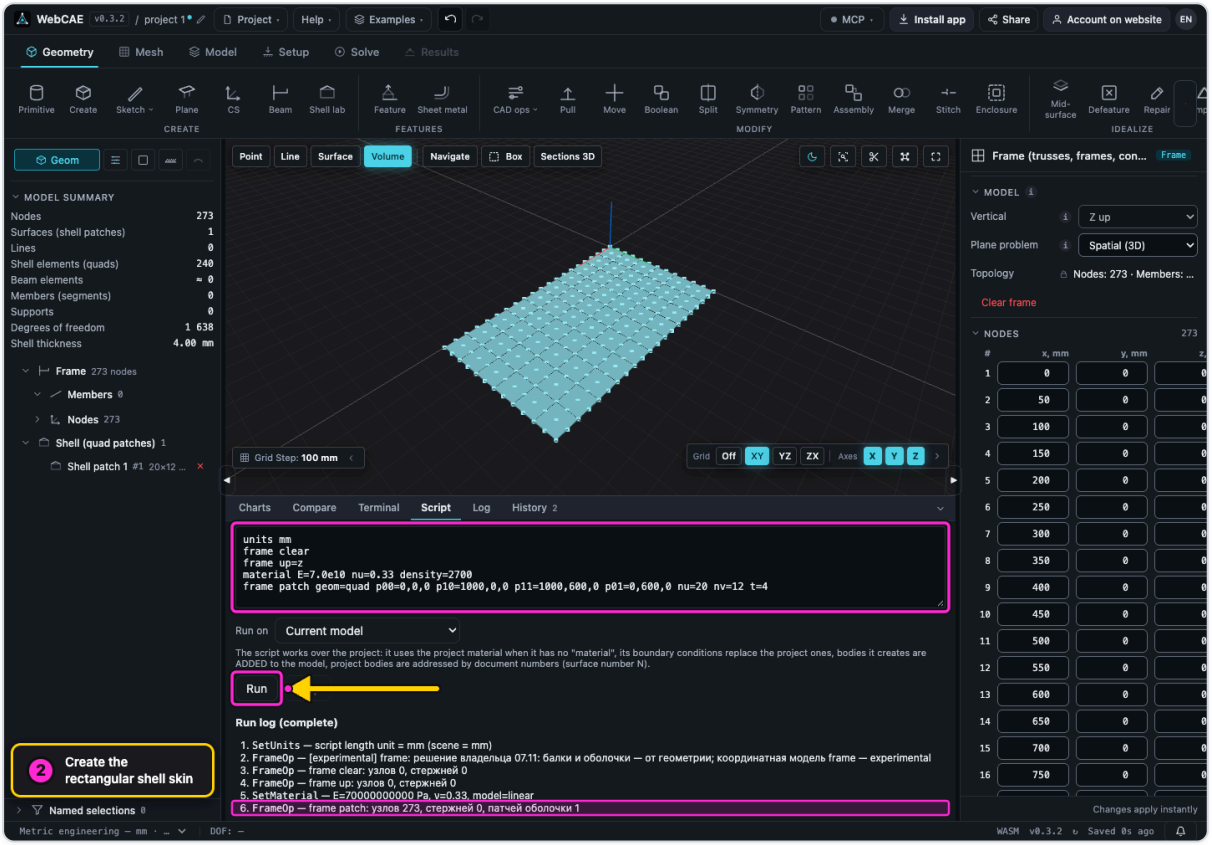
STEP 2

Create the rectangular shell skin

A planar quad shell patch spanning $1000\ \mathrm{mm}$ along X and $600\ \mathrm{mm}$ along Y, divided into $n_u = 20$ by $n_v = 12$ elements, with sheet thickness $t = 4\ \mathrm{mm}$ (corners `p00`, `p10`, `p11`, `p01` in order around the rectangle).

> **WHY** $n_v = 12$ puts grid rows at $y = 150, 300, 450\ \mathrm{mm}$ ($j = 3, 6, 9$): exact shared-node lines for three longitudinal stringers.

```
frame patch geom=quad p00=0,0,0 p10=1000,0,0 p11=1000,600,0 p01=0,600,0 nu=20 nv=12 t=4
```



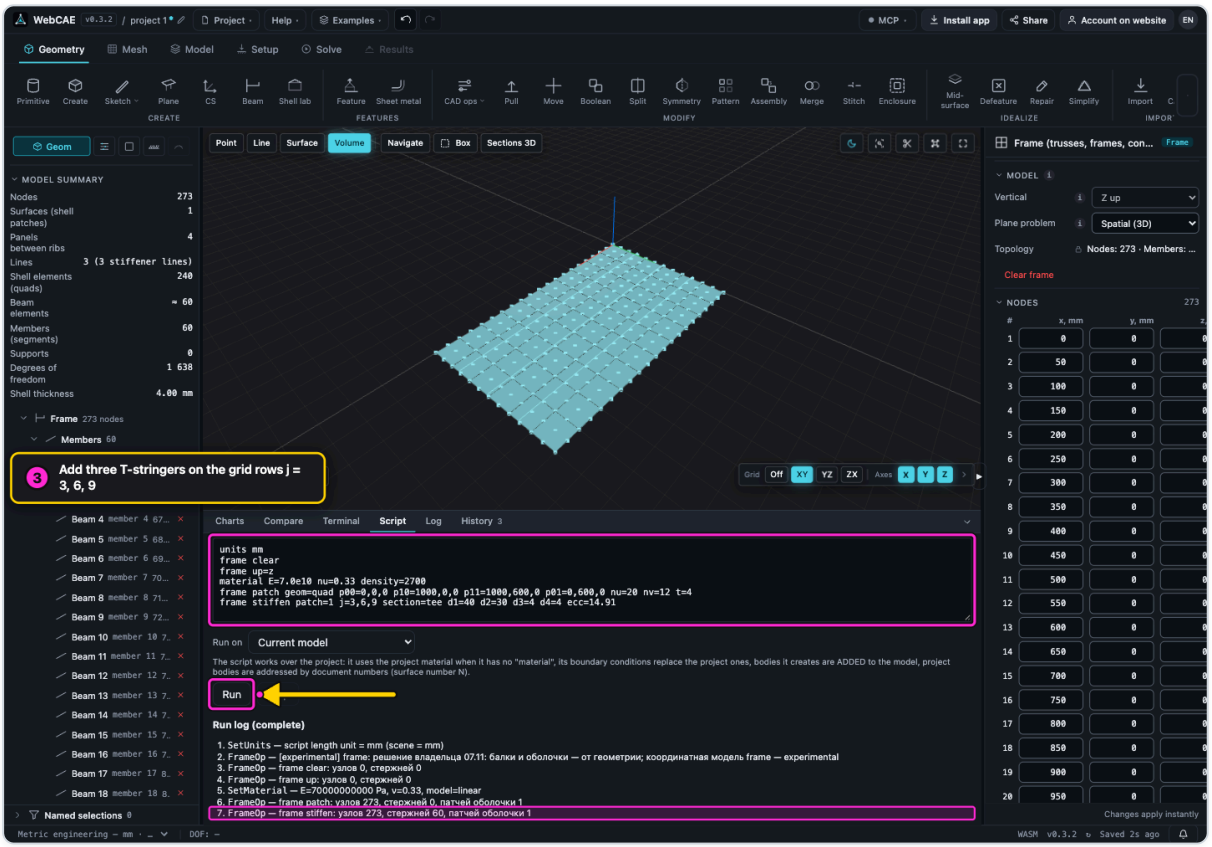
STEP 3

Add three T-stringers on the grid rows j = 3, 6, 9

`frame stiffen` puts beam members along the grid rows on the shared nodes of the shell. The T profile is $d_1 = H = 40\ \text{mm}$, $d_2 = B = 30\ \text{mm}$, $d_3 = d_4 = 4\ \text{mm}$ (area $A = 264\ \text{mm}^2$). Its centroid is $z_c = \frac{120 \cdot 2 + 144 \cdot 22}{264} = 12.91\ \text{mm}$ from the flange contact face, so the centroid sits $e = t/2 + z_c = 14.91\ \text{mm}$ from the skin mid-surface; this offset is passed explicitly as `ecc=14.91` (the `flush=` argument only accepts `outer` or `inner`). The sign of `ecc` chooses the side of the skin the stringers are on: with $+14.91$ and the pressure acting along the shell normal the stringers are in tension, with -14.91 they are in compression.

WHY The offset places the beam axis where the real section centroid is, so skin and stringer bend together without any multi-point constraint.

```
frame stiffen patch=1 j=3,6,9 section=tee d1=40 d2=30 d3=4 d4=4 ecc=14.91
```



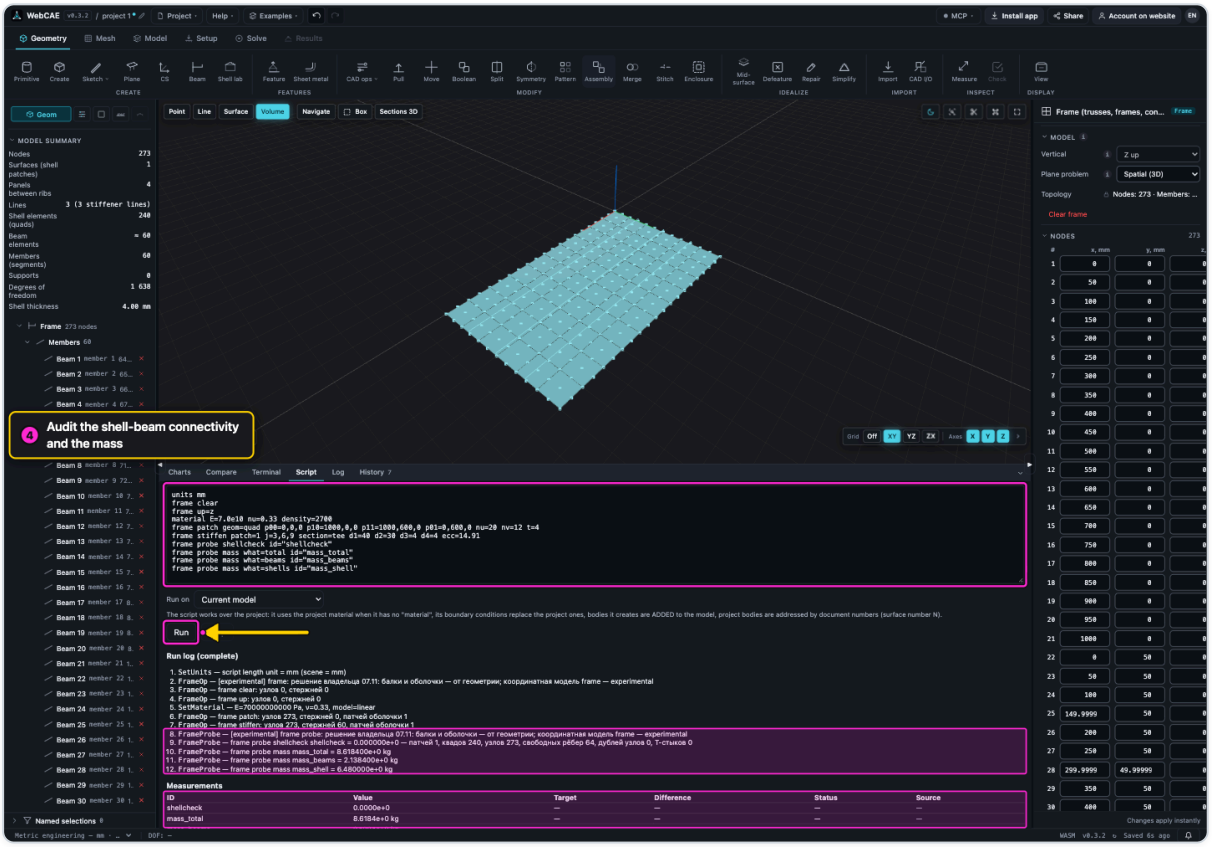
STEP 4

Audit the shell-beam connectivity and the mass

`frame probe shellcheck` must return 0 (no unconnected patch interfaces). `frame probe mass` returns the skin mass $6.48\ \mathrm{kg}$ ($1000 \cdot 600 \cdot 4 \cdot 2.7 \times 10^{-6}$), the stringer mass $2.1384\ \mathrm{kg}$ ($3 \cdot 264 \cdot 1000 \cdot 2.7 \times 10^{-6}$) and the total $8.6184\ \mathrm{kg}$; all three agree with the hand values to the printed digits.

WHY A mass audit catches duplicated flange volume or a missing stringer immediately.

```
frame probe shellcheck id="shellcheck"
frame probe mass what=total id="mass_total"
frame probe mass what=beams id="mass_beams"
frame probe mass what=shells id="mass_shell"
```



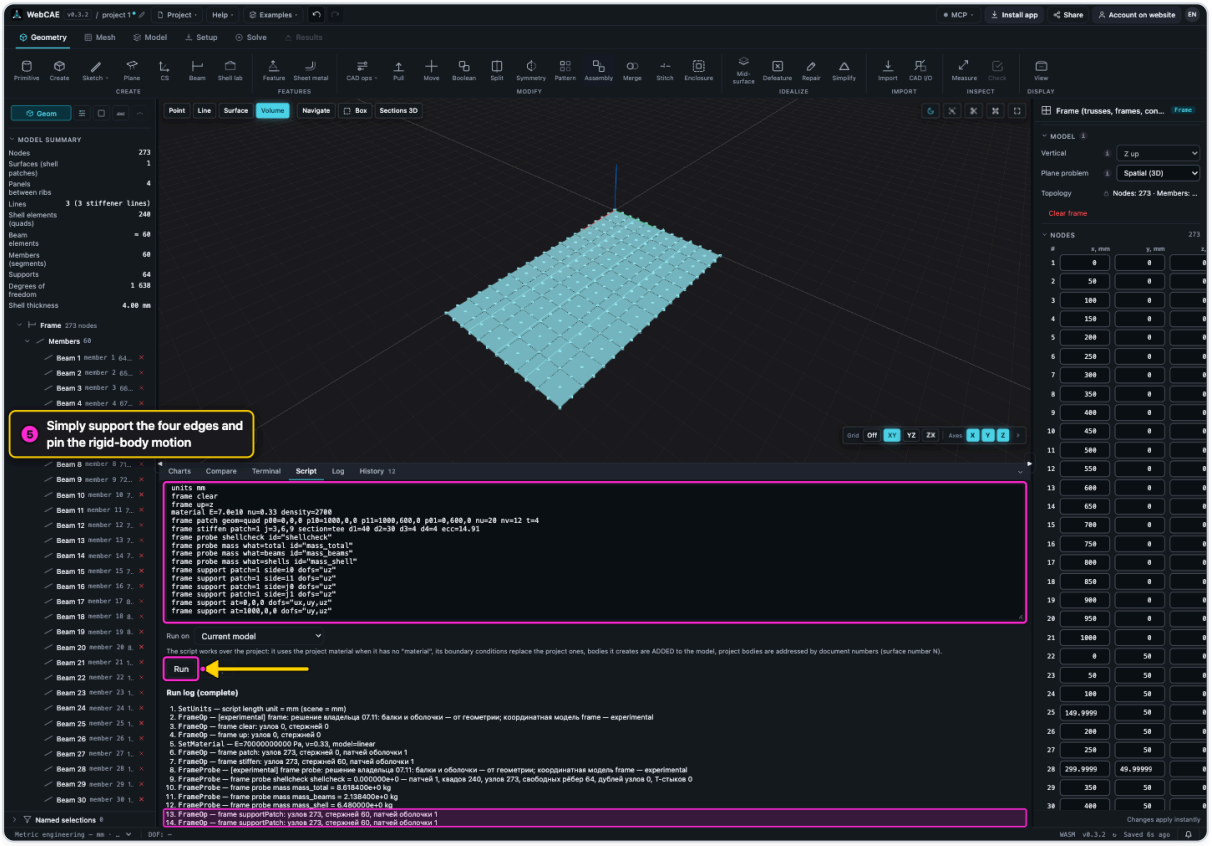
STEP 5

Simply support the four edges and pin the rigid-body motion

Constrain u_z along all four edges of the patch (`side=i0` , `i1` , `j0` , `j1`), then remove the in-plane rigid-body motion with two corner pins: corner $(0, 0, 0)$ fixed in u_x, u_y, u_z and corner $(1000, 0, 0)$ fixed in u_y, u_z. Do not use `side=all` : it fixes every node of the patch listed in `dofs` , not just the contour.

> **WHY** Simply supported edges represent a skin riveted to frames and stringer rails, and the two corner pins leave the panel free to expand in its plane.

```
frame support patch=1 side=i0 dofs="uz"
frame support patch=1 side=i1 dofs="uz"
frame support patch=1 side=j0 dofs="uz"
frame support patch=1 side=j1 dofs="uz"
frame support at=0,0,0 dofs="ux,uy,uz"
frame support at=1000,0,0 dofs="uy,uz"
```



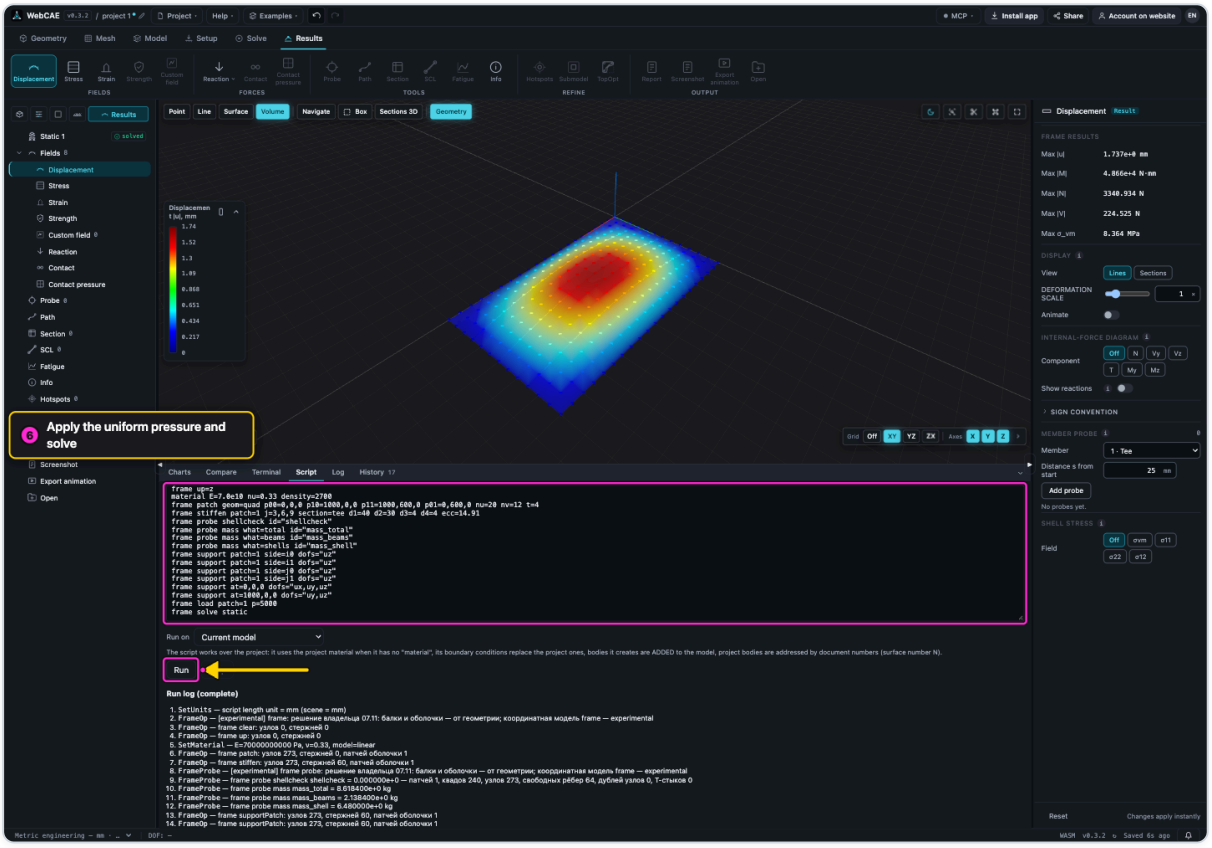
STEP 6

Apply the uniform pressure and solve

Apply $p = 0.005\ \mathrm{MPa} = 5000\ \mathrm{Pa}$ (a light aerodynamic or test load, `p=` is in Pascals) on the patch and solve. The small load keeps the stiffened panel in the linear regime ($\delta < t/2 = 2\ \mathrm{mm}$).

> **WHY** Uniform pressure loads the skin in biaxial bending and the stringers as beams.

```
frame load patch=1 p=5000
frame solve static
```



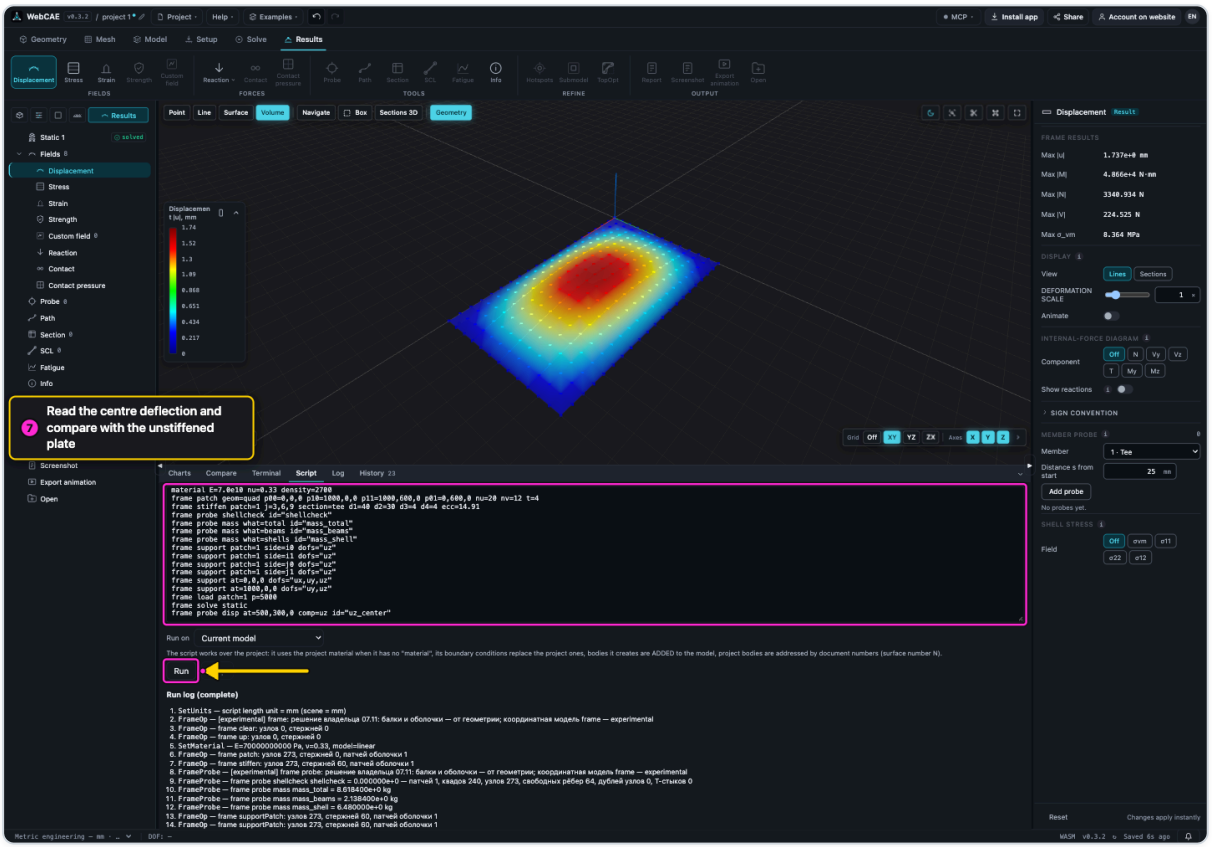
STEP 7 Read the centre deflection and compare with the unstiffened plate

The deflection at the panel centre $(500, 300, 0)$ is $1.737\ \text{mm}$ in WebCAE against $1.78\ \text{mm}$ from the orthotropic Navier solution (difference 2.44%).

Control: repeat the run without the `frame stiffen` line. The plain 4 mm sheet deflects $13.35\ \text{mm}$ (small-deflection Navier estimate $13.4\ \text{mm}$, difference 0.38%), which is $3.3\,t$ and would need geometric nonlinearity in practice; three lightweight stringers cut the deflection by a factor of 7.7 and keep the panel inside the linear range.

> **WHY** The stiffened centre deflection of $1.737\ \text{mm}$ is below $t/2 = 2\ \text{mm}$, so the linear result is trustworthy; the unstiffened value is only a small-deflection estimate.

```
frame probe disp at=500,300,0 comp=uz id="uz_center"
```



STEP 8 Read the stringer forces and the web-edge stress

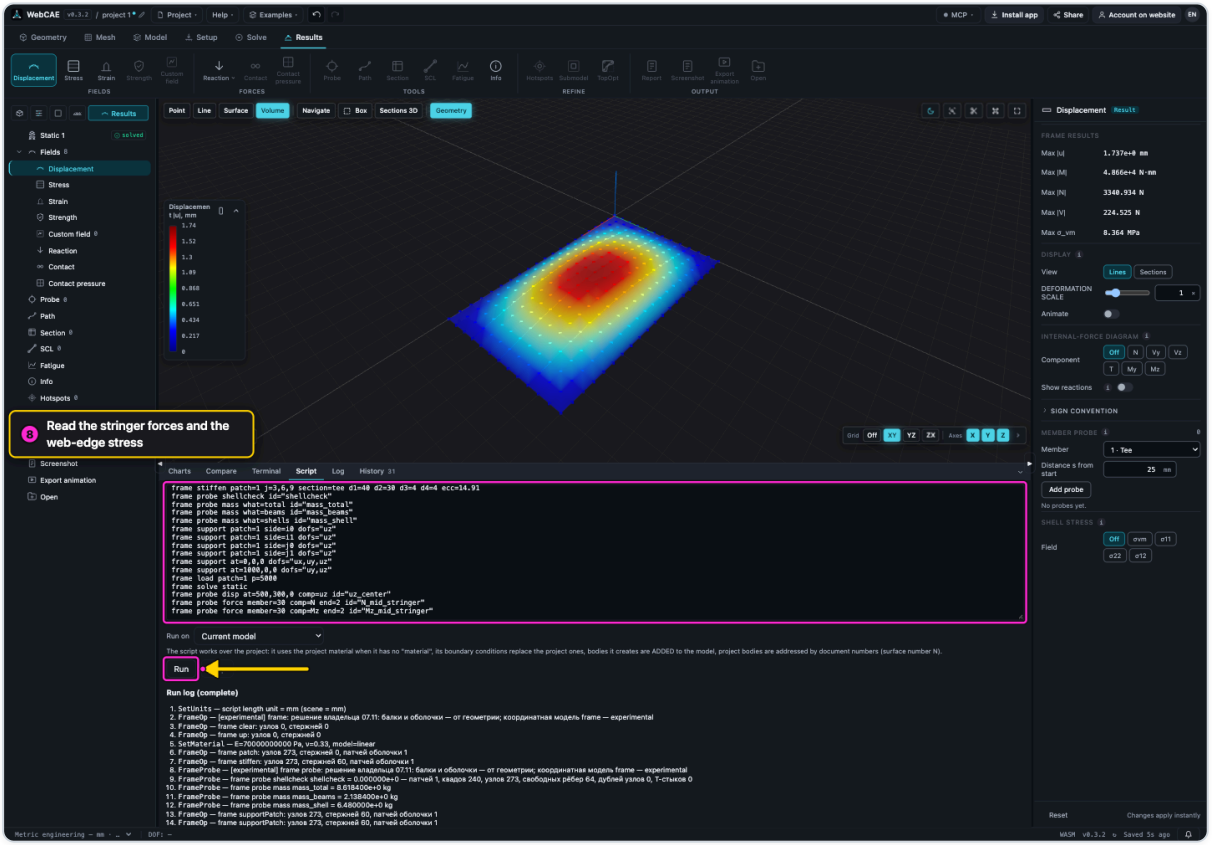
The beam solver gives forces, not stresses, so the stress is a hand post-processing of the section forces of the middle stringer at mid-span. Member 30 (the 10th of the 20 members of the middle stringer) ends at $x = 500\ \text{mm}$; WebCAE gives $N = 3341\ \text{N}$ (tension) and $M_z = 48.66\ \text{N}\cdot\text{m}$ at its end 2. The stress at the free edge of the web is

$$\sigma_x = \frac{N}{A} + \frac{Mc}{I_0} = \frac{3341}{264} + \frac{48\,660 \cdot 27.09}{41\,894} = 12.65 + 31.47 = 44.1\ \text{MPa}$$

with $c = H - z_c = 40 - 12.91 = 27.09\ \text{mm}$ and the own inertia $I_0 = 41\,894\ \text{mm}^4$ of the T section. The analytical value is $45.0\ \text{MPa}$ (difference 1.96%), far below the $325\ \text{MPa}$ yield strength. Run `frame info` to list the members and check the numbering.

> **WHY** Eccentric stringers pull the neutral axis toward the stringers, so the stringer web edge carries the highest tensile stress of the section.

```
frame probe force member=30 comp=N end=2 id="N_mid_stringer"
frame probe force member=30 comp=Mz end=2 id="Mz_mid_stringer"
```



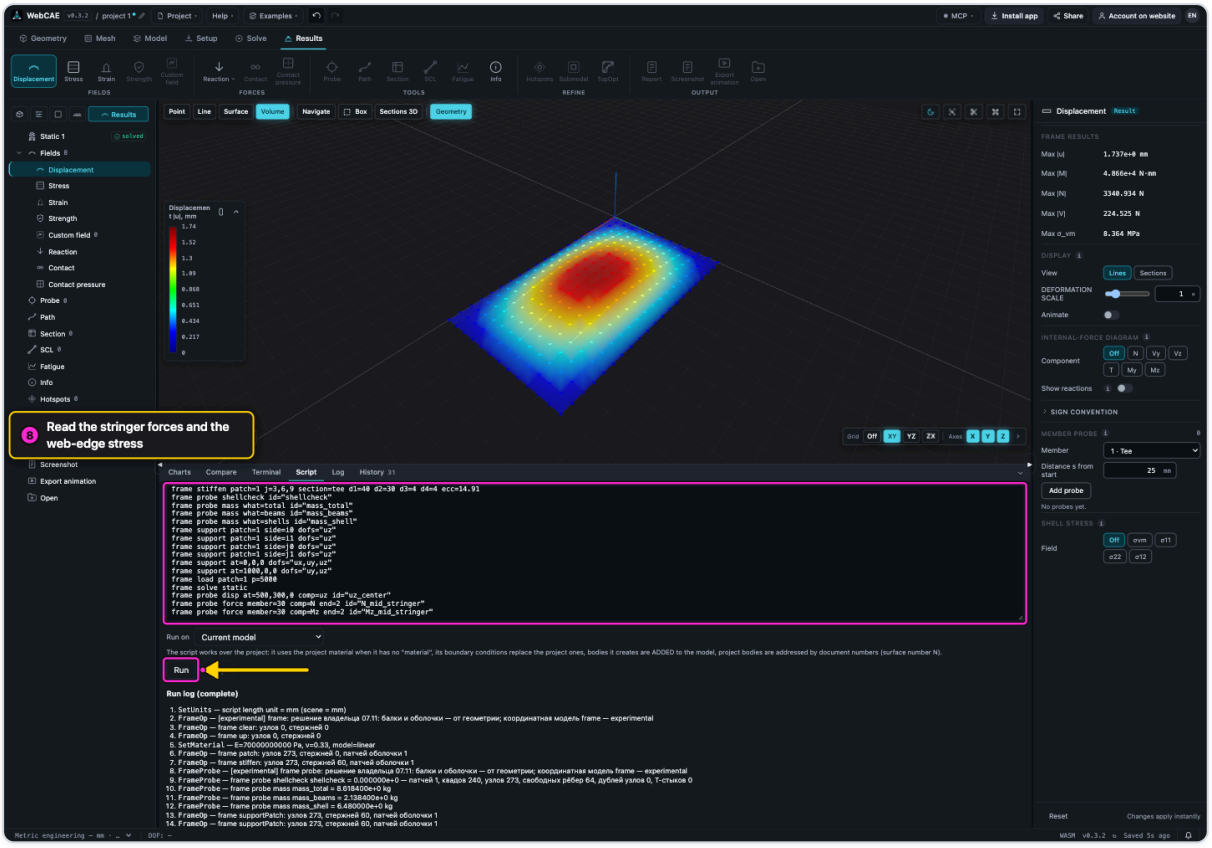
VERIFICATION BENCHMARK

Check against the formula

Configuration / Quantity	Analytical / Smeared Theory	WebCAE (desktop/native)	Difference
Stiffened panel centre deflection	Orthotropic smeared plate (Navier), composite neutral axis: $1.78\ \text{mm}$ at $p = 0.005\ \text{MPa}$	1.737 mm	-2.44%
Unstiffened plate small-deflection deflection	Navier series, $D_x = D_y = D = 418\,958\ \text{N}\cdot\text{mm}$: $13.4\ \text{mm}$	13.35 mm	-0.38%
Mass of the 3 stringers	$3 \times 264\ \text{mm}^2 \times 1000\ \text{mm} \times 2.7 \times 10^{-6}\ \text{kg/mm}^3 = 2.1384\ \text{kg}$	2.1384 kg	0.00%
Total panel mass	$6.48 + 2.1384 = 8.6184\ \text{kg}$	8.6184 kg	0.00%
Stringer web-edge axial stress σ_x at mid-span	$E\kappa_x(z_{\text{tip}} - \bar{y}) = 70\,000 \cdot 1.716 \times 10^{-5} \cdot 37.44 = 45.0\ \text{MPa}$	44.1 MPa (from $N = 3341\ \text{N}$, $M_z = 48.66\ \text{N}\cdot\text{m}$)	-1.96%

FORMULA

Orthotropic Smeared Stiffener Theory. Plate rigidity $D = \frac{Et^3}{12(1-\nu^2)} = \frac{70\,000 \cdot 64}{12(1-0.33^2)} = 418\,958\ \text{N}\cdot\text{mm}$; stringer pitch $s = 150\ \text{mm}$; the T section ($A = 264\ \text{mm}^2$, own inertia $I_0 = 41\,894\ \text{mm}^4$, $e = 14.91\ \text{mm}$ from the skin mid-surface) has the inertia about the skin mid-surface $I_{st} = I_0 + Ae^2 = 100\,576\ \text{mm}^4$. Because the panel edges are free in-plane (only corner pins), skin and stringer bend about their common neutral axis: with the skin strip $s \times t = 600\ \text{mm}^2$, $\bar{y} = \frac{264 \cdot 14.91}{864} = 4.56\ \text{mm}$ and $I_{\text{comp}} = (800 + 100\,576) - 864 \cdot 4.56^2 = 83\,445\ \text{mm}^4$. The orthotropic rigidities are $D_x = EI_{\text{comp}}/s = 3.894 \times 10^7\ \text{N}\cdot\text{mm}$, $D_y = D$ and $2H = 2D$ (stringer torsion would lower δ by only 1.5% and is neglected). Navier's double series $w(a/2, b/2) = \frac{16p}{\pi^6} \sum_{m,n\ \text{odd}} \frac{(-1)^{(m+n)/2-1}}{mn\,[D_x(m/a)^4 + 2H(m/a)^2(n/b)^2 + D_y(n/b)^4]}$ with $a = 1000\ \text{mm}$, $b = 600\ \text{mm}$ gives $1.78\ \text{mm}$ at $p = 0.005\ \text{MPa}$ (the unstiffened plate, $D_x = D_y = D$, gives $13.4\ \text{mm}$). Assuming the stringers bend about the skin mid-surface (no neutral-axis shift) would give $D_x = 4.74 \times 10^7$ and $1.49\ \text{mm}$, 18% lower: the shift matters. The web-edge stress is $E\kappa_x(z_{\text{tip}} - \bar{y})$ with the centre curvature $\kappa_x = 1.716 \times 10^{-5}\ \text{mm}^{-1}$, $z_{\text{tip}} = 2 + 40 = 42\ \text{mm}$ and $\bar{y} = 4.56\ \text{mm}$.



TROUBLESHOOTING & NOTES

Common mistakes

- **Forgetting the stringer eccentricity:** concentric stringers ($e = 0$) lose the Ae^2 parallel-axis term and I_{comp} drops from $83\,445$ to $42\,693\ \text{mm}^4$, roughly halving the stringer contribution to the stiffness. Always set `ecc`.
- **Using `flush=ecc`:** the `flush=` argument accepts only `outer` or `inner`; give the offset explicitly as `ecc=14.91`, whose sign also decides whether the stringers are in tension (+) or compression (-) under the pressure.
- **Using `side=all` for the edge supports:** it fixes all listed degrees of freedom on every node of the patch (the first version of this model had zero displacement); use `side=edges` (the whole rim at once) or the four sides `i0`, `i1`, `j0`, `j1`, plus two corner pins.
- **Non-conforming mesh divisions:** if n_v is not a multiple of the stringer rows, the stringers miss shell nodes and cannot transfer shear flow.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#26 — Support Reactions: A Plate-and-Lever Shell Model and a Solid Bracket with a Remote Load (F26)** to verify force and moment equilibrium across shell and solid models.

WebCAE 101

LESSON 26 · F26

Support Reactions: A Plate-and-Lever Shell Model and a Solid Bracket with a Remote Load

Goal Read support reactions in two models. First a clamped shell plate carrying a pipe lever (frame model), solved with `frame solve static nlgeom=on`, which returns reactions under shells. Then a solid bracket with a bore, loaded by a lever force through a remote load (a rigid link to the bore wall), checking the force balance.

Outcome Two reaction checks on the native solver: the shell-plate model gives $\sum R_z = 500.000\ \mathrm{N}$ for a $500\ \mathrm{N}$ load and a force and moment balance residual of 0.26%, and the solid bracket gives $\sum R_z = 500.0001\ \mathrm{N}$ with $\sum R_y = -5 \cdot 10^{-5}\ \mathrm{N}$. Frame and solid models cannot share one document, and the product has no RBE3 element: the rigid link of the remote load is of the RBE2 kind.

9–11 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F26

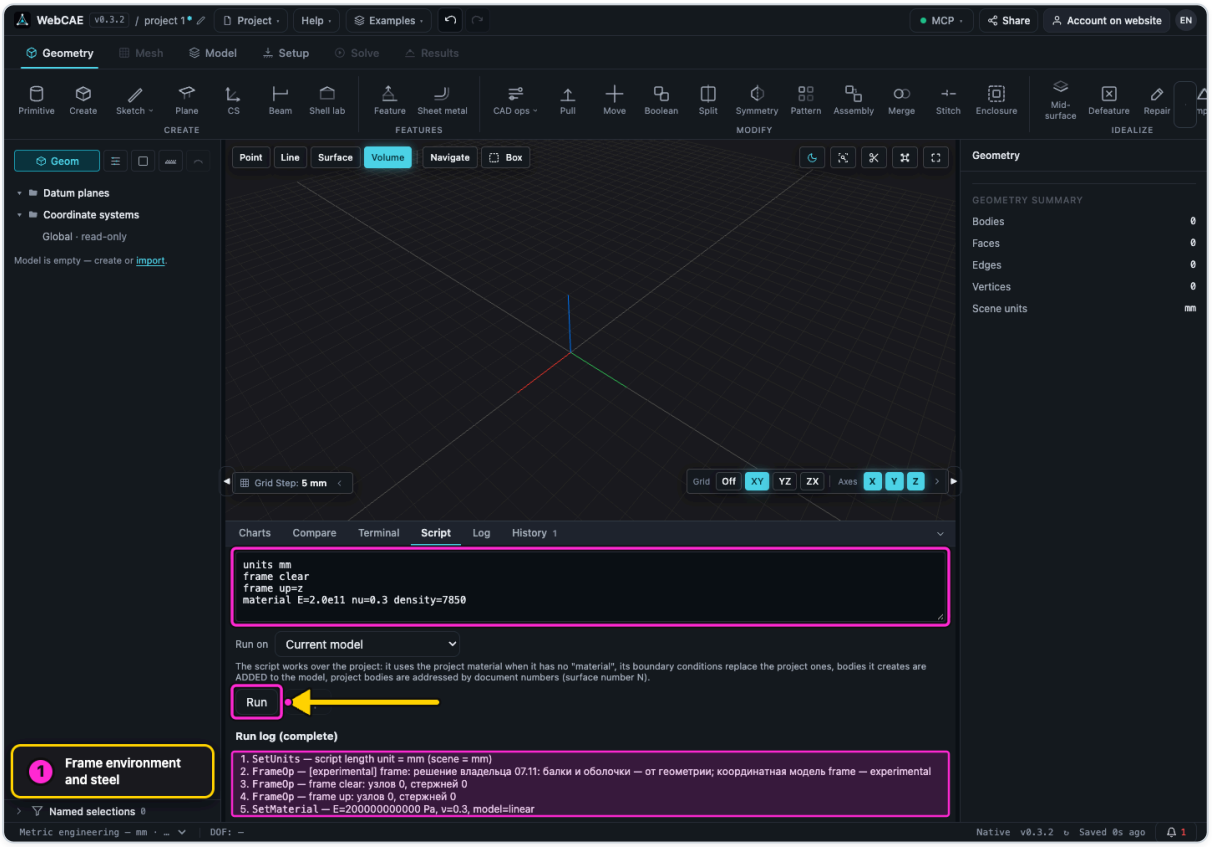
STEP 1

Frame environment and steel

Open the **Frame** workspace (`frame clear`, Z up) and assign structural steel: $E = 2.0 \times 10^{11}$ Pa, $\nu = 0.3$, $\rho = 7850$ kg/m^3.

WHY The frame workspace holds beams and shells in one document; solids belong to the other mode.

```
units mm
frame clear
frame up=z
material E=2.0e11 nu=0.3 density=7850
```



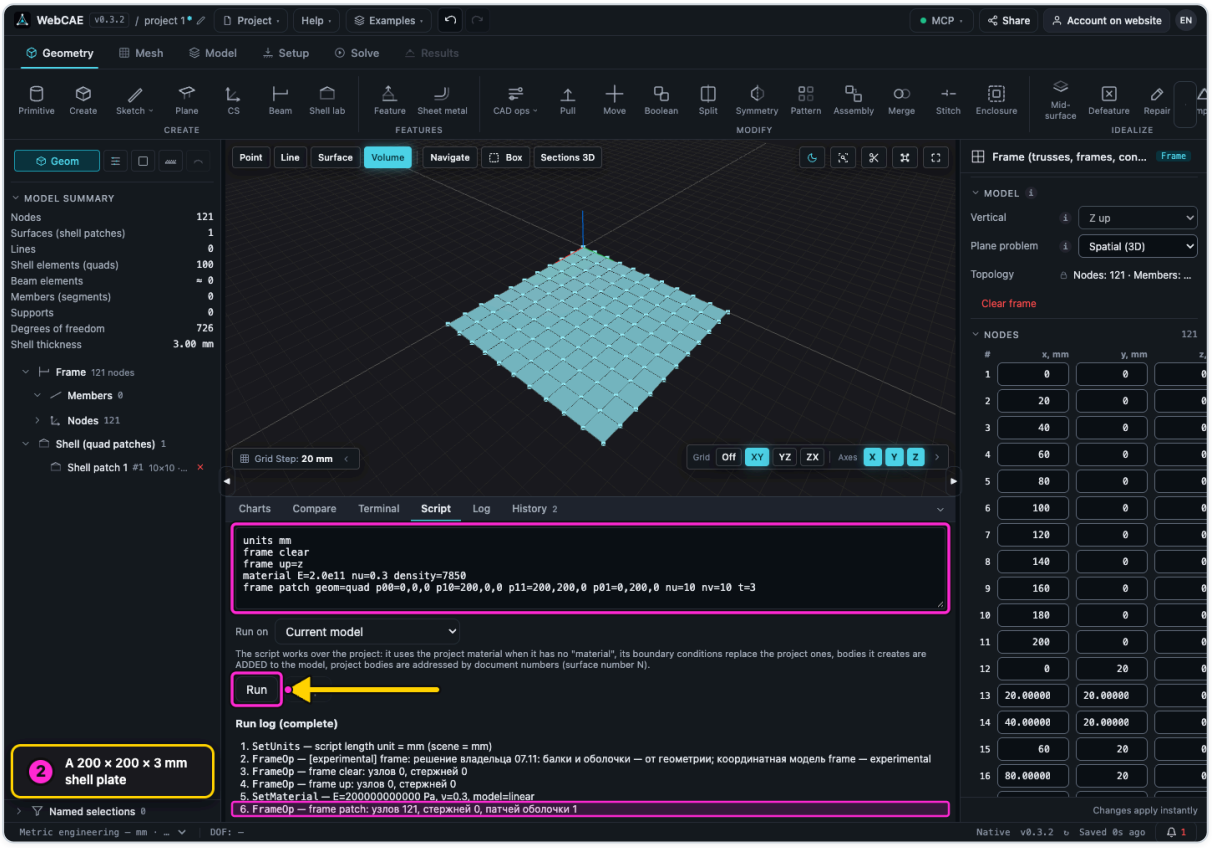
STEP 2

A 200 × 200 × 3 mm shell plate

`frame patch geom=quad` makes a MITC4 shell patch 200×200 mm, $t = 3$ mm, with a 10×10 mesh (121 nodes; node 61 is the centre).

WHY MITC4 shell elements carry bending and membrane action with 6 degrees of freedom per node.

```
frame patch geom=quad p00=0,0,0 p10=200,0,0 p11=200,200,0 p01=0,200,0 nu=10 nv=10 t=3
```

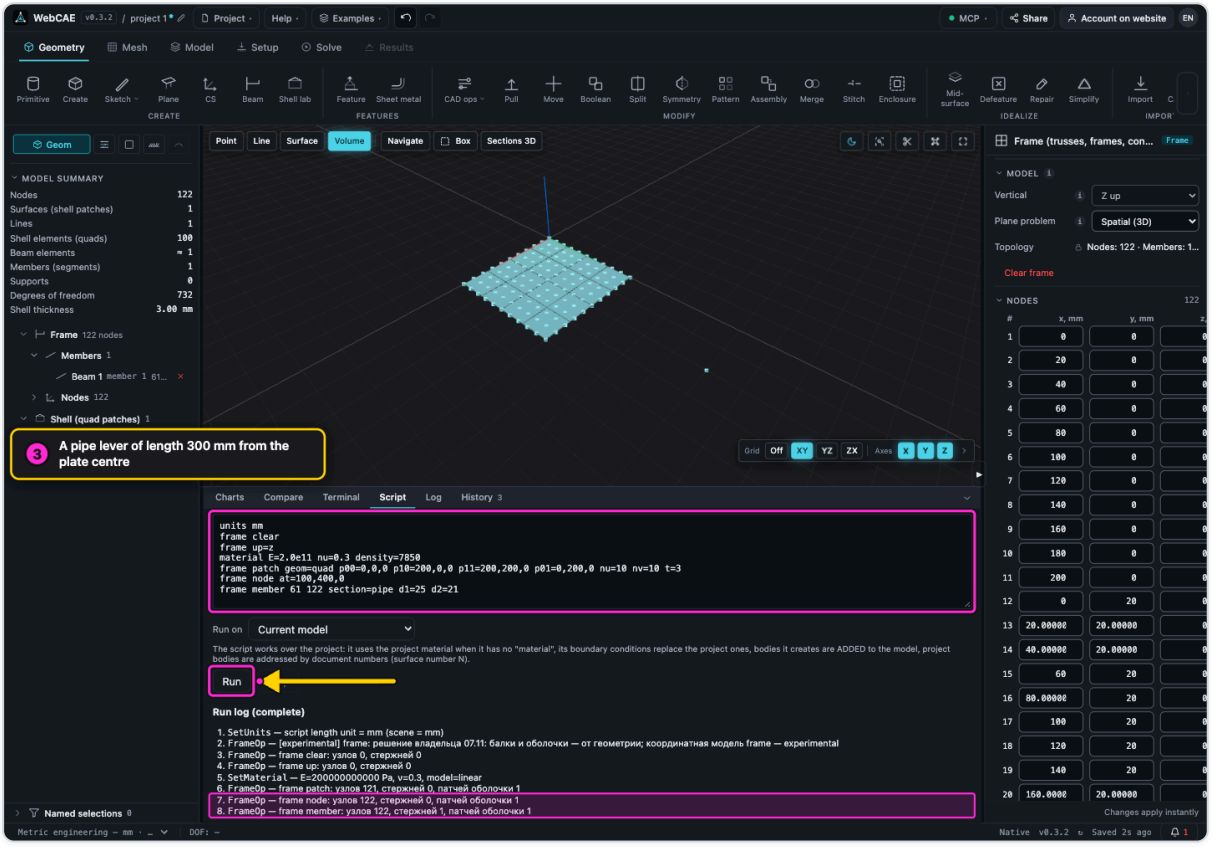


STEP 3 A pipe lever of length 300 mm from the plate centre

`frame node at=100,400,0` adds node 122 (it follows the 121 plate nodes); `frame member 61 122 section=pipe d1=25 d2=21` joins the plate centre to it: a pipe ⌀ 25 × ⌀ 21, lever arm $L = 300\ \mathrm{mm}$. A force at the lever end loads the plate with the force and with the moment $M = 500 \cdot 300 = 150\,000\ \mathrm{N\,mm} = 150\ \mathrm{N\,m}$.

WHY The lever turns a single tip force into force plus moment on the plate, which the supports must resist.

```
frame node at=100,400,0
frame member 61 122 section=pipe d1=25 d2=21
```

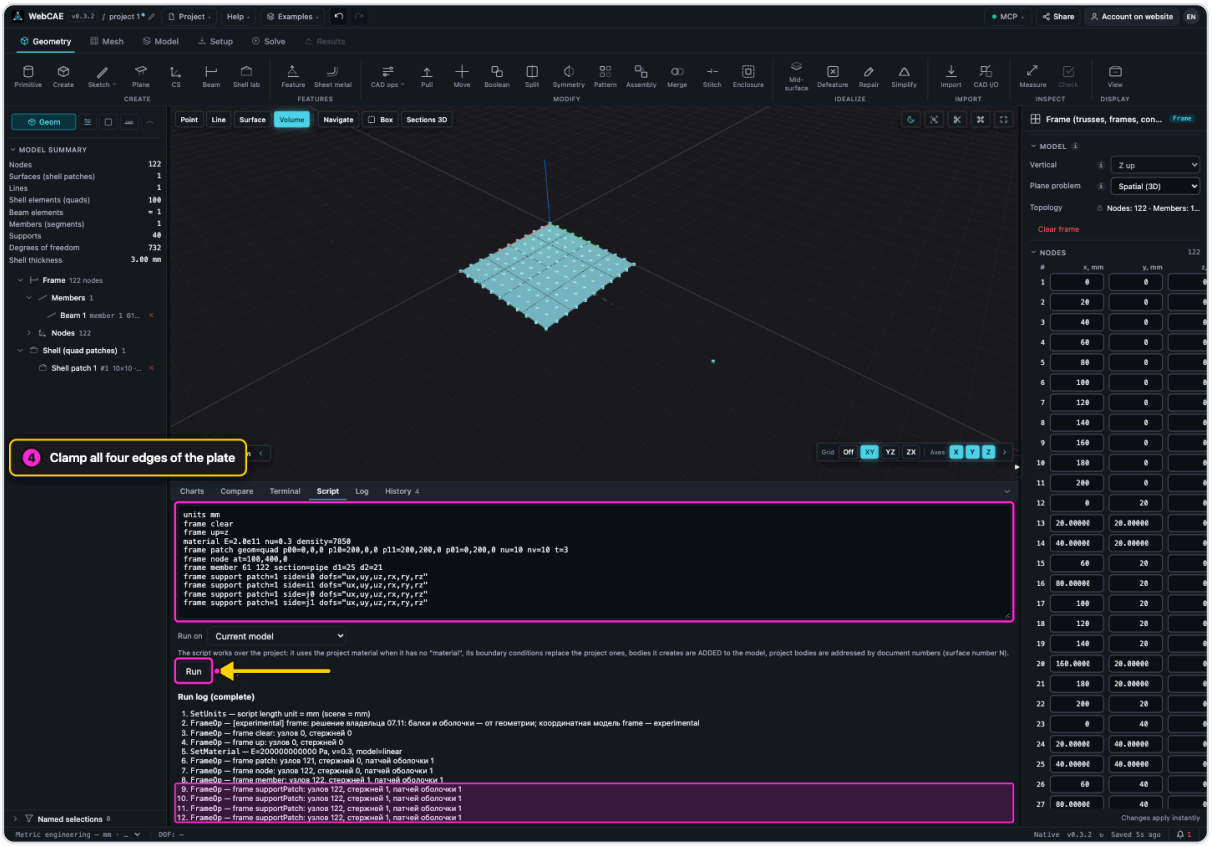


STEP 4 Clamp all four edges of the plate

`frame support patch=1 side=i0|i1|j0|j1` fixes the nodes of each edge of the patch with all six degrees of freedom. **Do not use `side=all` for a rim support** (use `side=edges` for the whole rim): `all` fixes every node of the patch, so the plate becomes rigid and the model checks nothing but statics of a rigid base (the tip deflection then drops to 2.35 mm, the pure beam value, instead of 25.8 mm).

WHY A clamped rim keeps the plate flexible, so the reactions are shared by the four edges.

```
frame support patch=1 side=i0 dofs="ux,uy,uz,rx,ry,rz"
frame support patch=1 side=i1 dofs="ux,uy,uz,rx,ry,rz"
frame support patch=1 side=j0 dofs="ux,uy,uz,rx,ry,rz"
frame support patch=1 side=j1 dofs="ux,uy,uz,rx,ry,rz"
```

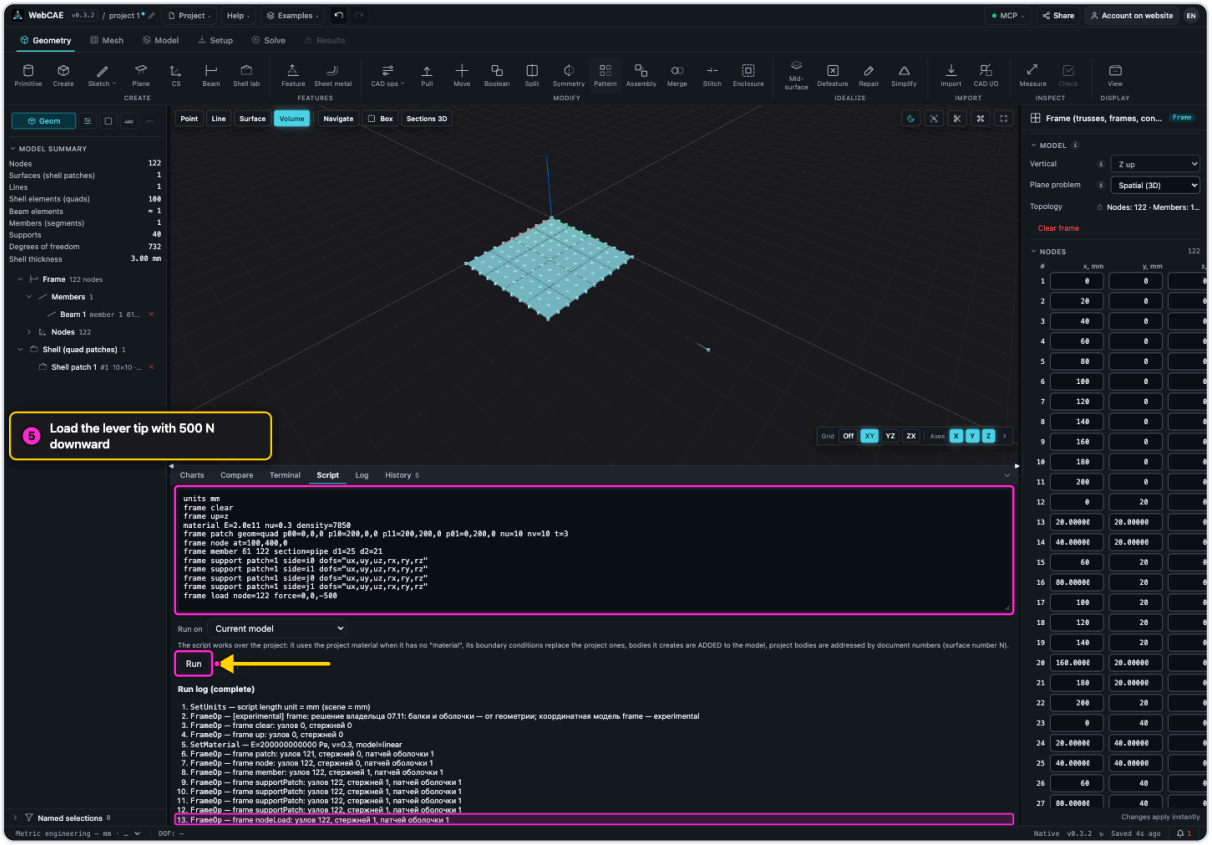


STEP 5 Load the lever tip with 500 N downward

`frame load node=122 force=0,0,−500` applies $P = 500\ \mathrm{N}$ in $-z$ at the lever end.

WHY The expected total reaction is the load itself: $\sum R_z = 500\ \mathrm{N}$.

```
frame load node=122 force=0,0,−500
```

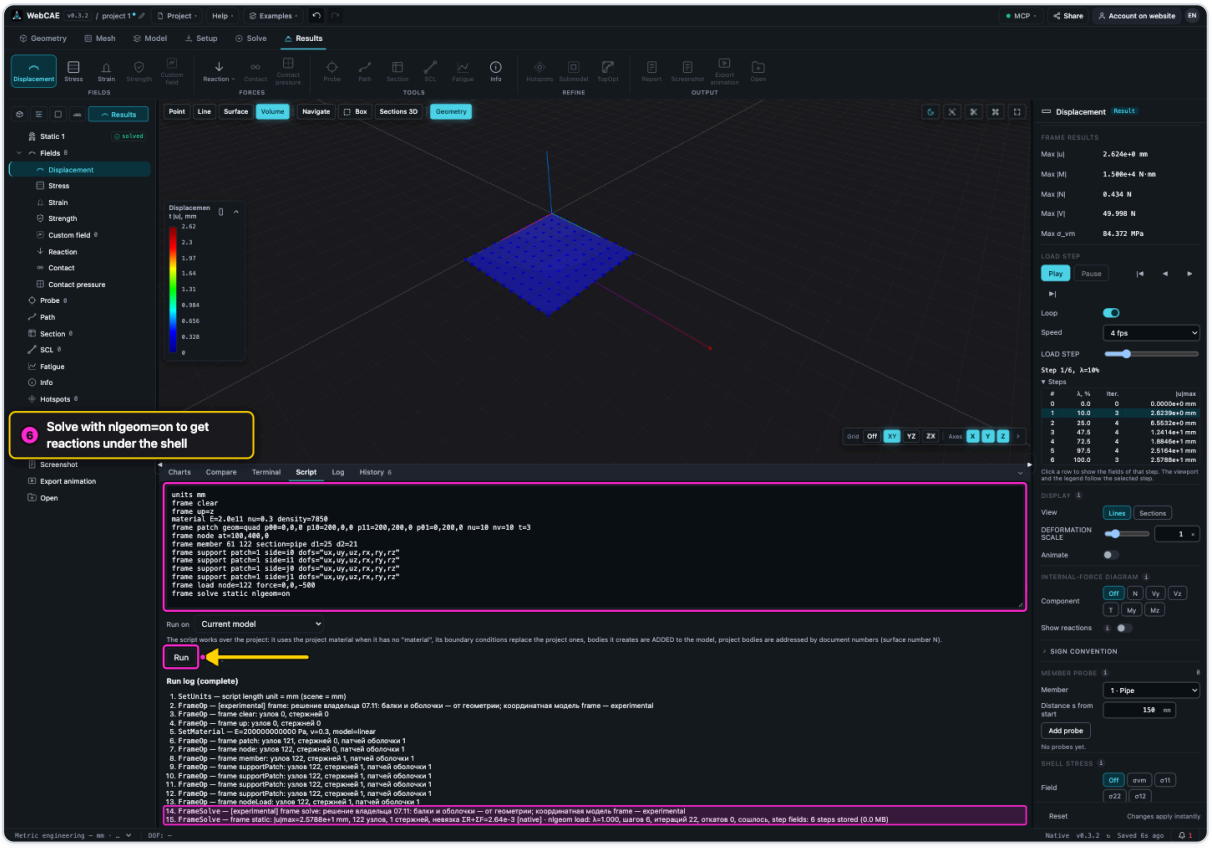


STEP 6 Solve with nlgeom=on to get reactions under the shell

A linear `frame solve static` refuses reaction probes at supports that hold a shell: the linear core does not return nodal forces of quads, and the product says so and names the way out. `frame solve static nlgeom=on` runs the native nonlinear core, which returns the reactions of every support node, including those under the shell; for small displacements it agrees with the linear solution. Here the lever tip moves by $25.8\ \mathrm{mm}$ on a $300\ \mathrm{mm}$ lever, so the solution is mildly geometrically nonlinear.

> **WHY** Reactions under shells need the native nonlinear core; the refusal in the linear path is deliberate, not a missing number.

```
frame solve static nlgeom=on
```

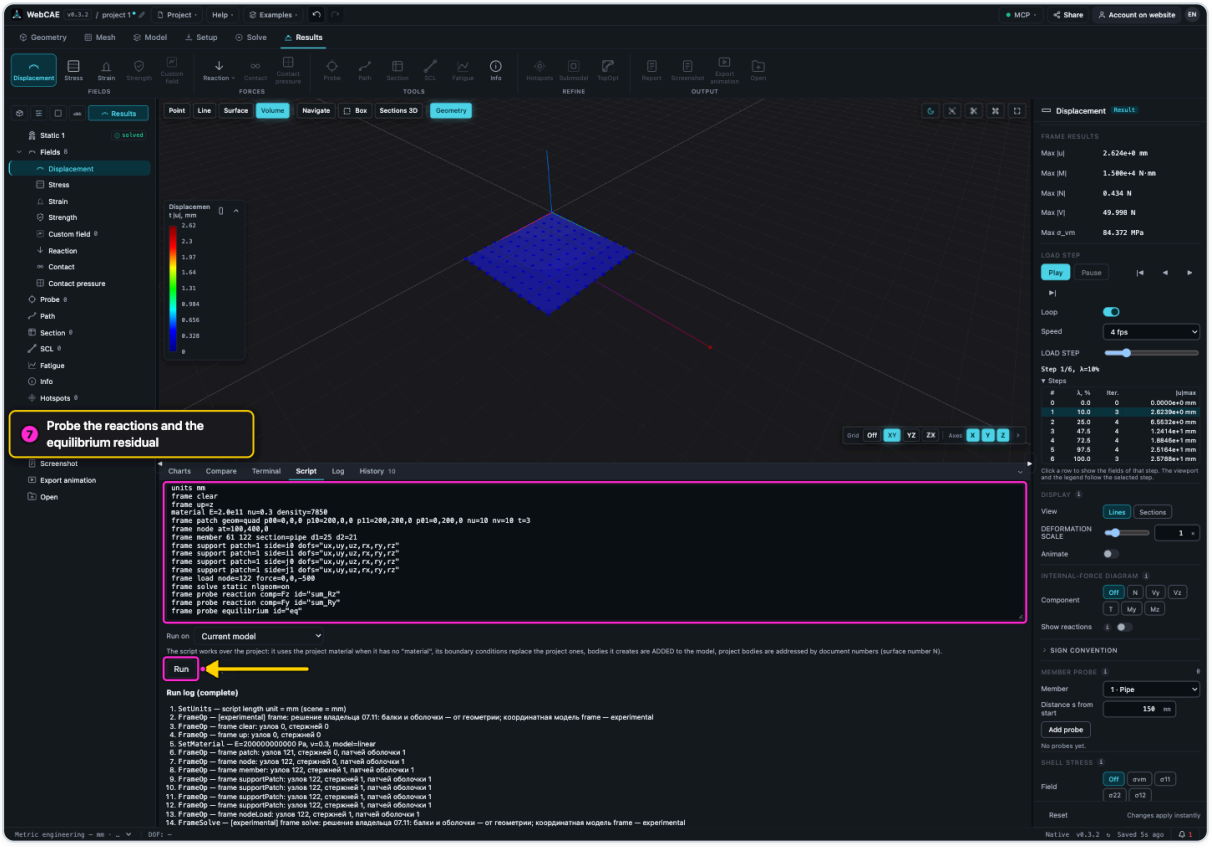


STEP 7 Probe the reactions and the equilibrium residual

`frame probe reaction comp=Fz` sums the vertical reactions of all supports: $500.000\ \mathrm{N}$ against the load of $500\ \mathrm{N}$. $\sum R_x$ and $\sum R_y$ vanish ($5 \cdot 10^{-13}$ and $2 \cdot 10^{-8}\ \mathrm{N}$). `frame probe equilibrium` returns the larger of the relative force residual and the relative moment residual of the whole model (reactions against loads, with positions): $0.0026 = 0.26\%$. Older versions of the probe `comp=Mx` summed only the **own moment components** of the supports ($-67.75\ \mathrm{N\ m}$ here), which is not a moment balance: the shear forces of the edges carry part of the moment as a couple. The probe now returns the moment of the reactions **about a point**, $\sum[M_i + (r_i - p) \times F_i]$: `about=x,y,z` sets the point (default: the origin, the same moment as in `frame probe equilibrium`; with `node=` — that node itself).

> **WHY** Force and moment balance of the whole model is the check that cannot be fooled by a rigid or a flexible support.

```
frame probe reaction comp=Fz id="sum_Rz"
frame probe reaction comp=Fy id="sum_Ry"
frame probe equilibrium id="eq"
```



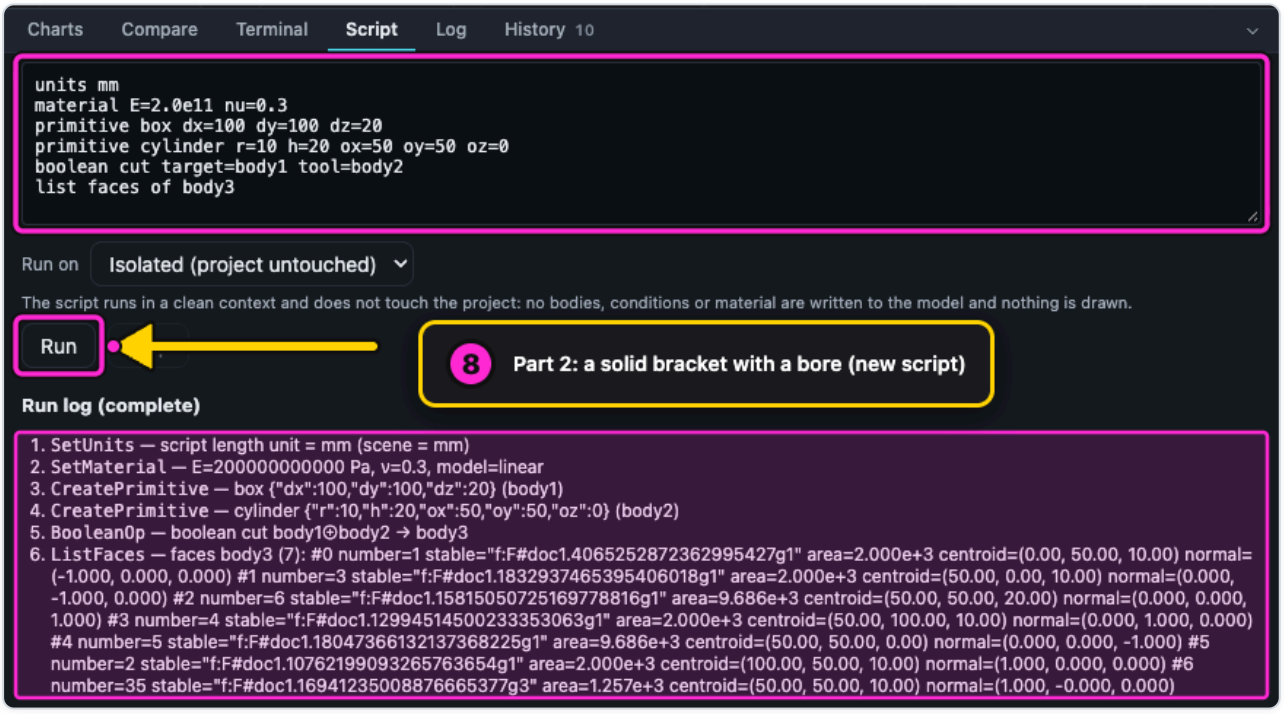
STEP 8

Part 2: a solid bracket with a bore (new script)

A new script, solid mode (no `frame` commands in it; set **Run on** to **Isolated (project untouched)** so the face numbers start from a clean context): a $100 \times 100 \times 20$ mm block with a ⌀ 20 bore (cylinder of radius 10 cut from the block, `body3`). `list faces of body3` shows the numbering: the bottom face is `surface number 5` and the bore wall is `surface number 35`.

WHY A solid and a frame model are different solve modes: they need separate documents.

```
units mm
material E=2.0e11 nu=0.3
primitive box dx=100 dy=100 dz=20
primitive cylinder r=10 h=20 ox=50 oy=50 oz=0
boolean cut target=body1 tool=body2
list faces of body3
```



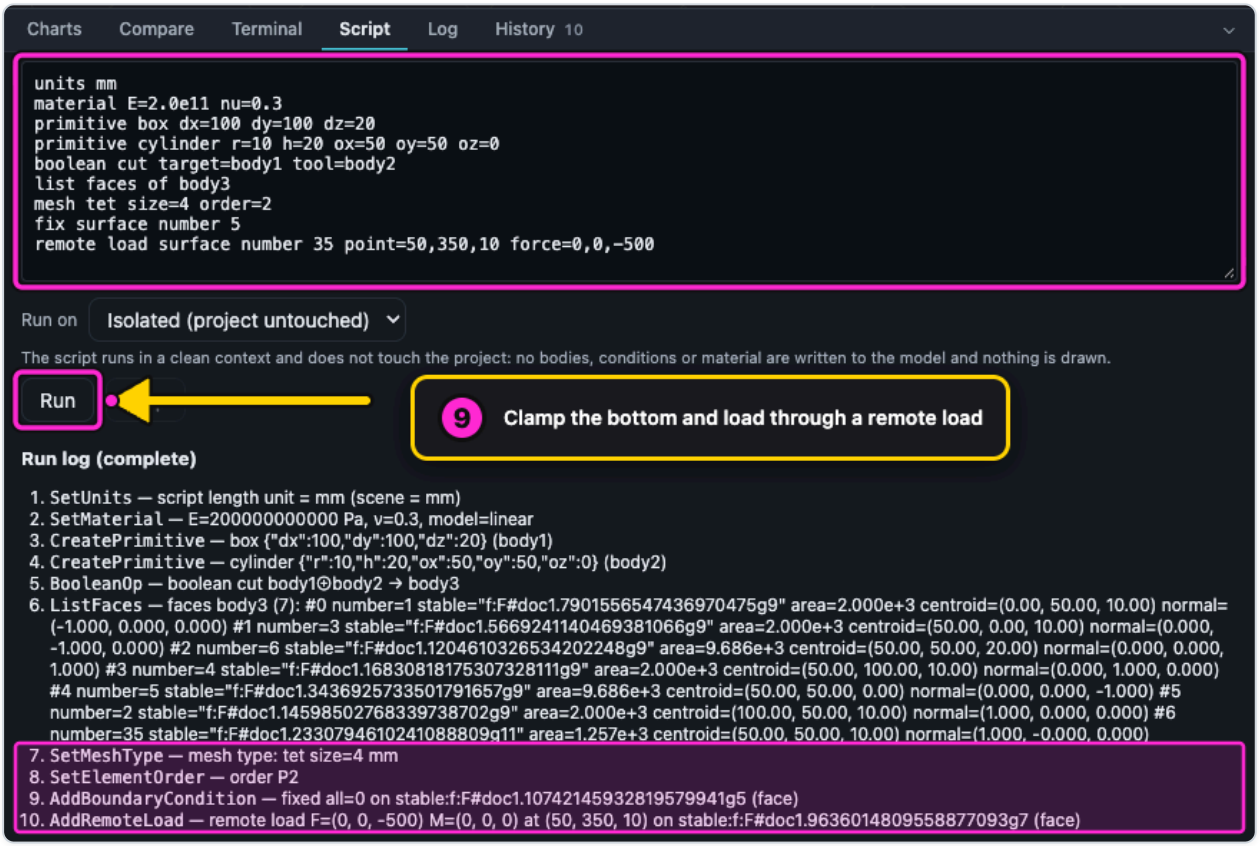
STEP 9

Clamp the bottom and load through a remote load

`fix surface number 5` clamps the bottom. `remote load surface number 35 point=50,350,10 force=0,0,−500` ties the bore wall rigidly to a virtual master point 300 mm from the bore axis (an RBE2-type rigid link) and applies 500 N downward there: the bracket sees a force and the moment $150\,000\ \mathrm{N\ mm}$. Mesh Tet10 with 4 mm elements.

> **WHY** A remote load models a lever without meshing it; the rigid link stiffens the bore wall, which is a property of an RBE2 link, not of the bracket.

```
mesh tet size=4 order=2
fix surface number 5
remote load surface number 35 point=50,350,10 force=0,0,−500
```



STEP 10 Solve and check the reactions

`solve`, then `probe reaction_z` : $\sum R_z = 500.0001\ \mathrm{N}$ against $500\ \mathrm{N}$ (+3e-5%), $\sum R_y = -5 \cdot 10^{-5}\ \mathrm{N}$. The peak von Mises stress is $35.7\ \mathrm{MPa}$ (no analytical reference: it sits at the clamp and at the link). Solid reaction probes return forces only, not moments.

> **WHY** The force balance of a solid model with a rigid link is exact to solver precision; the stress peaks are not part of this check.

```
solve
probe reaction_z id="Rz"
probe reaction_y id="Ry"
minmax vonMises
```



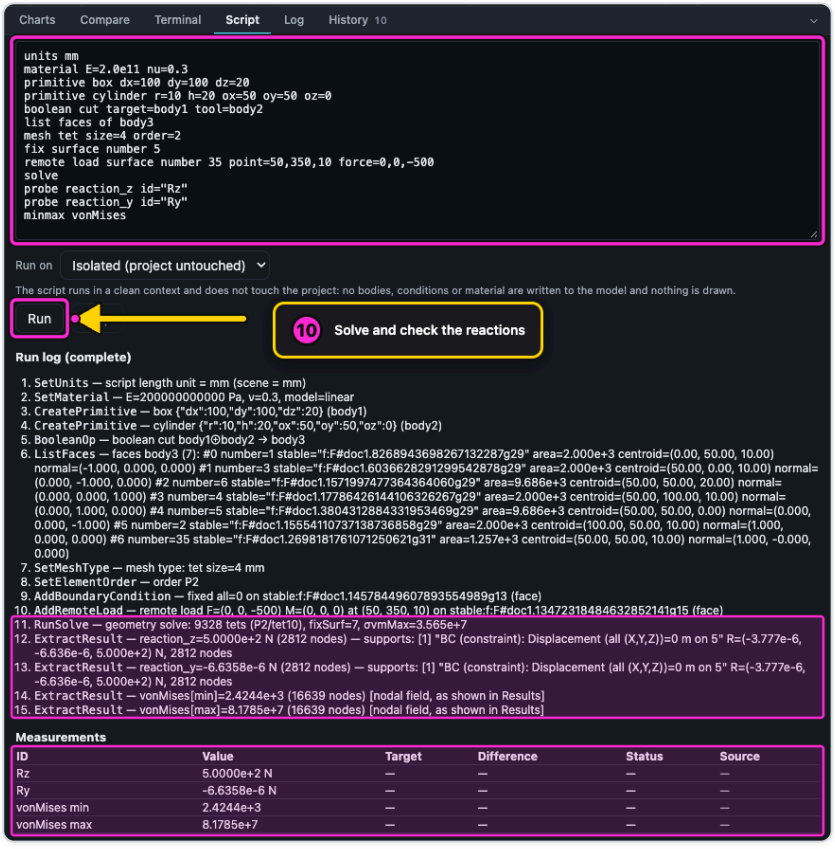
VERIFICATION BENCHMARK

Check against the formula

Quantity	Expected	WebCAE (desktop/native)	Difference
Shell plate + lever: $\sum R_z$, $P = 500$ N	500.000 N	500.000 N	0.00%
Shell plate + lever: force and moment balance residual	0	0.26% (0.0026)	—
Solid bracket: $\sum R_z$	500.000 N	500.0001 N	0.00%
Solid bracket: $\sum R_y$	0	-5e-5 N	—

FORMULA

$$\sum R_z = -P = 500\ \text{N}, \quad \sum R_x = \sum R_y = 0, \quad \text{moment about any point: } \sum(\mathbf{r} \times \mathbf{R} + \mathbf{M}_R) + \mathbf{r}_P \times \mathbf{P} = 0$$



TROUBLESHOOTING & NOTES

Common mistakes

- `frame support patch=… side=all`: fixes every node of the patch, not the rim; the plate turns rigid and the tip deflection is the beam value 2.35 mm instead of 25.8 mm.
- **Probing reactions after a linear** `frame solve static`: it refuses under shells; use `frame solve static nlgeom=on`.
- **Reading** `comp=Mx` **as the moment balance:** it sums only the reaction moments; the vertical reactions also carry a couple.
- **Mixing frame commands and solid bodies in one script:** they are different modes; use two scripts.
- **Expecting RBE3 behaviour from** `remote load`: the link is rigid (RBE2); a distributing RBE3 element does not exist in the product.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#27 — Unilateral Contact: Hertz Line Contact of a Cylinder on a Block (F27)** to solve frictionless unilateral contact and verify the Hertz pressure profile.

WebCAE 101

LESSON 27 · F27

Unilateral Contact: Hertz Line Contact of a Cylinder on a Block, Contact Pressure

Goal Solve a frictionless contact between a steel cylinder segment (R = 100 mm) and a steel block in plane strain, refine the mesh at the contact line so that the half-width of the contact patch is resolved by at least four elements, and compare the peak contact pressure with the Hertz solution.

Outcome A two-body contact model whose peak contact pressure is $601.2\ \mathrm{MPa}$ against the Hertz $p_0 = 591.4\ \mathrm{MPa}$ (+1.65%) on a 0.25 mm mesh at the contact line (about four elements across the half-width $b = 1.076\ \mathrm{mm}$), and $574.0\ \mathrm{MPa}$ (−2.95%) read through 0.3 mm probe windows. A 0.5 mm mesh (two elements across b) reads +6.3% and is outside the 5% budget, so the lesson uses 0.25 mm. The contact force balance is known to be short by about 1% inside the solver (see the check table).

10–12 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F27

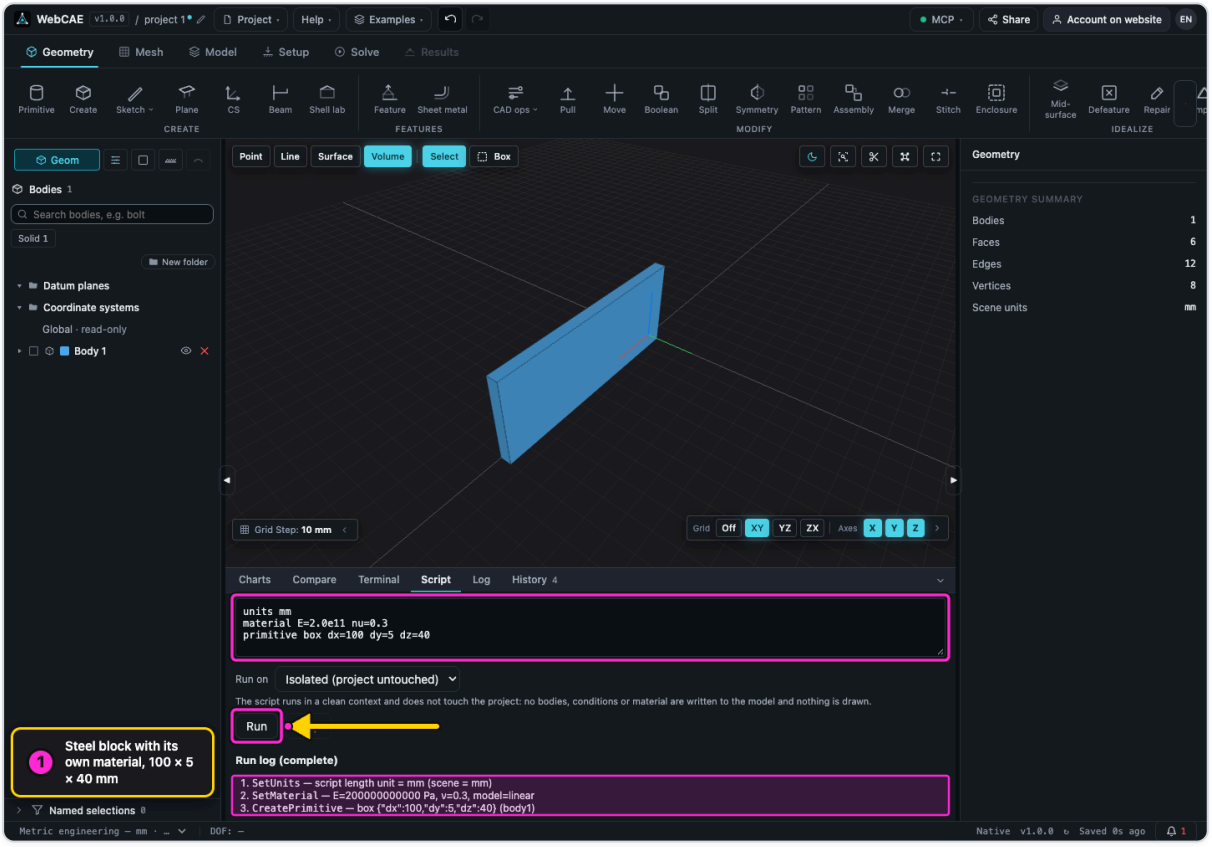
STEP 1 Steel block with its own material, 100 × 5 × 40 mm

Open the bottom dock, tab **Script**, set **Run on** to **Isolated (project untouched)**, paste the code blocks of the steps in order into one script and click **Run**.

Units mm. The block is $100 \times 5 \times 40$ mm: the 5 mm thickness (Y) is a plane-strain slice. Steel: $E = 2.0 \times 10^{11}$ Pa, $\nu = 0.3$; the combined plane-strain contact modulus is $E^* = \frac{E}{2(1-\nu^2)} = 109\,890$ MPa. **`material` goes before the `primitive` it belongs to**: the snapshot is taken when the body is created, and `material` has no `body=` argument.

> **WHY** Cylinder-on-flat contact has an exact Hertz solution, which makes it the standard benchmark for a contact formulation.

```
units mm
material E=2.0e11 nu=0.3
primitive box dx=100 dy=5 dz=40
```

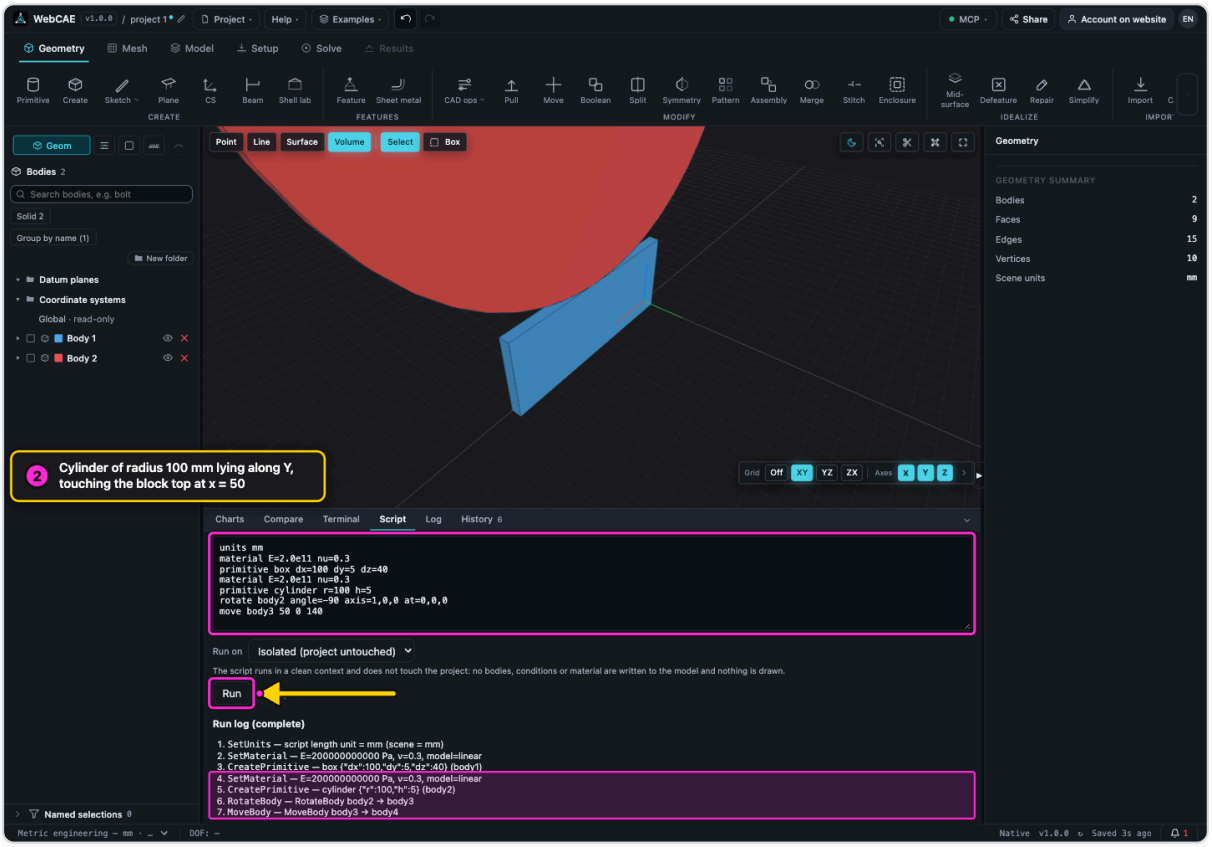


STEP 2 Cylinder of radius 100 mm lying along Y, touching the block top at x = 50

A cylinder of $R = 100\ \text{mm}$ and length 5 mm is created along Z, turned by -90° about X so that its axis runs along Y, and moved by `move body3 50 0 140`: the lowest generatrix is the line $x = 50,\ z = 140 - 100 = 40$, tangent to the top of the block.

WHY The two bodies touch along a line, the Hertz configuration; there is no penetration to start from.

```
material E=2.0e11 nu=0.3
primitive cylinder r=100 h=5
rotate body2 angle=-90 axis=1,0,0 at=0,0,0
move body3 50 0 140
```



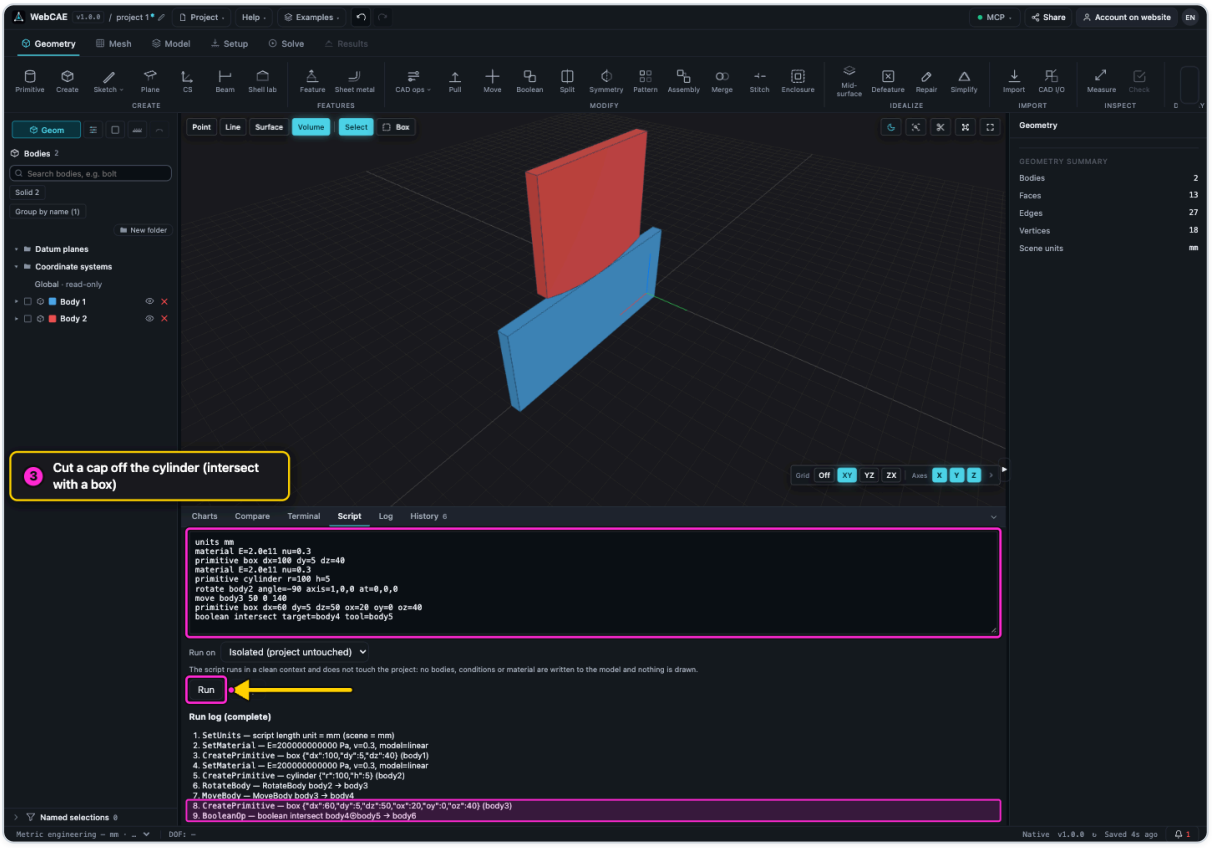
STEP 3

Cut a cap off the cylinder (intersect with a box)

A full cylinder is far larger than needed. A box $60 \times 5 \times 50$ mm at $(20, 0, 40)$ is intersected with it (`boolean intersect target=body4 tool=body5`): the cap `body6` is the lower part of the cylinder, 50 mm high, with a flat top at $z = 90$ mm for the load.

> **WHY** A small cap keeps the mesh small without changing the contact: the stress field at the line decays within a few contact widths.

```
primitive box dx=60 dy=5 dz=50 ox=20 oy=0 oz=40
boolean intersect target=body4 tool=body5
```

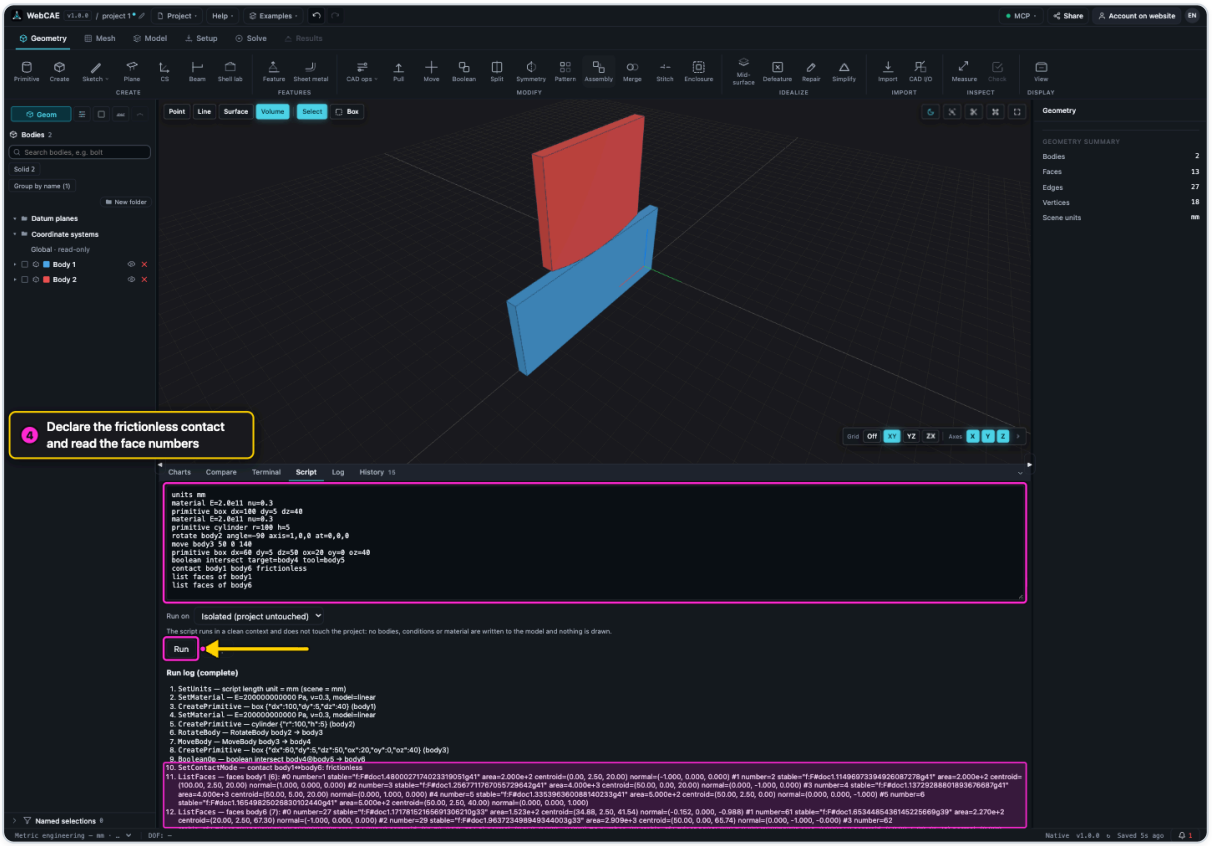


STEP 4 Declare the frictionless contact and read the face numbers

`contact body1 body6 frictionless` creates the unilateral pair (the slave side is the second body, the cap). `list faces` prints the numbering used below: the block bottom is `surface number 5`, the two block side faces (Y) are `surface number 3` and `4`, the two cap side faces (Y) are `surface number 28` and `29`, the cap top is `surface number 63`. These are the document numbers of **this exact command order**; a different order shifts them.

> **WHY** Unilateral contact enforces $g_n \geq 0,\ p_n \geq 0,\ g_n p_n = 0$: the surfaces separate freely outside the patch.

```
contact body1 body6 frictionless
list faces of body1
list faces of body6
```

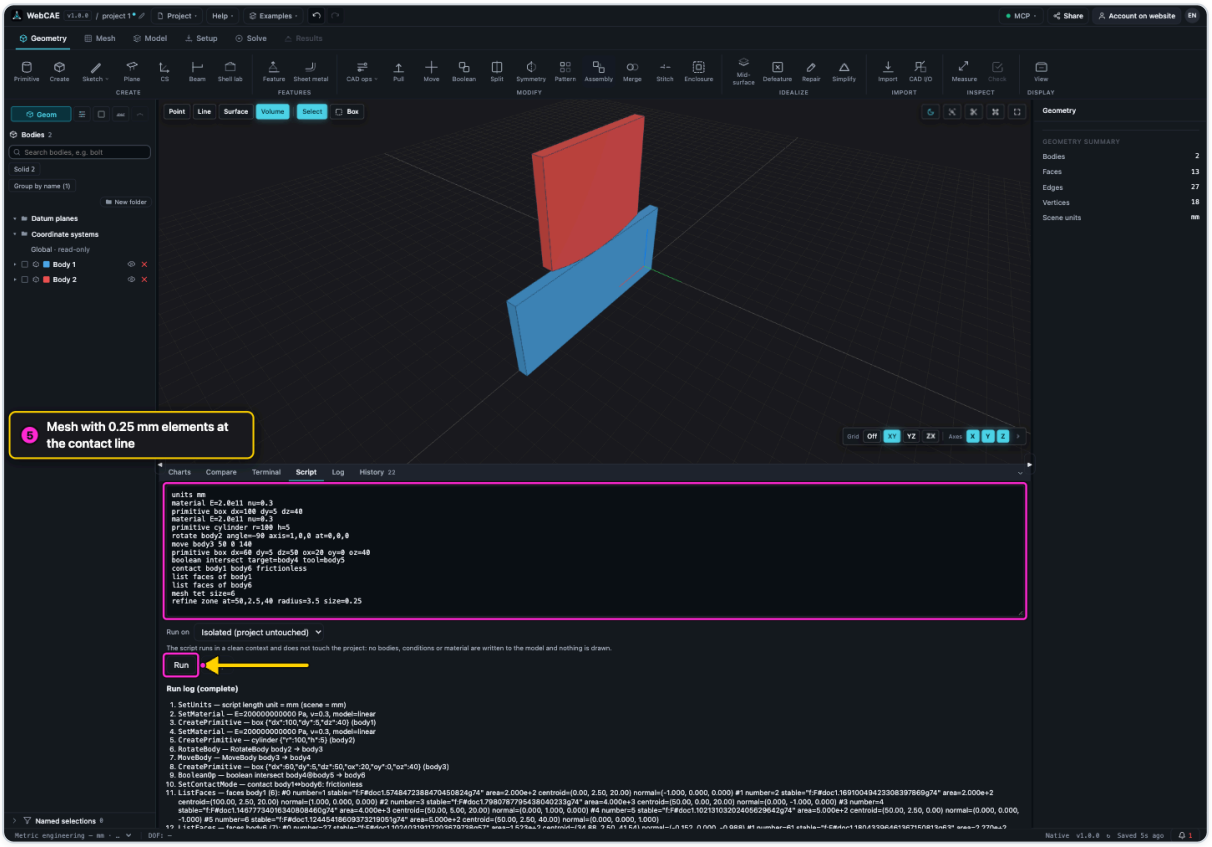


STEP 5 Mesh with 0.25 mm elements at the contact line

`mesh tet size=6` sets the global size and `refine zone at=50,2.5,40 radius=3.5 size=0.25` puts 0.25 mm elements in a sphere around the contact line. The Hertz half-width is $b = \sqrt{\frac{4wR}{\pi E^*}}$ with $w = 1000\ \mathrm{N/mm}$ (5000 N on the 5 mm slice) $= \sqrt{\frac{4 \cdot 1000 \cdot 100}{\pi \cdot 109\,890}} = 1.0764\ \mathrm{mm}$, so 0.25 mm gives $b/h \approx 4.3$. **The pressure peak needs** $b/h \gtrsim 4$: with 0.5 mm ($b/h \approx 2$) the peak reads $628.7\ \mathrm{MPa}$, $+6.3\%$ over Hertz.

> **WHY** The contact pressure is a nodal force divided by an area: with only two elements across the patch the peak is resolved too coarsely.

```
mesh tet size=6
refine zone at=50,2.5,40 radius=3.5 size=0.25
```

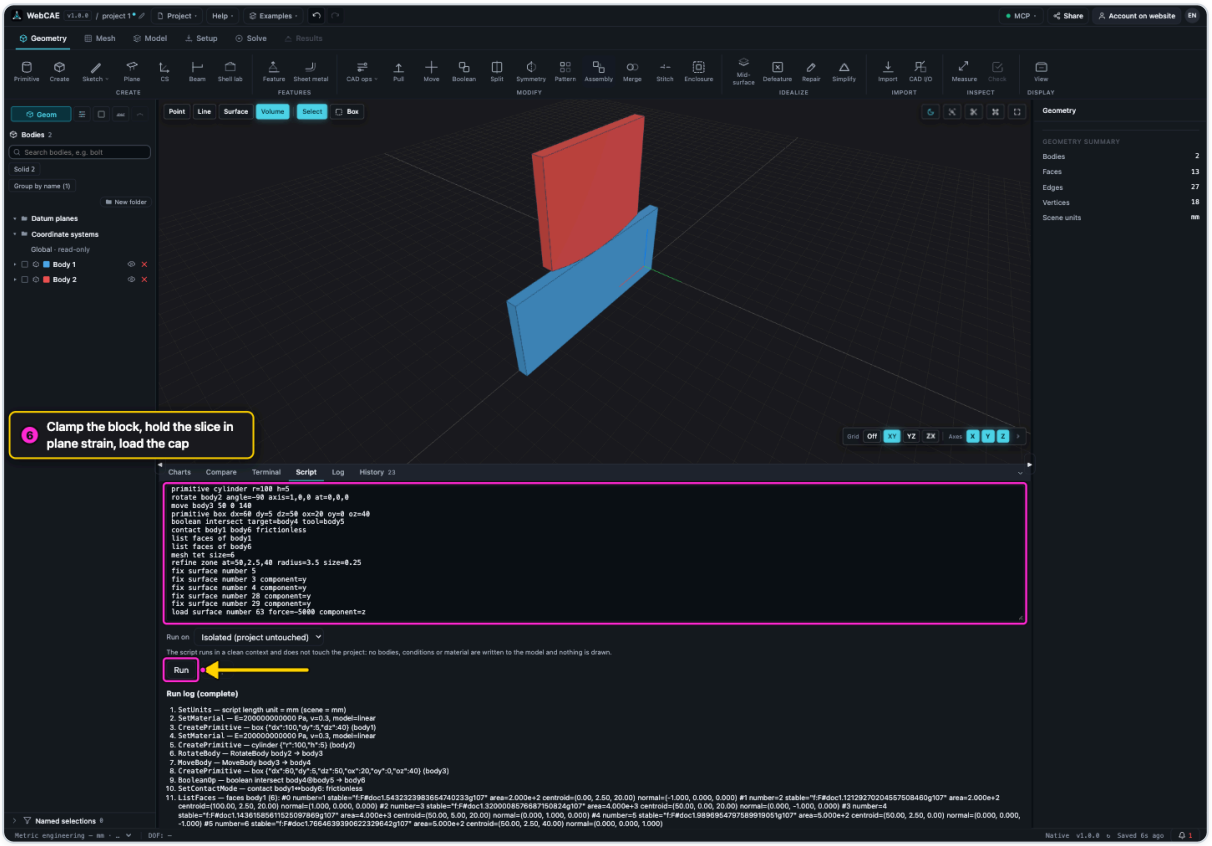


STEP 6 Clamp the block, hold the slice in plane strain, load the cap

`fix surface number 5` clamps the block bottom. The four Y-faces (`surface number 3`, `4`, `28`, `29`) get `component=y`: $u_y = 0$ on all of them makes the 5 mm slice a plane-strain strip. `load surface number 63 force=-5000 component=z` presses the cap top with $5000\ \mathrm{N}$, i.e. $w = 5000/5 = 1000\ \mathrm{N/mm}$.

WHY Plane strain is what the Hertz line-contact formula assumes.

```
fix surface number 5
fix surface number 3 component=y
fix surface number 4 component=y
fix surface number 28 component=y
fix surface number 29 component=y
load surface number 63 force=-5000 component=z
```

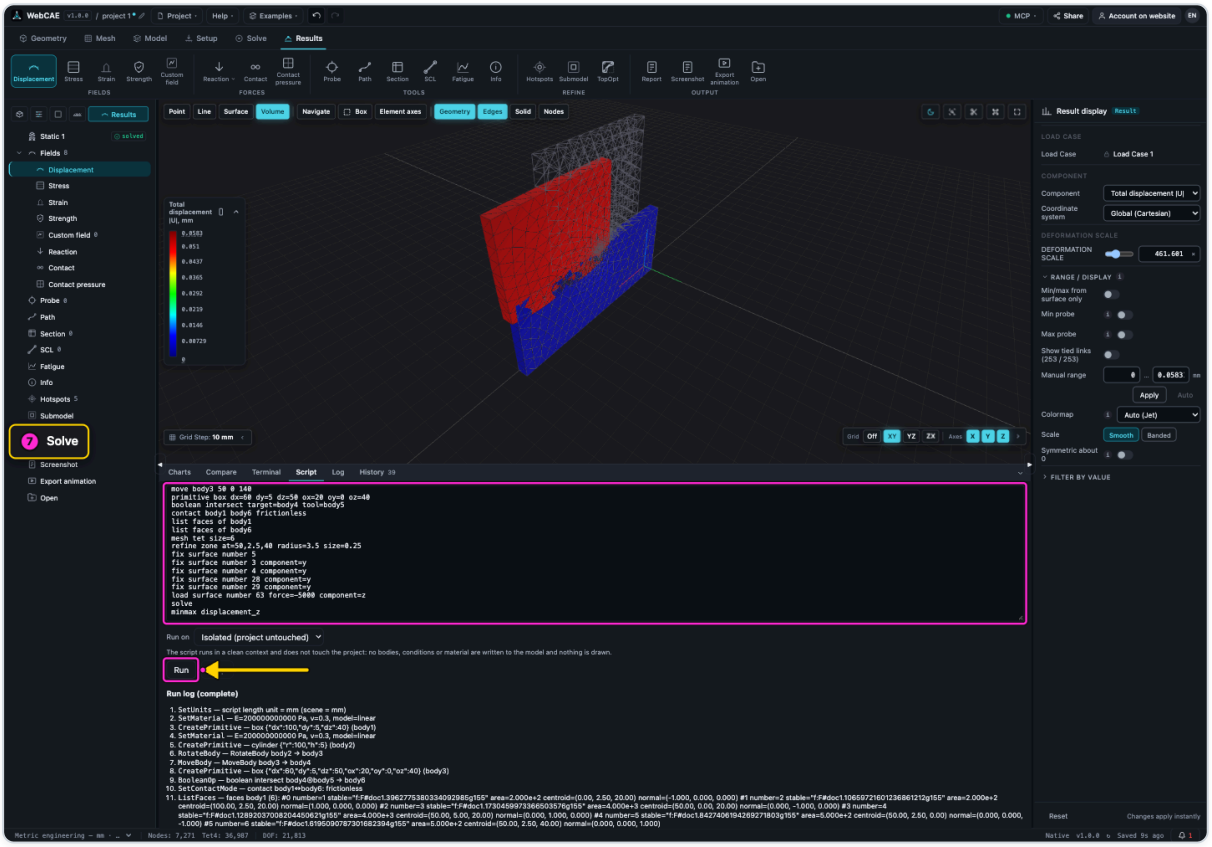


STEP 7 Solve

`solve` runs the nonlinear contact solution (about 37 000 tetrahedra, roughly 100 s native). The log names the assembly branch: `assembly contact: body1↔body6=frictionless`. `minmax displacement_z` gives the approach: $-0.0268\ \mathrm{mm}$ at the cap top.

> **WHY** The pair is solved as an assembly with the contact active set; both bodies take part (no silent drop of the second body).

```
solve
minmax displacement_z
```

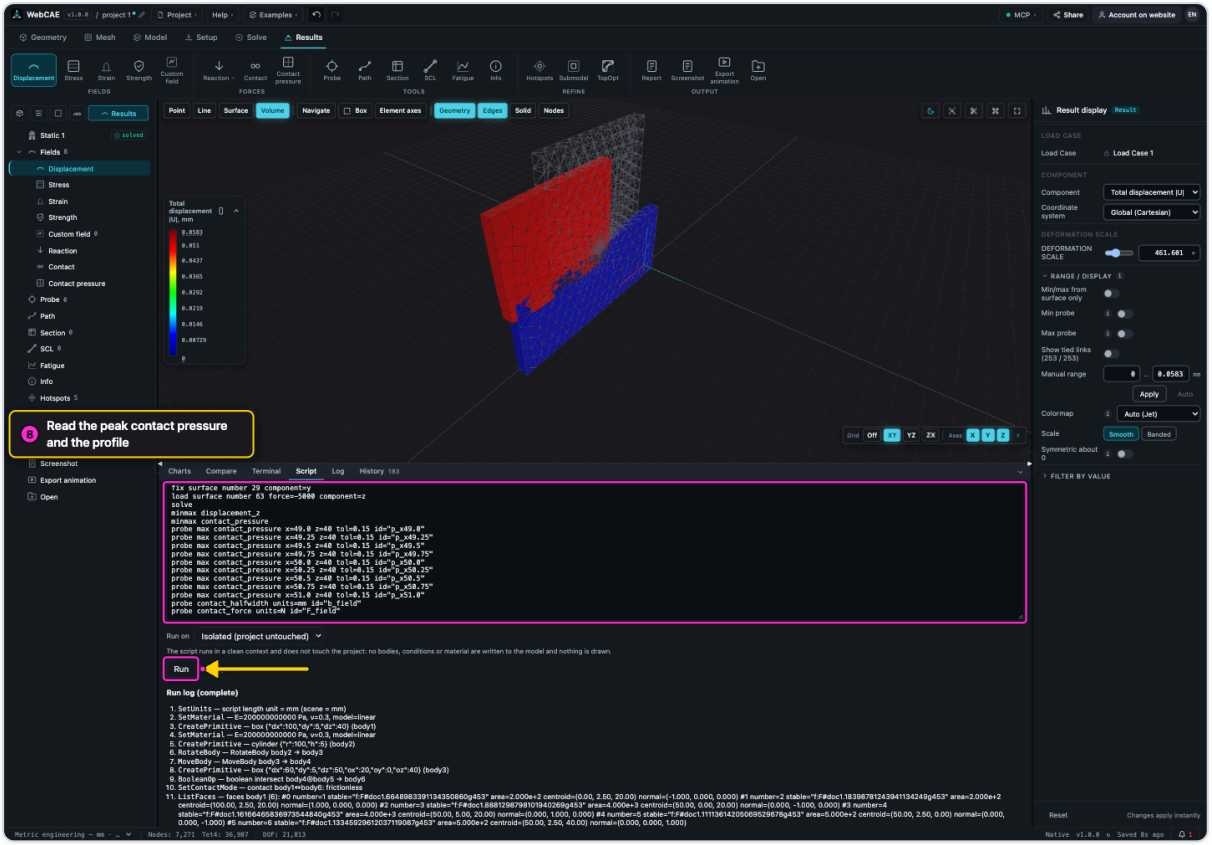


STEP 8 Read the peak contact pressure and the profile

`minmax contact_pressure` returns the maximum of the contact-pressure field: $601.2\ \mathrm{MPa}$ against the Hertz $p_0 = \frac{2w}{\pi b} = \frac{2 \cdot 1000}{\pi \cdot 1.0764} = 591.4\ \mathrm{MPa}$ (+1.65%). The window probes (`tol=0.15`, 0.3 mm wide) read the profile along x: $222.7,\ 401.5,\ 464.6,\ 545.4,\ \mathbf{574.0},\ 548.9,\ 484.4,\ 396.2,\ 114.4\ \mathrm{MPa}$ at $x = 49.0 \ldots 51.0$ (step 0.25). The centre window reads $574.0\ \mathrm{MPa}$ (−2.95%): windows average a 0.3 mm strip and cut the peak. Two more probes read the pressure field itself instead of windows: `probe contact_halfwidth` returns the half-width of the contact zone, $b = 2\sqrt{M_2/M_0}$ from the second moment of the pressure field across the strip ($1.112\ \mathrm{mm}$ against $1.0764\ \mathrm{mm}$, +3.3%), and `probe contact_force` returns $\sum p_i A_i$ over the contact nodes ($4950.5\ \mathrm{N}$ against the $5000\ \mathrm{N}$ load, −0.99%). The 0.25 mm window profile is not used for these two: sampling the profile in 0.25 mm windows reads a half-width of $1.0\ \mathrm{mm}$ (-7.1%) and a trapezoid sum of $4701.5\ \mathrm{N}$ (-5.97%), which are limits of that sampling, not of the field (shown in the table for reference only).

WHY The field maximum, the field half-width and the field force are each within 5% of Hertz; the windowed values are shown only to explain why windows are the wrong tool for b and $\int p$.

```
minmax contact_pressure
probe max contact_pressure x=49.0 z=40 tol=0.15 id="p_x49.0"
probe max contact_pressure x=49.25 z=40 tol=0.15 id="p_x49.25"
probe max contact_pressure x=49.5 z=40 tol=0.15 id="p_x49.5"
probe max contact_pressure x=49.75 z=40 tol=0.15 id="p_x49.75"
probe max contact_pressure x=50.0 z=40 tol=0.15 id="p_x50.0"
probe max contact_pressure x=50.25 z=40 tol=0.15 id="p_x50.25"
probe max contact_pressure x=50.5 z=40 tol=0.15 id="p_x50.5"
probe max contact_pressure x=50.75 z=40 tol=0.15 id="p_x50.75"
probe max contact_pressure x=51.0 z=40 tol=0.15 id="p_x51.0"
probe contact_halfwidth units=mm id="b_field"
probe contact_force units=N id="F_field"
```

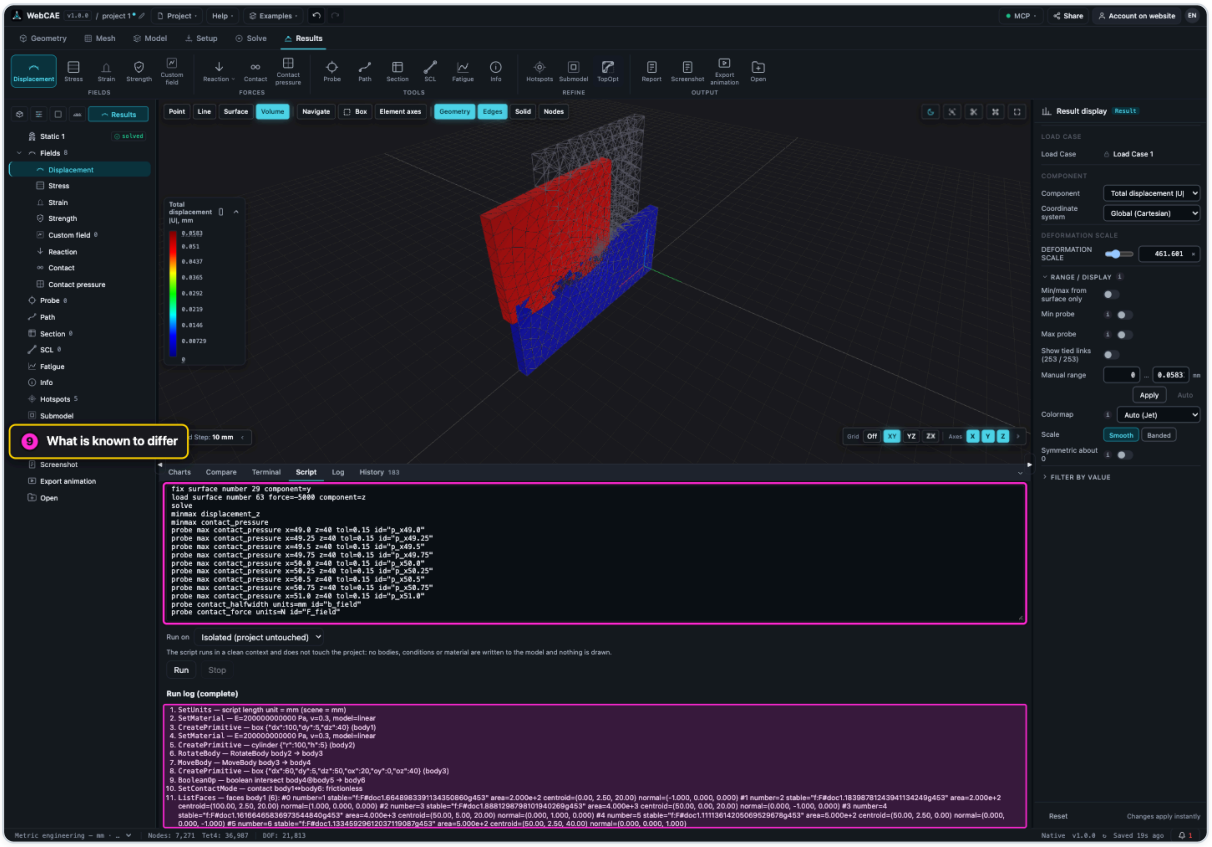

Read the peak contact pressure and the profile
Run
Run log (complete)

STEP 9

What is known to differ

Two things differ from the ideal and are worth knowing. (1) The contact force is read from the pressure field and equals the solver total by construction (the post-processor renormalizes the field to $\sum F_i$ of the solver), so the -0.99% is the solver itself: the total is $4950.5\ \mathrm{N}$ against the $5000\ \mathrm{N}$ load because the contact post-processing of the solver misses its floor-stiffness term by $1/1.01$ (a known core discrepancy, not fixed). (2) The half-width is a second-moment estimate over the discrete contact nodes: it reads $+3.3\%$ on this mesh and depends on the mesh as any nodal field does. The mesh dependence of the peak is scripted: changing the refinement line to `refine zone at=50,2.5,40 radius=2.5 size=0.15` ($b/h \approx 7$, 73 714 tetrahedra, about 285 s native) gives `minmax contact_pressure` $= 586.7\ \mathrm{MPa}$ (−0.80%). A 0.5 mm mesh is not part of the lesson (two elements across b, peak +6.3%).

WHY Stating which numbers the script reproduces and which it does not keeps the comparison honest.



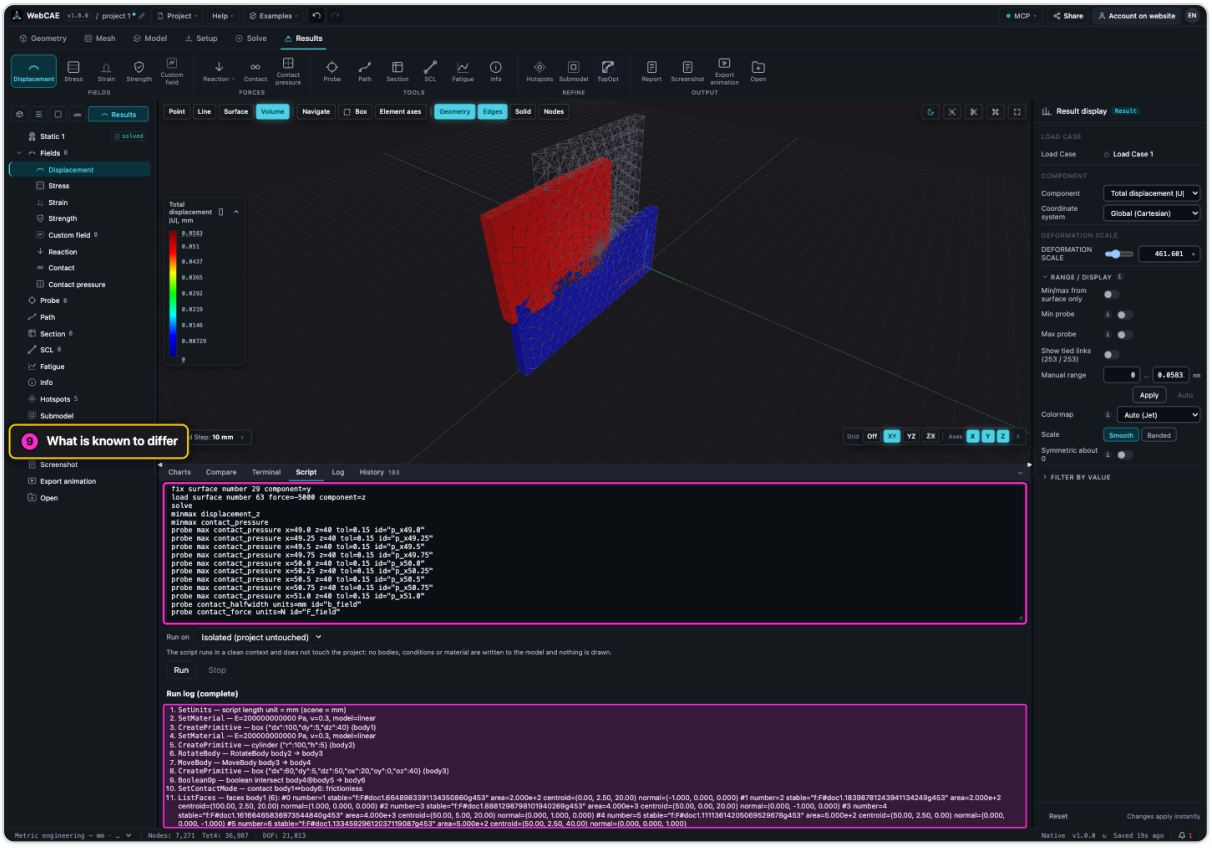
VERIFICATION BENCHMARK

Check against the formula

Quantity	Hertz analytical	WebCAE (desktop/native)	Difference
Peak contact pressure, field maximum (`minmax contact_pressure` , 0.25 mm)	$p_0 = \frac{2w}{\pi b} = 591.4\ \mathrm{MPa}$	601.2 MPa	+1.65%
Peak contact pressure, centre window of the profile (0.3 mm wide)	$591.4\ \mathrm{MPa}$	574.0 MPa	-2.95%
Peak, finer mesh 0.15 mm (b/h about 7, `minmax contact_pressure`)	$591.4\ \mathrm{MPa}$	586.7 MPa	-0.80%
Peak on a 0.5 mm mesh (b/h about 2), for contrast	$591.4\ \mathrm{MPa}$	628.7 MPa	+6.3%
Half-width, from the pressure field (`probe contact_halfwidth` , second moment)	$b = 1.0764\ \mathrm{mm}$	1.112 mm	+3.3%
Contact force from the pressure field, $\sum p_i A_i$ (`probe contact_force`)	$5000\ \mathrm{N}$	4950.5 N	-0.99%

FORMULA

Hertz line contact (plane strain). $E^* = \frac{E}{2(1-\nu^2)} = 109\,890\ \mathrm{MPa}$, $R^* = 100\ \mathrm{mm}$, load per length $w = 1000\ \mathrm{N/mm}$: $b = \sqrt{\frac{4wR^*}{\pi E^*}} = 1.0764\ \mathrm{mm}$, $p_0 = \frac{2w}{\pi b} = 591.4\ \mathrm{MPa}$, $p(x) = p_0\sqrt{1-(x/b)^2}$, $\int_{-b}^{b} p\,dx = w$.



TROUBLESHOOTING & NOTES

Common mistakes

- **Using a 0.5 mm mesh (two elements across the half-width):** the peak reads +6.3%; refine to at least four elements across b (0.25 mm here).
- **Reading the peak through wide windows:** a 0.3 mm window cuts the peak by about 3% and a 0.25 mm profile step gives 7% on the half-width and 6% on the force; use `minmax contact_pressure` for the peak and `probe contact_halfwidth` / `probe contact_force` for the patch.
- **`material … body=N`:** the argument does not exist and is silently ignored; the material must precede its `primitive`.
- **Leaving the Y-faces free:** without $u_y = 0$ on all four Y-faces the slice is not in plane strain and the Hertz comparison does not apply.
- **Confusing tied and frictionless contact:** tied contact glues the bodies and carries tension across the patch; use `contact … frictionless` for a unilateral patch.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#28 — Snap-Arm Release Force** to work with prescribed displacements and support reactions on a solid cantilever.

WebCAE 101

LESSON 28 · F28

Snap-Arm Release Force: Prescribed Displacement and Support Reactions

Goal Compute the force needed to deflect a flexible polymer snap arm by its undercut depth, using a displacement-driven boundary condition and the support reaction instead of a prescribed force, and compare it with the cantilever beam formula.

Outcome A solid cantilever snap arm ($L = 30\ \mathrm{mm}$, $b = 10\ \mathrm{mm}$, $h = 2\ \mathrm{mm}$, POM) deflected by $\delta = 1.2\ \mathrm{mm}$ through its thickness with the release force read as a support reaction: $2.595\ \mathrm{N}$ (WebCAE desktop/native) against the beam value $F_{\mathrm{pinch}} = 3EI\delta/L^3 = 2.49\ \mathrm{N}$, difference 4.2%.

6–8 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F28

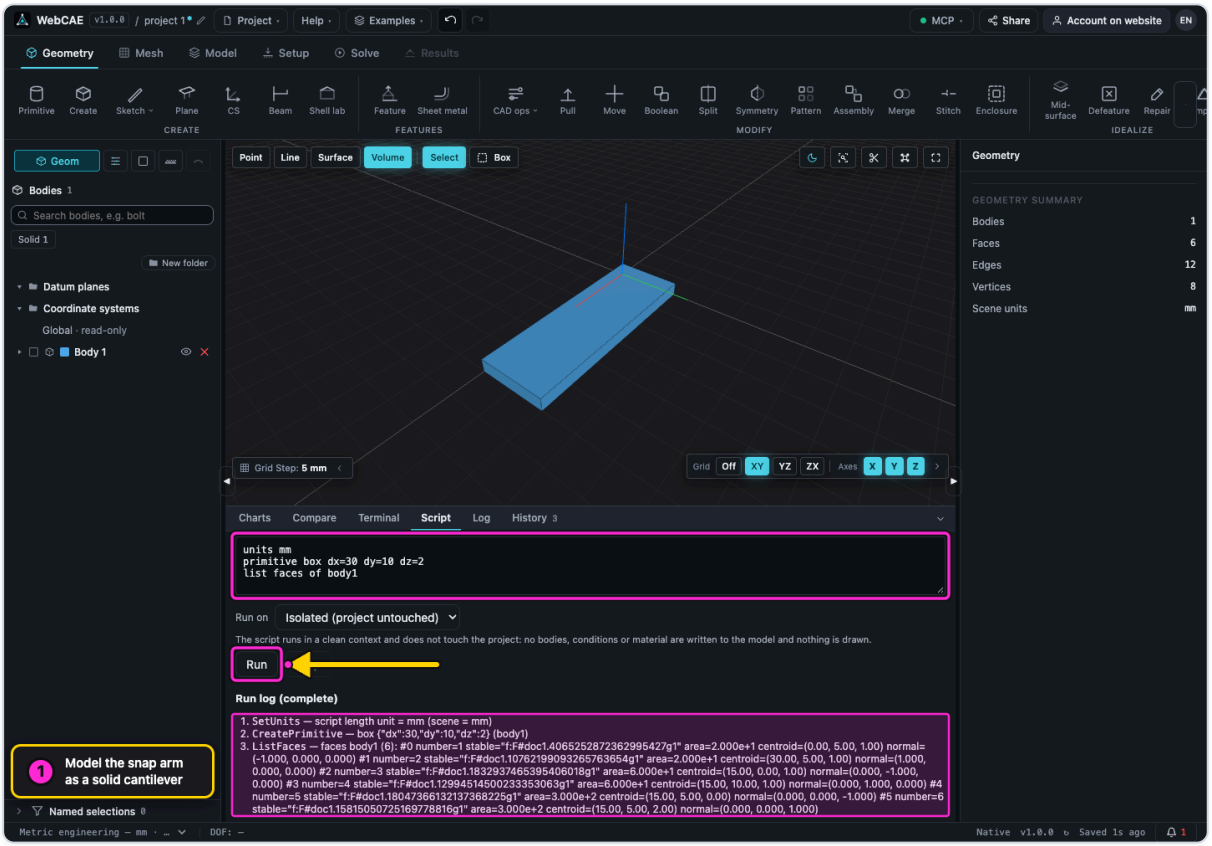
STEP 1 Model the snap arm as a solid cantilever

Open the bottom dock, tab **Script**, set **Run on** to **Isolated (project untouched)**, paste the code blocks of the steps in order into one script and click **Run**.

The flexible arm is a box $L = 30\ \mathrm{mm}$ along X, $b = 10\ \mathrm{mm}$ along Y and $h = 2\ \mathrm{mm}$ along Z. Its faces are numbered `surface number 1` (the root, $x = 0$) and `surface number 2` (the free end, $x = 30$); check them with `list faces of body1`. The barb wedge of a real clip is not modeled here: only the straight arm, which is what the beam formula describes.

WHY Cantilever snap-fits are ubiquitous in housings; the release force of the arm is the first design check, before any contact or insertion simulation.

```
units mm
primitive box dx=30 dy=10 dz=2
list faces of body1
```



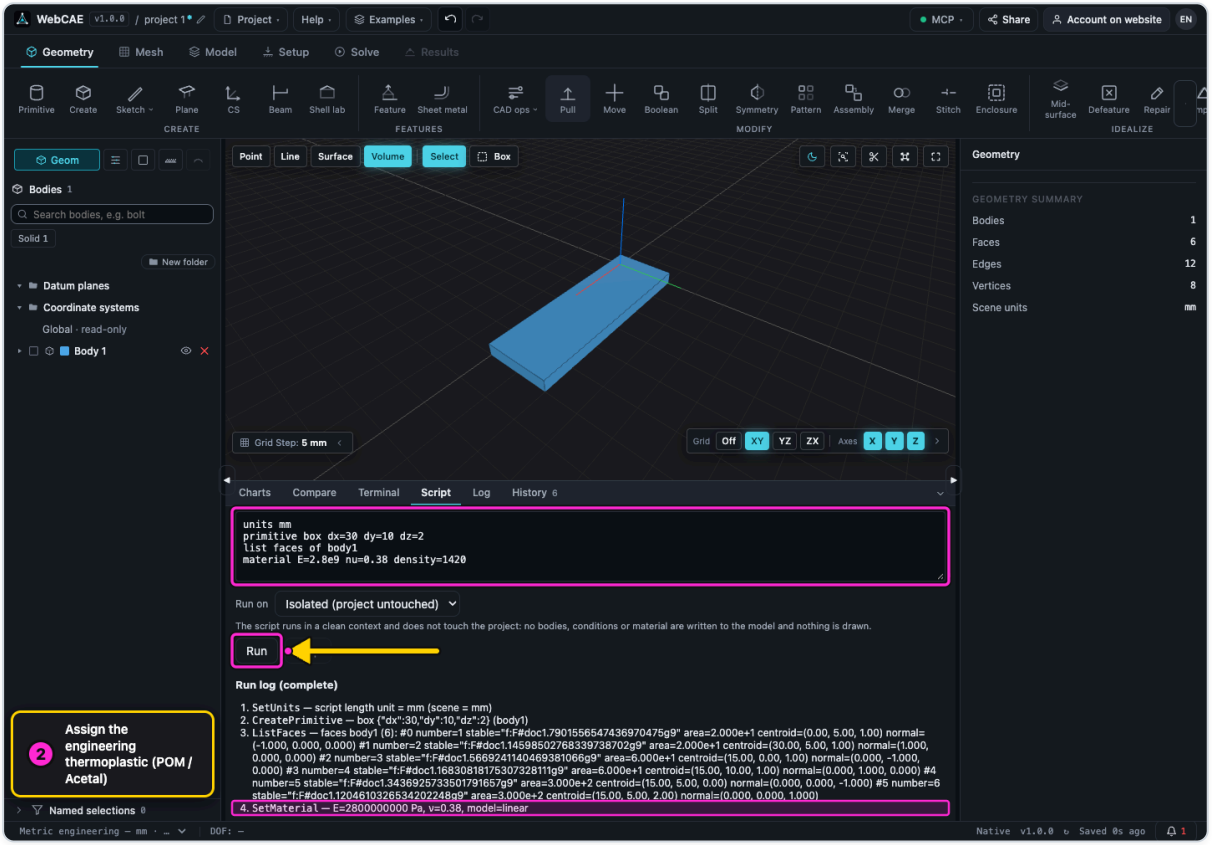
STEP 2

Assign the engineering thermoplastic (POM / Acetal)

Polyoxymethylene (Delrin): Young's modulus $E = 2800\ \mathrm{MPa} = 2.8 \times 10^9\ \mathrm{Pa}$ (the DSL takes Pascals), Poisson's ratio $\nu = 0.38$, density $1420\ \mathrm{kg/m^3}$; the yield strength $\sigma_y = 65\ \mathrm{MPa}$ is not used by the linear analysis.

WHY The low modulus of a polymer is what allows the large elastic deflection of a snap arm.

```
material E=2.8e9 nu=0.38 density=1420
```



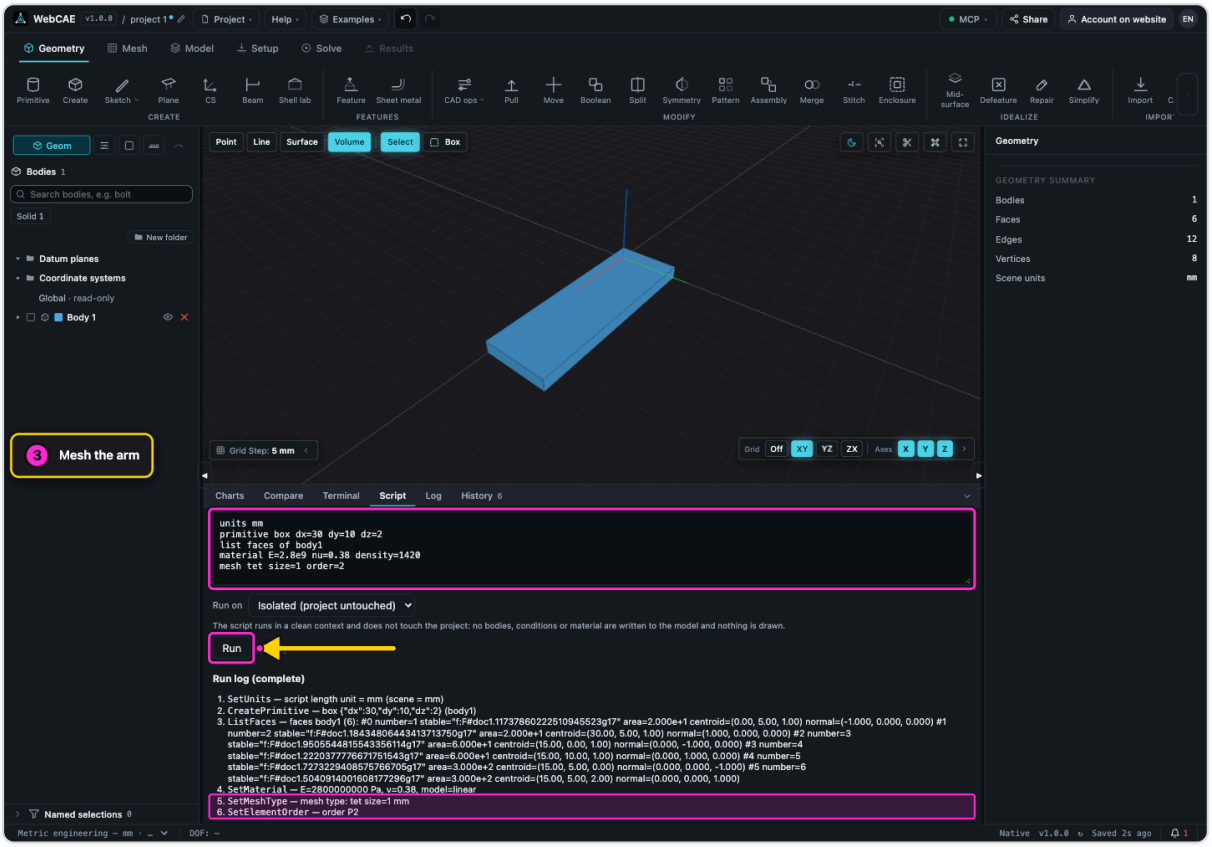
STEP 3

Mesh the arm

Quadratic tetrahedra with a 1 mm element size give 3595 Tet10 elements, two elements through the 2 mm thickness.

WHY Bending needs more than one quadratic element through the thickness; one layer would lock.

```
mesh tet size=1 order=2
```



STEP 4

Clamp the root and prescribe the tip displacement

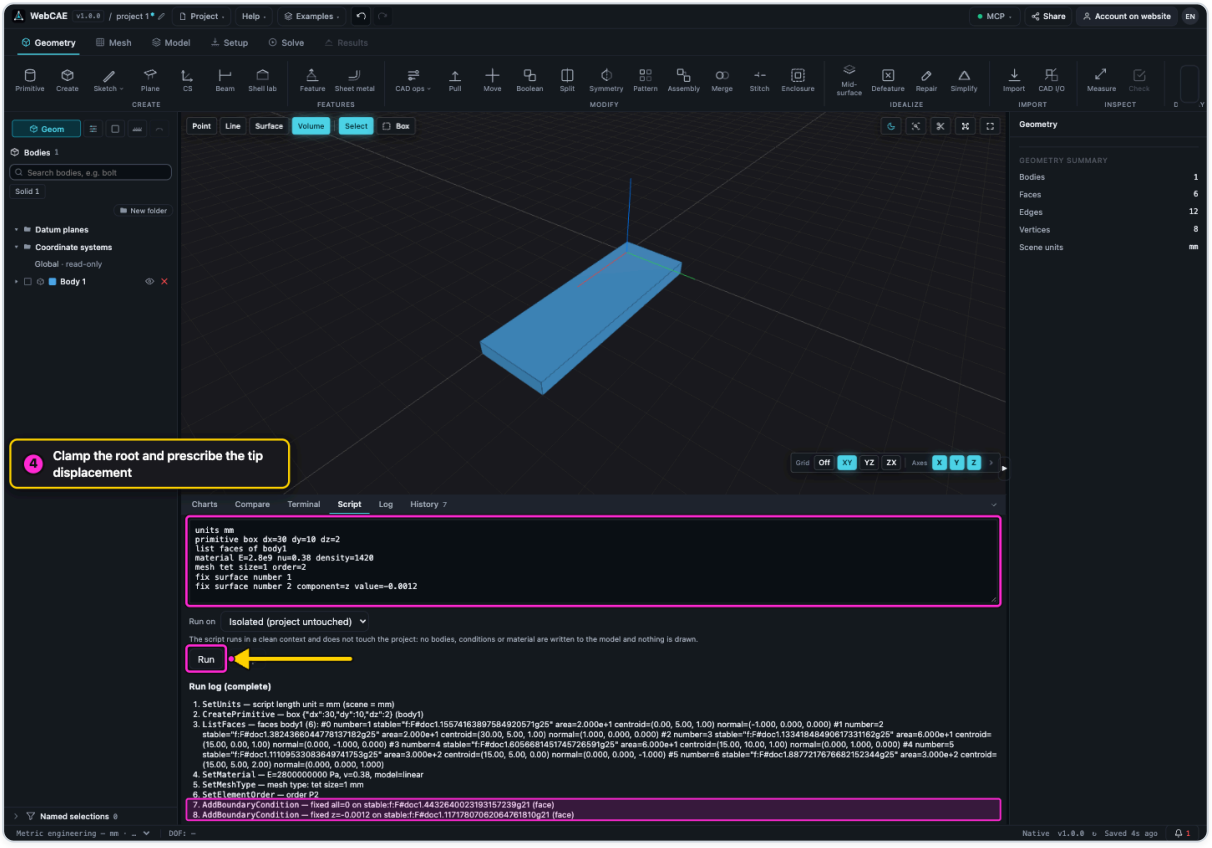
Fix the root face and push the free end down by the undercut depth $\delta = 1.2\ \mathrm{mm}$ through the thickness (the Z direction):

- `fix surface number 1` clamps the root.
- `fix surface number 2 component=z value=-0.0012` prescribes u_z on the tip face. **`value=` is in meters:** $-0.0012\ \mathrm{m} = -1.2\ \mathrm{mm}$; writing `value=-1.2` would impose 1.2 m.

Displacement control is the natural way to measure the force of a snap arm: it stays stable where a force-controlled run would run away at the peak of a snap-through.

> **WHY** A prescribed displacement gives the force as a reaction, exactly what a user feels when pressing the clip.

```
fix surface number 1
fix surface number 2 component=z value=-0.0012
```



STEP 5

Solve and read the support reactions

`solve` runs the linear static analysis; `reactions` prints the reaction of every support. The root reaction is $R_z = +2.595\ \mathrm{N}$ and the prescribed tip face carries $-2.598\ \mathrm{N}$ (the two balance to 0.1%); `minmax displacement_z` confirms the imposed $-1.2000\ \mathrm{mm}$.

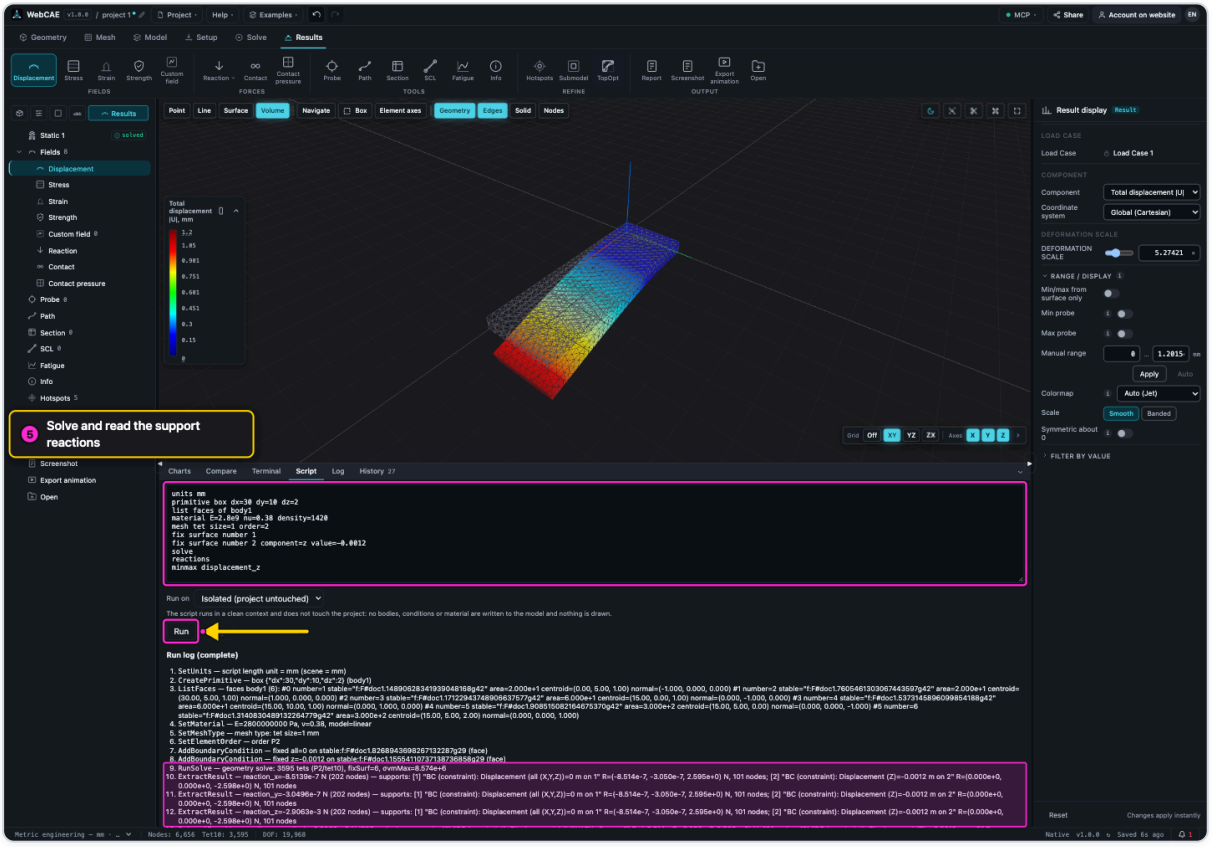
Beam theory: $I = bh^3/12 = 10 \cdot 2^3/12 = 6.667\ \mathrm{mm}^4$ (bending through the 2 mm thickness) and

$$F_{\mathrm{pinch}} = \frac{3EI\delta}{L^3} = \frac{3 \times 2800 \times 6.667 \times 1.2}{30^3} = 2.49\ \mathrm{N}$$

WebCAE gives $2.595\ \mathrm{N}$, which is 4.2% higher. The solid is stiffer than the 1D formula: the strip is wide ($b/h = 5$), so it is partly in plane strain (the plane-strain limit would add $1/(1-\nu^2) - 1 = 17\%$) and the clamped face also restrains the contraction of the root section.

WHY Pinch force verification shows whether servicing or battery removal stays comfortable without tools.

```
solve
reactions
minmax displacement_z
```



STEP 6

Insertion over a wedge with Coulomb friction (desktop, nonlinear static)

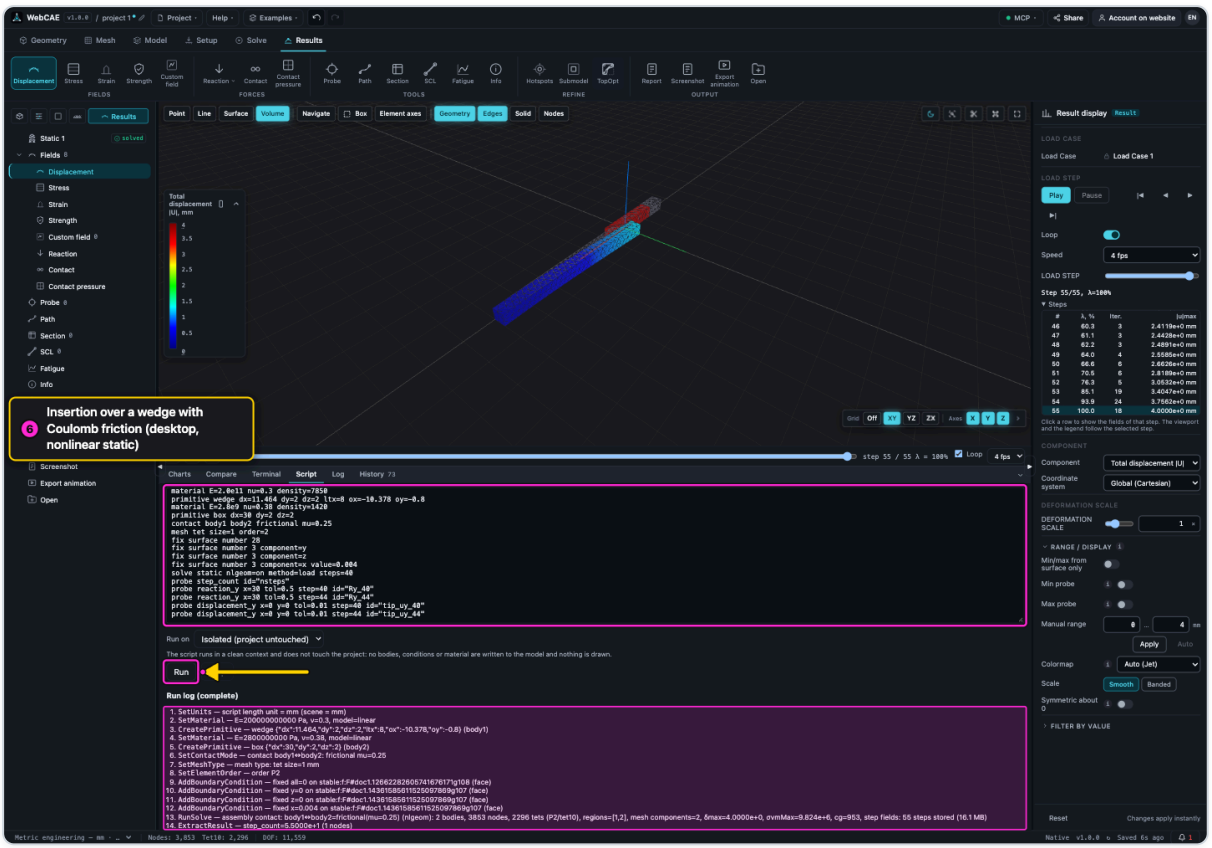
The release force above is read at rest. The insertion itself is a stroke: a steel wedge with a 30° ramp is driven along x under a clamped arm and bends it up. This is a separate, smaller model: arm $30 \times 2 \times 2$ mm (POM, $I = bh^3/12 = 1.333\ \text{mm}^4$), wedge from `primitive wedge` (ramp, then a flat top 1.2 mm above the arm), contact `frictional mu=0.25`, wedge driven by a prescribed displacement of 4 mm with `fix surface number 3 component=x value=0.004` (meters). **Coulomb friction in a static solve needs `nlgeom=on` and the desktop app**: without `nlgeom on` the solve is refused with the recipe, and in the browser it is refused as desktop-only, never run without friction. The run takes about 3 minutes natively.

`probe reaction_y x=30 tol=0.5 step=40` sums the nodal reactions of the nodes in the window $x = 30 \pm 0.5$ (the clamp) at load step 40; `probe displacement_y x=0 y=0 tol=0.01 step=40` reads the tip deflection at the same step. The clamp reaction is the bending force of the arm. At the end of the ramp (step 44, tip deflection $\delta = 1.1996$ mm) it is $R_y = 0.5021$ N against the beam value $3EI\delta/L^3 = 3 \cdot 2800 \cdot 1.333 \cdot 1.1996/27000 = 0.4977$ N (+0.9%); at step 40 ($\delta = 1.1889$ mm) 0.4965 N against 0.4932 N (+0.7%).

What is not compared. The wedge force along x, which by the Coulomb wedge law is $F = F_z(\tan\alpha + \mu)/(1 - \mu\tan\alpha) = 0.97\,F_z$ on the ramp and μF_z on the flat top, is not published as a check here: on this 1 mm mesh only two to four contact nodes carry the stroke and their stick/slip switching makes the settled steps read $F/F_z = 0.53$ to 0.55 with step-to-step scatter of 0.4 to 1.8. The law itself holds when sliding is not intermittent: at $\mu = 2$ the flat-top sliding force is $F/F_z = 1.98$ against $\mu = 2$ (-1.0%, checked by the script test of the desktop build); the wedge law of the core is verified at 0.00% on an aligned mesh in the solver tests.

> **WHY** The insertion force is what the user feels while snapping a clip in; the bending reaction during the stroke is the stable part of it, and the friction part needs a mesh with many nodes in contact.

```
units mm
material E=2.0e11 nu=0.3 density=7850
primitive wedge dx=11.464 dy=2 dz=2 ltx=8 ox=-10.378 oy=-0.8
material E=2.8e9 nu=0.38 density=1420
primitive box dx=30 dy=2 dz=2
contact body1 body2 frictional mu=0.25
mesh tet size=1 order=2
fix surface number 28
fix surface number 3 component=y
fix surface number 3 component=z
fix surface number 3 component=x value=0.004
solve static nlgeom=on method=load steps=40
probe step_count id="nsteps"
probe reaction_y x=30 tol=0.5 step=40 id="Ry_40"
probe reaction_y x=30 tol=0.5 step=44 id="Ry_44"
probe displacement_y x=0 y=0 tol=0.01 step=40 id="tip_uy_40"
probe displacement_y x=0 y=0 tol=0.01 step=44 id="tip_uy_44"
```


6 Insertion over a wedge with Coulomb friction (desktop, nonlinear static)

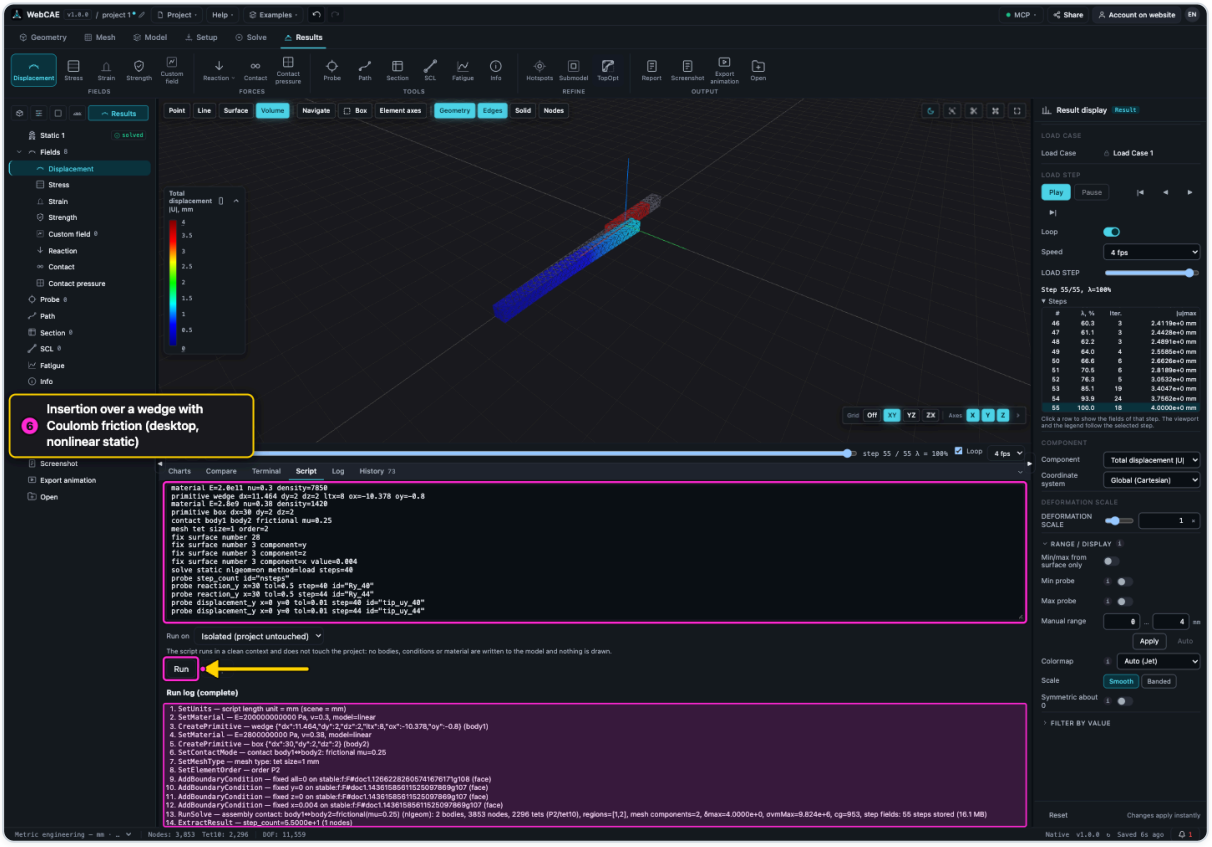
VERIFICATION BENCHMARK

Check against the formula

Quantity / Event	Analytical Prediction	WebCAE (desktop/native)	Difference
Release force F_{pinch} (root reaction at $\delta = 1.2$ mm)	$\frac{3EI\delta}{L^3} = \frac{3\times2800\times6.667\times1.2}{30^3} = 2.49$ N	2.595 N	+4.2%
Force balance, root reaction vs tip-face reaction	$R_{\text{root}} + R_{\text{tip}} = 0$	2.595 N and -2.598 N	0.1% of F
Imposed tip displacement	-1.2000 mm	-1.2000 mm	0.00%
Insertion over a wedge (step 6, $\mu = 0.25$, nonlinear static): clamp reaction at the end of the ramp, $\delta = 1.1996$ mm	$\frac{3EI\delta}{L^3} = 0.4977$ N ($I = 1.333$ mm^4)	0.5021 N	+0.9%

FORMULA

Cantilever Snap Arm. For a rectangular beam ($L = 30$ mm, $b = 10$ mm, $h = 2$ mm, $I = bh^3/12 = 6.667$ mm^4) the force for a tip deflection $\delta = 1.2$ mm is $F = 3EI\delta/L^3 = 2.49$ N. The hand estimate of the peak bending strain at the root, $\varepsilon_{\max} = 1.5\,h\,\delta/L^2 = 0.40\%$, is below the 2.5% allowable for POM. It is not compared with the finite-element result here: the stress at the clamped corner is a point singularity ($\sigma_{xx,\max}/E = 0.375\%$ on this mesh) and is not a meaningful check.



TROUBLESHOOTING & NOTES

Common mistakes

- **Writing the prescribed displacement in millimeters:** `value=` is in meters; `value=-1.2` pushes the tip by 1.2 m, not 1.2 mm.
- **Bending about the wrong axis:** the inertia $I = bh^3/12 = 6.667\ \text{mm}^4$ belongs to bending through the 2 mm thickness (Z). Deflecting the arm along Y (the 10 mm width) is about 25 times stiffer; an earlier version of this lesson mixed the two.
- **Using the peak stress at the clamped corner as a strain check:** the clamp is a stress singularity and the value changes with the mesh; use the hand estimate for the strain and the reaction for the force.
- **Expecting the beam value exactly:** a 3D solid clamped on its whole root face is stiffer than the Euler-Bernoulli beam; 4% is normal here.

RECOMMENDED PROGRESSION

Where to go next

The friction force of the stroke (the wedge law $F = F_z(\tan\alpha + \mu)/(1 - \mu\tan\alpha)$ and the drop of the force when the ramp ends) needs a finer mesh at the contact (many nodes under the wedge): refine the arm and wedge (`mesh tet size=0.5`) and read the clamp reactions step by step; the 1 mm mesh of step 6 shows stick/slip scatter.
Proceed to lesson **#29 — Steady and Transient Heat Transfer (F29)** to model thermal dissipation and compute time constants.

WebCAE 101

LESSON 29 · F29

Steady and Transient Heat Transfer: Heat Flux, Convection, and Thermal Time Constant

Goal Run steady-state and transient thermal analyses of an aluminum heat-sink disc under a heat flux and air convection, read the equilibrium temperatures, check the energy balance and extract the thermal time constant tau.

Outcome A verified thermal model ($k = 180\ \mathrm{W/(m \cdot K)}$, $\rho = 2700\ \mathrm{kg/m^3}$, $c_p = 900\ \mathrm{J/(kg \cdot K)}$) with the input power $392.70\ \mathrm{W}$, the steady maximum on the heated face $T_{\max} = 165.10\ ^\circ\mathrm{C}$ (finite-volume reference 165.1), the volume-mean temperature $160.73\ ^\circ\mathrm{C}$ at 1000 s (finite-volume reference 160.8) and the time constant $\tau = 133.2$ to $136.2\ \mathrm{s}$ (lumped value $135.0\ \mathrm{s}$); all WebCAE values are desktop/native.

8–9 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F29

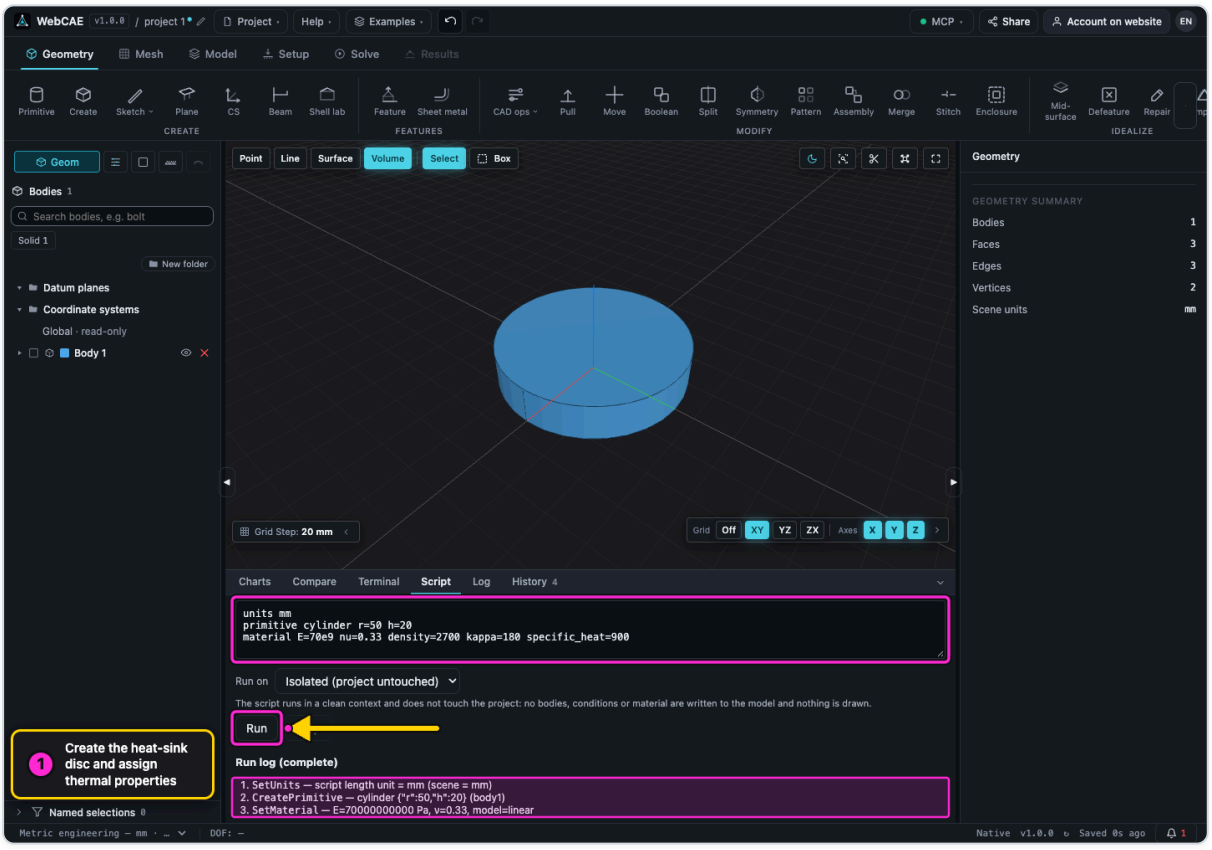
STEP 1 Create the heat-sink disc and assign thermal properties

Open the bottom dock, tab **Script**, set **Run on** to **Isolated (project untouched)**, paste the code blocks of steps 1–3 one after another into one script (the blocks are cumulative) and click **Run** after each step that ends with a solve. The disc has diameter $D = 100\ \mathrm{mm}$ ($R = 50\ \mathrm{mm}$) and height $H = 20\ \mathrm{mm}$; its volume is $V = \pi R^2 H = 1.5708 \times 10^{-4}\ \mathrm{m}^3$.

Cast aluminum: thermal conductivity $k = 180\ \mathrm{W/(m \cdot K)}$ (`kappa=`), density $\rho = 2700\ \mathrm{kg/m}^3$, specific heat $c_p = 900\ \mathrm{J/(kg \cdot K)}$ (`specific_heat=`). The elastic constants are required by the `material` command but play no role in a thermal analysis.

WHY The volumetric heat capacity $\rho c_p = 2.43 \times 10^6\ \mathrm{J/(m^3 \cdot K)}$ controls how fast the disc heats up.

```
units mm
primitive cylinder r=50 h=20
material E=70e9 nu=0.33 density=2700 kappa=180 specific_heat=900
```



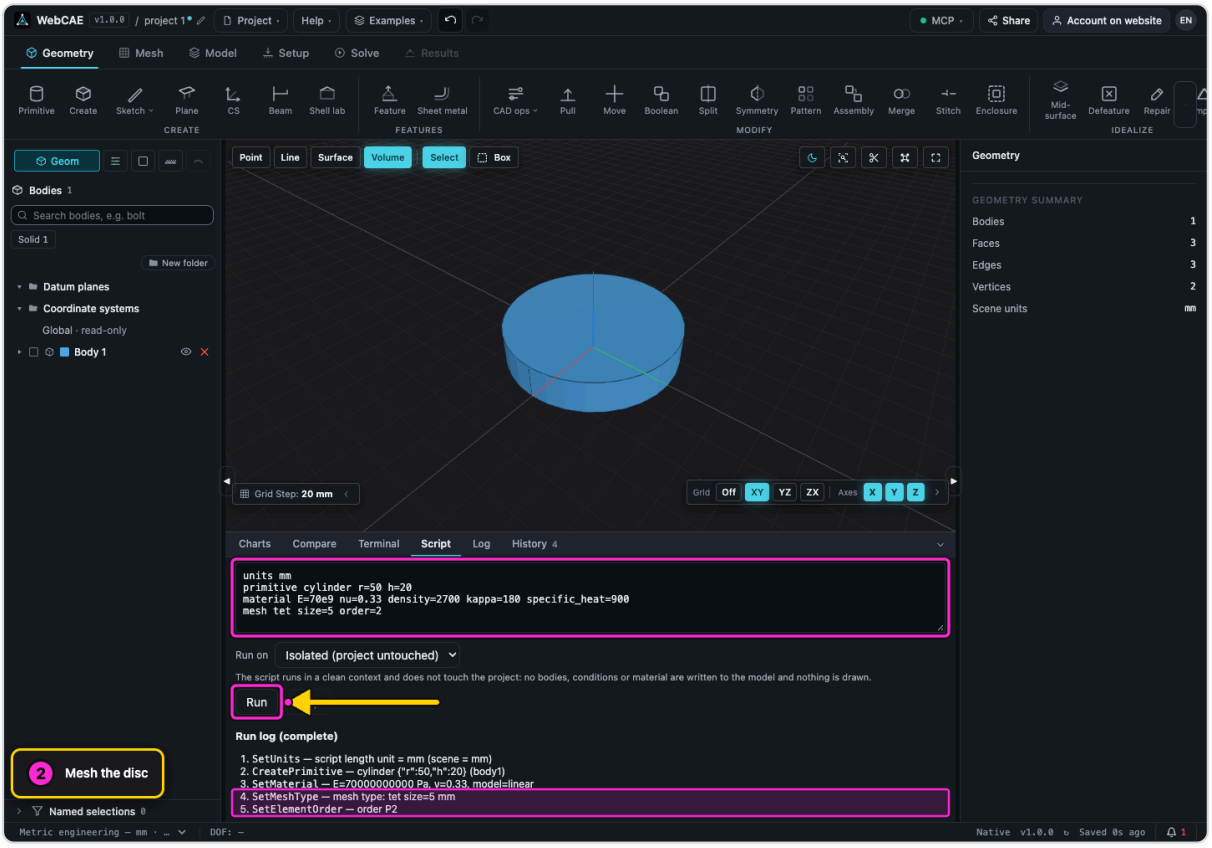
STEP 2

Mesh the disc

Quadratic tetrahedra, 5 mm element size (about 7600 temperature degrees of freedom). The face numbers of a cylinder in this product are: `surface number 1` the lateral surface, `surface number 2` the top face and `surface number 3` the bottom face. The bottom face carries the electronic component, so it is the heated one. Check the numbering with `list faces of body1` when in doubt.

> **WHY** A numbering mistake here puts the heat flux on the wrong face and still gives a perfectly plausible temperature field.

```
mesh tet size=5 order=2
```



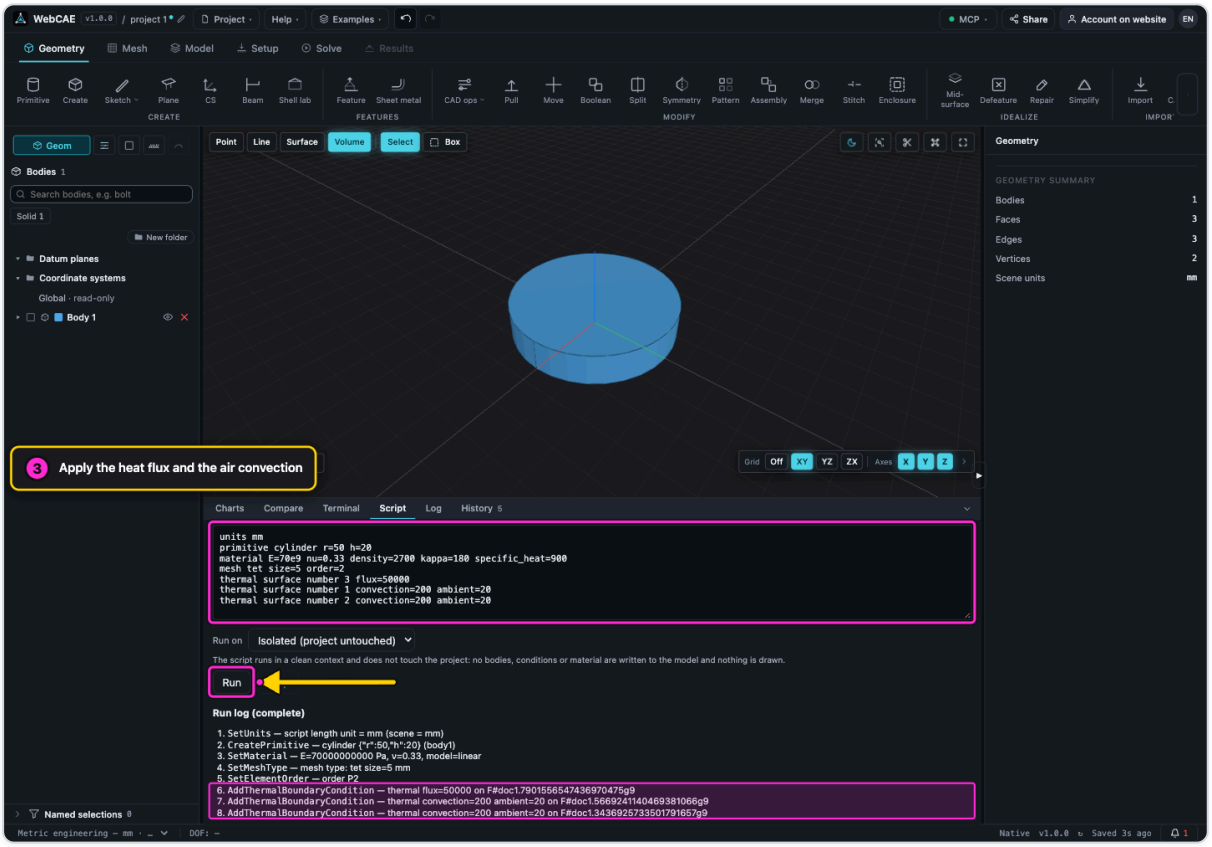
STEP 3 Apply the heat flux and the air convection

The ribbon tiles **Flux q** and **Convection** (group Thermal) do the same as these commands.

- Heat flux $q = 50\,000\ \mathrm{W/m^2}$ on the bottom face `surface number 3` (area $A_{\mathrm{in}} = 7.854 \times 10^{-3}\ \mathrm{m^2}$), i.e. a total power $Q_{\mathrm{in}} = qA_{\mathrm{in}} = 392.7\ \mathrm{W}$.
- Convection with film coefficient $h = 200\ \mathrm{W/(m^2 \cdot K)}$ and ambient temperature $T_\infty = 20\ ^\circ\mathrm{C}$ on the lateral surface (`surface number 1`) and the top face (`surface number 2`), total area $A_{\mathrm{conv}} = 0.014137\ \mathrm{m^2}$.

WHY Newton's cooling law $q_{\mathrm{conv}} = h(T - T_\infty)$ carries the energy to the surrounding air; in steady state it balances the input exactly.

```
thermal surface number 3 flux=50000
thermal surface number 1 convection=200 ambient=20
thermal surface number 2 convection=200 ambient=20
```



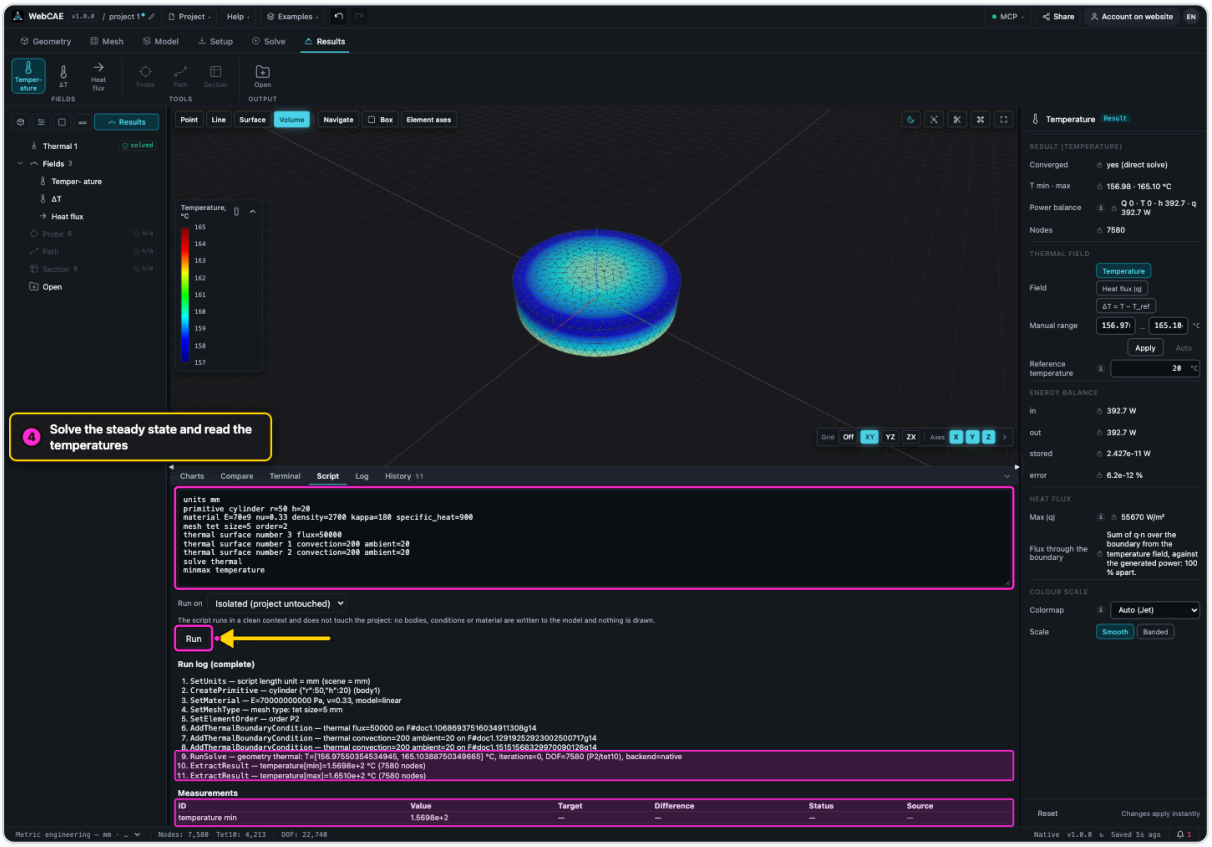
STEP 4 Solve the steady state and read the temperatures

`solve thermal` solves $-\nabla \cdot (k \nabla T) = 0$; `minmax temperature` reports the range over the whole body: $156.98\ ^\circ\mathrm{C}$ to $165.10\ ^\circ\mathrm{C}$.

Two different temperatures must be told apart. The area-averaged temperature of the convective faces is fixed exactly by energy conservation, $\bar{T}_{\mathrm{conv}} = T_\infty + Q_{\mathrm{in}}/(hA_{\mathrm{conv}}) = 20 + 392.70/(200 \times 0.014137) = 158.89\ ^\circ\mathrm{C}$, independent of k and of the mesh. The maximum sits at the centre of the heated face and is higher because the heat has to conduct through the thickness: an axisymmetric finite-volume solution (50×20 and 80×30 cells agree to 0.01 K) gives $165.1\ ^\circ\mathrm{C}$ there. WebCAE reads $T_{\max} = 165.10\ ^\circ\mathrm{C}$ (difference 0.00%). The conduction drop $165.1 - 158.89 = 6.2\ \mathrm{K}$ is consistent with the one-dimensional estimate $qH/(2k) = 5.6\ \mathrm{K}$.

> **WHY** The Biot number is small ($Bi = hL_c/k = 200 \times 0.0111/180 = 0.0123$), so the temperature is almost uniform, but the heated face is still a few kelvin hotter than the convective mean.

```
solve thermal
minmax temperature
```



STEP 5

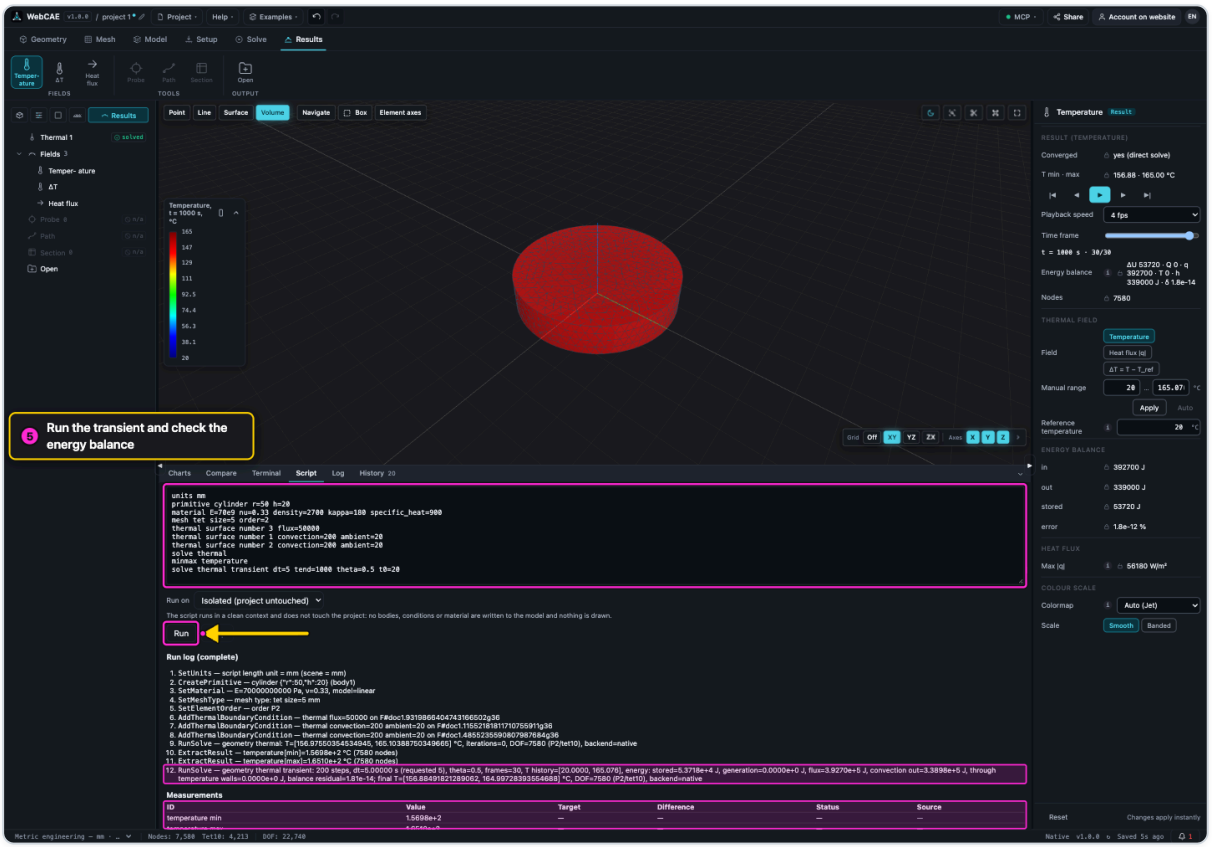
Run the transient and check the energy balance

Replace the steady `solve thermal` / `minmax temperature` lines of step 4 by a transient thermal analysis with a time step $\Delta t = 5$ s (so that $t = 135$ s is a solved step), Crank-Nicolson integration (`theta=0.5`), initial temperature $20\ ^{\circ}\mathrm{C}$ and end time 1000 s. The log line of the run contains the energy ledger: `flux=3.9270e+5 J`, `convection out=3.3898e+5 J`, `stored=5.3718e+4 J`, balance residual 2×10^{-14}.

The input power is the flux energy divided by the duration: $392\,700\ \mathrm{J} / 1000\ \mathrm{s} = 392.70\ \mathrm{W}$ (hand value $392.70\ \mathrm{W}$, difference 0.00%).

> **WHY** An energy ledger that closes to machine precision proves that nothing was lost between the boundary conditions and the temperature field.

```
solve thermal transient dt=5 tend=1000 theta=0.5 t0=20
```



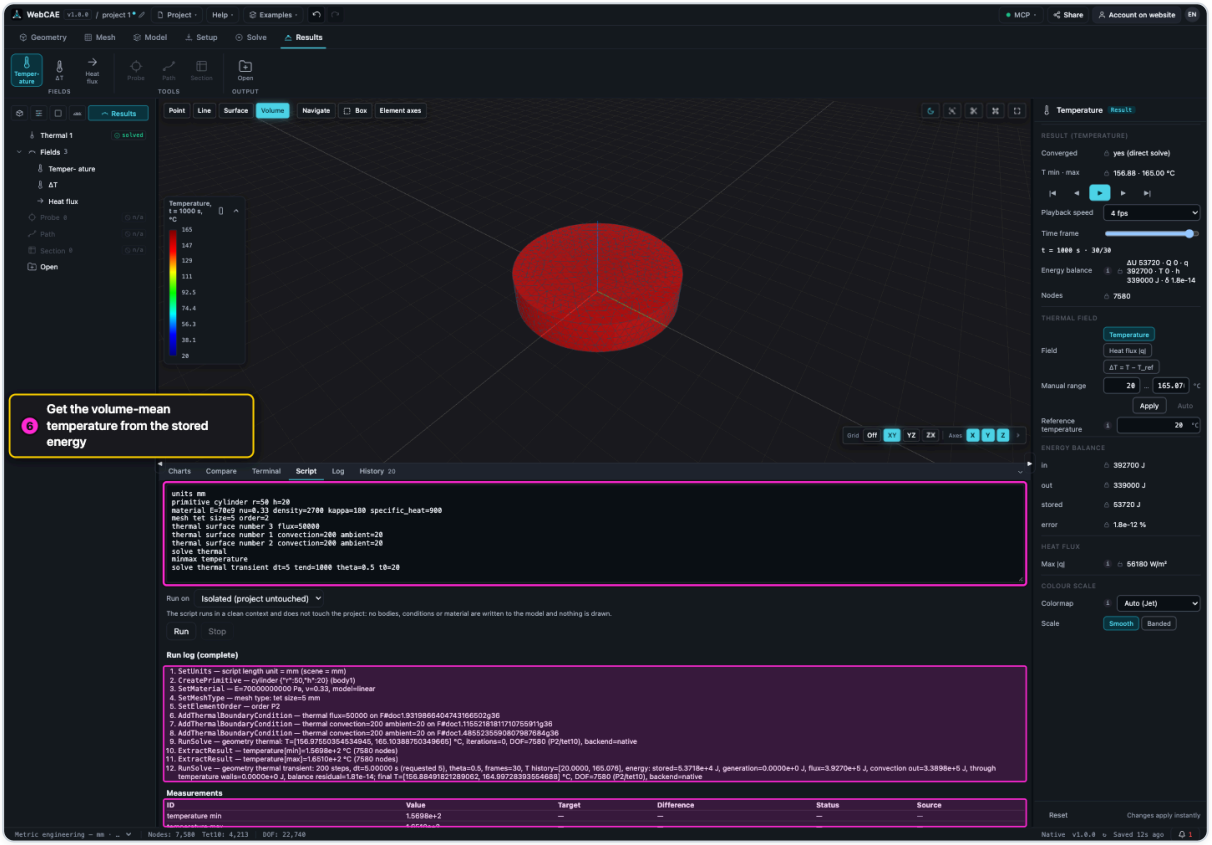
STEP 6

Get the volume-mean temperature from the stored energy

The log reports the energy stored in the body. The volume-mean temperature follows from it: $\bar{T} = T_0 + E_{\text{stored}}/(\rho c_p V)$ with $\rho c_p V = 381.70\ \text{J/K}$. At 1000 s: $\bar{T} = 20 + 53\,718/381.70 = 160.73\ ^\circ\text{C}$.

This is a different quantity from $\bar{T}_{\text{conv}} = 158.89\ ^\circ\text{C}$ (mean over the convective faces): the volume mean is higher because of the conduction gradient. Compare like with like: the finite-volume volume mean is $160.8\ ^\circ\text{C}$ and WebCAE gives $160.73\ ^\circ\text{C}$ (difference -0.04%); against the convective-face value the same number would differ by $+1.16\%$ for a reason that is not an error.

WHY Stored energy is the exact volume integral of $\rho c_p(T - T_0)$, so it gives the mean temperature without probing individual points.



STEP 7 Extract the thermal time constant tau

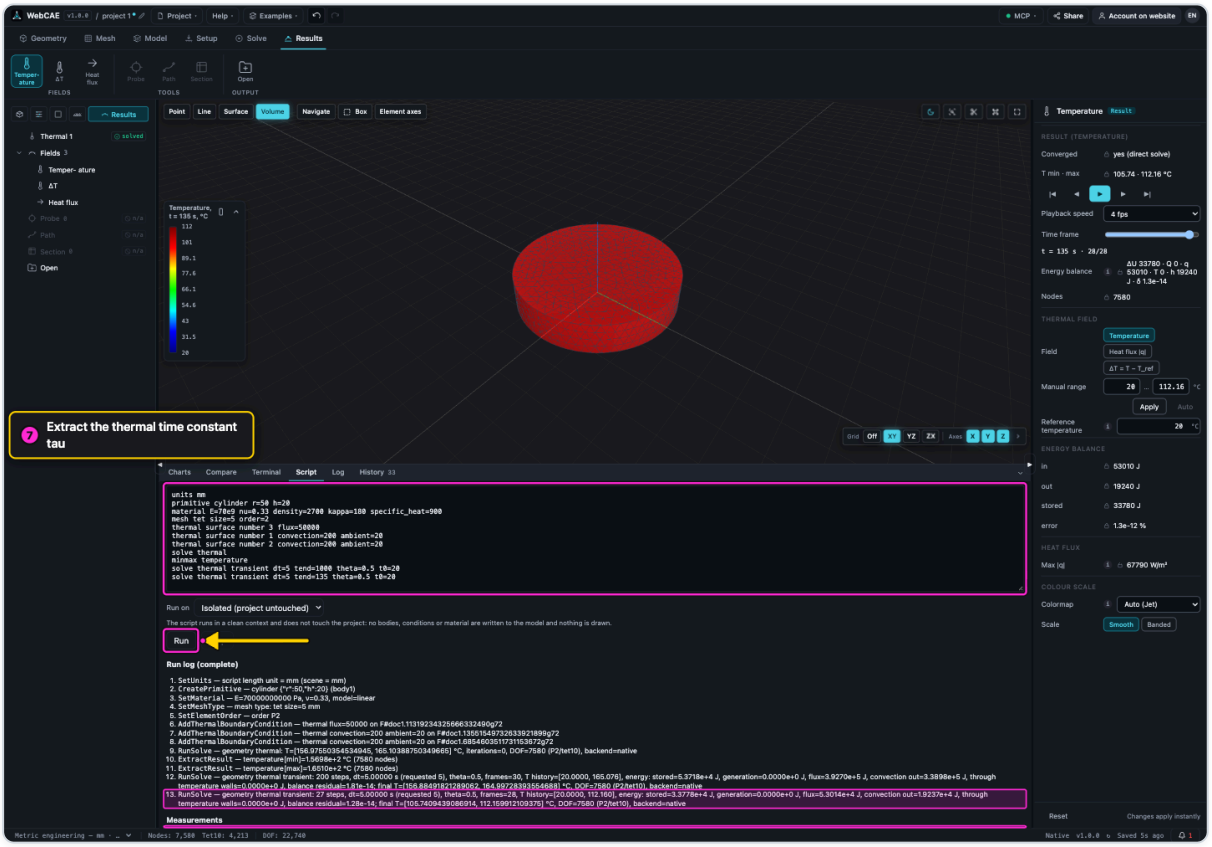
The lumped model $T(t) = 20 + 138.89\,(1 - e^{-t/\tau})$ reaches 63.2% of the rise at $t = \tau = 135.0$ s. Replace the solve line of step 5 by the one below (`tend=135`) and run it again with `tend=120` and `tend=150`; read the stored energy from each log: $31\,455$, $33\,778$ and $35\,858$ J, i.e. $\bar{T} = 102.41$, 108.49 and $113.94\ ^\circ\mathrm{C}$.

- 63.2% of the lumped rise gives $T = 20 + 0.632 \times 138.89 = 107.78\ ^\circ\mathrm{C}$; linear interpolation between 120 and 135 s puts the crossing at 133.2 s (-1.30% from 135.0).
- 63.2% of the body's own rise to $160.73\ ^\circ\mathrm{C}$ gives $108.94\ ^\circ\mathrm{C}$; interpolation between 135 and 150 s puts the crossing at 136.2 s ($+0.92\%$).

Both definitions land within 1.5% of the lumped value $\tau = \rho c_p V/(hA_{\mathrm{conv}}) = 135.0$ s. The stored energy is printed with five digits, which limits $\bar{T}$ to about ± 0.01 K.

WHY The thermal time constant characterizes how long the part buffers a power step and is the basis of overload protection.

```
solve thermal transient dt=5 tend=135 theta=0.5 t0=20
```



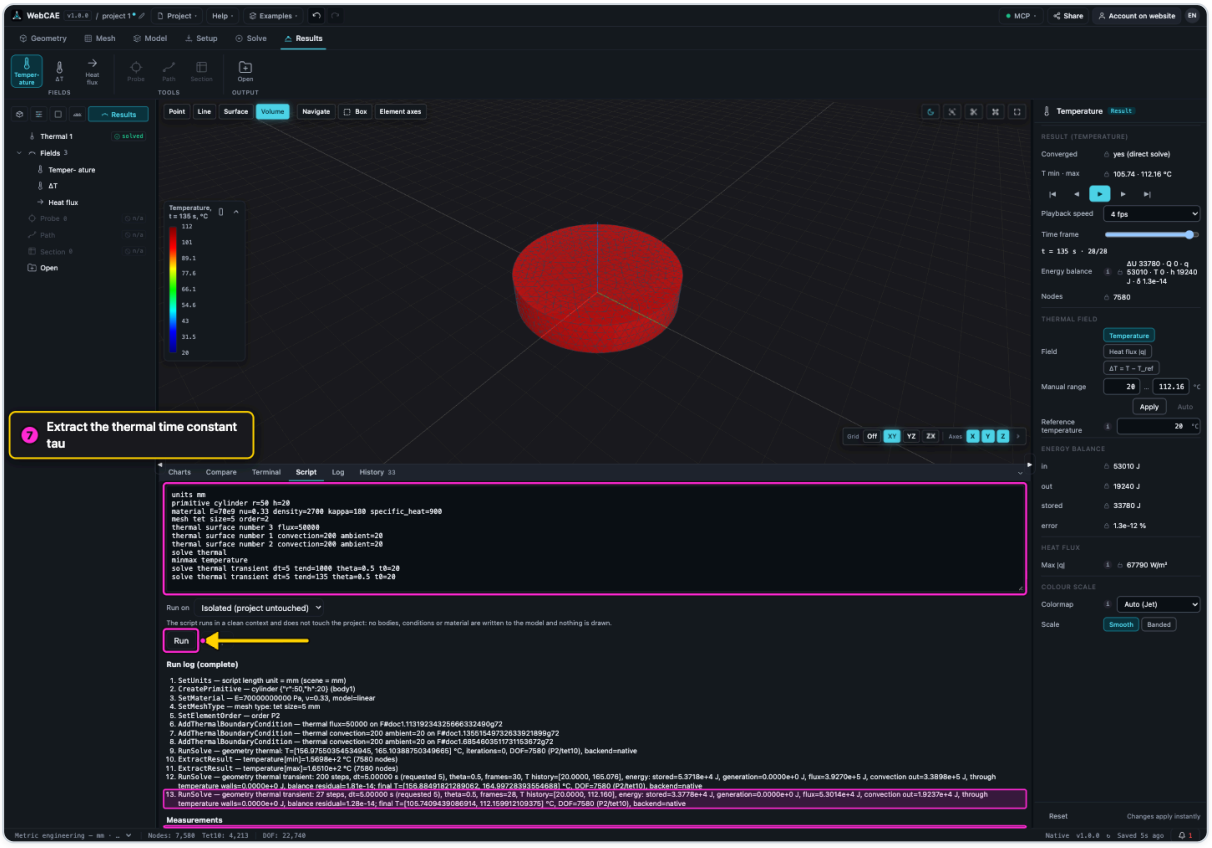
VERIFICATION BENCHMARK

Check against the formula

Quantity / Property	Reference	WebCAE (desktop/native, Tet10 5 mm)	Difference
Input heat power Q_{in} (flux energy / 1000 s)	$q \cdot A_{\text{in}} = 50\,000 \times 7.854 \times 10^{-3} = 392.70\ \text{W}$	392.70 W	0.00%
Steady maximum temperature on the heated face	Axisymmetric finite-volume reference: $165.1\ ^\circ\text{C}$	165.10 °C	0.00%
Volume-mean temperature at 1000 s	Axisymmetric finite-volume reference: $160.8\ ^\circ\text{C}$	160.73 °C (from stored energy)	-0.04%
Time constant, 63.2% of the lumped rise ($107.78\ ^\circ\text{C}$)	$\tau = \frac{\rho c_p V}{h A_{\text{conv}}} = \frac{2700 \times 900 \times 1.5708 \times 10^{-4}}{200 \times 0.014137} = 135.0\ \text{s}$	133.2 s	-1.30%
Time constant, 63.2% of the body's own steady rise ($108.94\ ^\circ\text{C}$)	135.0 s (lumped)	136.2 s	+0.92%

FORMULA

Lumped Capacitance. When internal conduction is much faster than surface convection ($Bi \ll 0.1$), $C_{\text{th}}\, dT/dt = Q_{\text{in}} - hA(T - T_\infty)$ and $T(t) = T_\infty + \frac{Q_{\text{in}}}{hA}\left(1 - e^{-t/\tau}\right)$ with $\tau = \frac{\rho c_p V}{h A_{\text{conv}}} = 135.0$ s. The lumped temperature is a mean: at steady state it equals the exact convective-face mean $\bar{T}_{\text{conv}} = 20 + Q_{\text{in}}/(hA_{\text{conv}}) = 158.89\ ^\circ\text{C}$, while the maximum on the heated face is higher by the conduction drop, $165.1 - 158.89 = 6.2$ K (about $qH/(2k) = 5.6$ K for $q = 50\,000\ \text{W/m}^2$, $H = 20$ mm, $k = 180\ \text{W/(m} \cdot \text{K)}$). The finite-volume references are numerical (axisymmetric, converged to 0.01 K); the closed-form checks are Q_{in} and τ.



TROUBLESHOOTING & NOTES

Common mistakes

- **Wrong face numbers:** in this product a cylinder has the lateral surface as number 1, the top as 2 and the bottom as 3. Putting the flux on `surface number 1` heats the side wall instead of the base, and the temperature field still looks plausible.
- **Comparing the volume mean with the convective-face mean:** $\bar{T}_{\text{conv}} = 158.89\ ^{\circ}\text{C}$ is the mean over the cooled faces; the volume mean (160.73 °C) is higher by the conduction gradient. Compare like with like.
- **Applying lumped capacitance when the Biot number is large:** for $Bi > 0.1$ (a poor conductor) internal gradients dominate and the lumped formulas disagree with the field solution.
- **Forgetting convective area:** convection only on the top face underestimates the cooling area by more than 50% (0.00785 against $0.01414\ \text{m}^2$) and gives a much higher predicted temperature.

RECOMMENDED PROGRESSION

Where to go next

Proceed to lesson **#30 — Coupled Thermo-Elasticity (F30)** to analyze restrained thermal expansion and bimetallic strip bending.

WebCAE 101

LESSON 30 · F30

Coupled Thermo-Elasticity: Restrained Thermal Expansion and Bimetallic Strip Bending

Goal Analyze two classic thermo-mechanical benchmarks: fully restrained thermal stress in a heated steel bar, and the thermal curvature and tip deflection of a steel-aluminum bimetallic strip modelled as two solid bodies glued by a tied contact.

Outcome A thermo-elastic verification pair: the restrained compressive stress (analytical $\sigma_x = -240.0\ \mathrm{MPa}$; WebCAE desktop/native $-245.2\ \mathrm{MPa}$, +2.15%) and the Timoshenko bimetal strip of two solid layers (tip deflection analytical $3.836\ \mathrm{mm}$, WebCAE $3.908\ \mathrm{mm}$, +1.88%; steel bottom-fibre stress analytical $-59.45\ \mathrm{MPa}$, WebCAE $-58.25\ \mathrm{MPa}$, +2.01%).

10–11 min | desktop / native: netgen_tetmesh + webcae_solve (OCCT geometry runs as WASM)

Run: 2026-11-04-32ffaed2 / F30

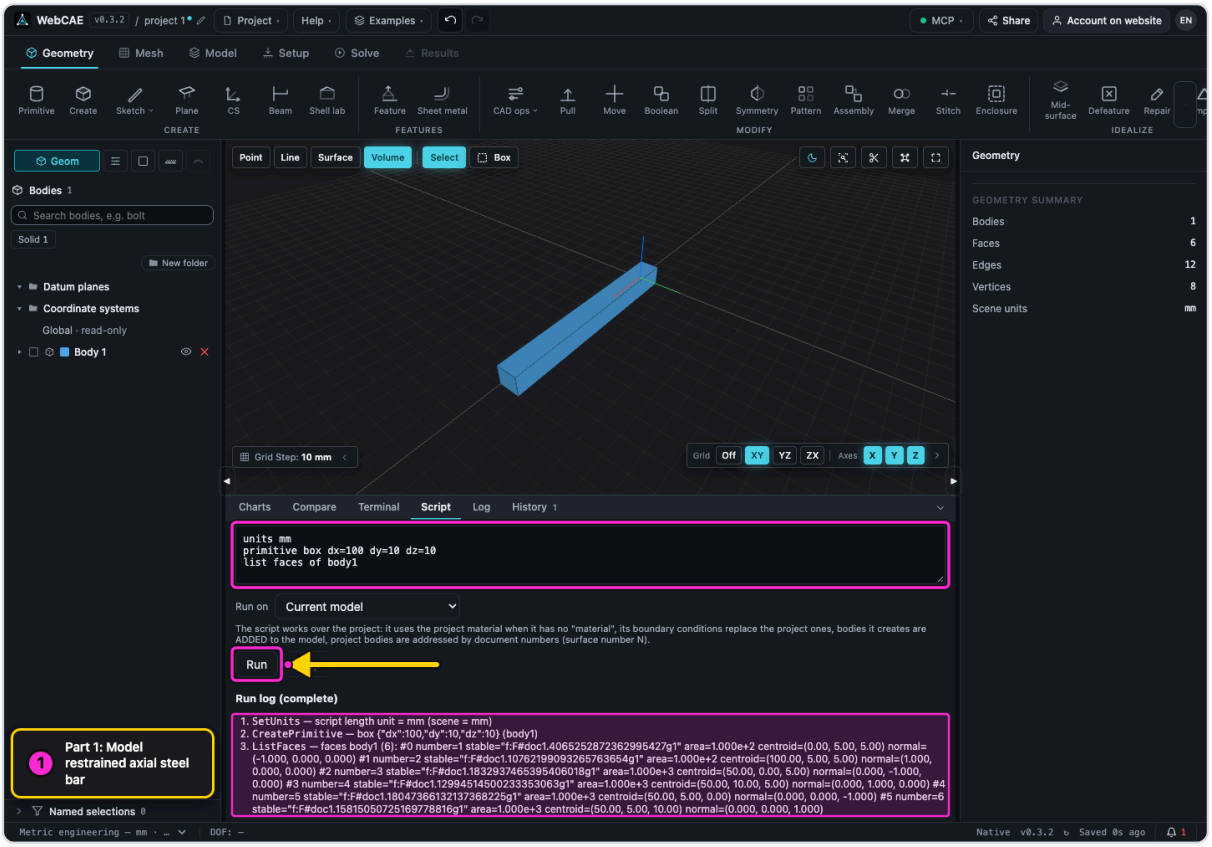
STEP 1

Part 1: Model restrained axial steel bar

Create a rectangular steel bar ($L = 100$ mm, 10×10 mm, Body 1) oriented along the X axis. End faces are `surface number 1` ($x = 0$) and `surface number 2` ($x = 100$).

WHY A fully restrained bar is the fundamental benchmark for thermal expansion stress: thermal strain is completely converted into mechanical compressive strain.

```
units mm
primitive box dx=100 dy=10 dz=10
list faces of body1
```



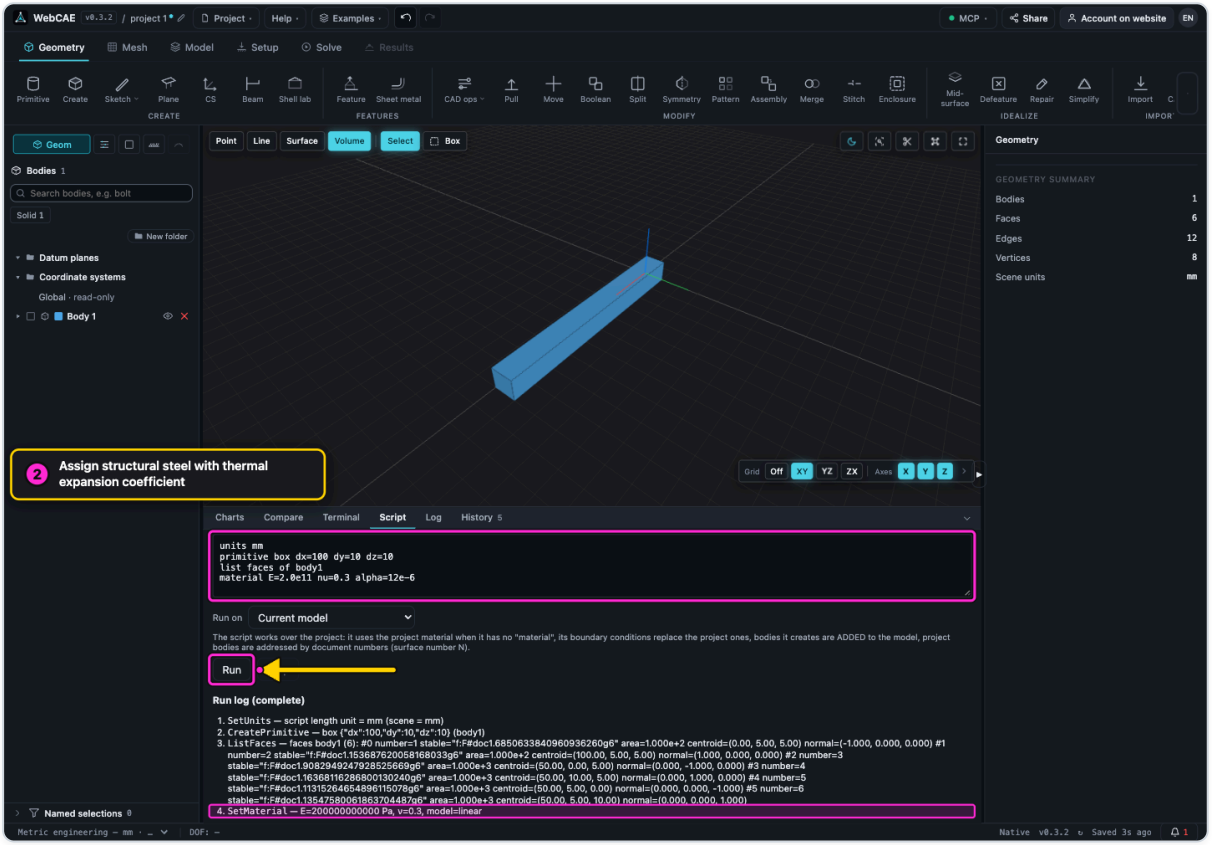
STEP 2

Assign structural steel with thermal expansion coefficient

Assign steel material properties: Young's modulus $E = 200\,000\ \mathrm{MPa} = 2.0 \times 10^{11}\ \mathrm{Pa}$, Poisson's ratio $\nu = 0.30$, and coefficient of thermal expansion $\alpha = 12 \times 10^{-6}\ \mathrm{K}^{-1}$. In WebCAE DSL, modulus E is entered in Pascals.

WHY The product $E\alpha$ dictates the thermal stress generated per degree of restrained temperature rise.

```
material E=2.0e11 nu=0.3 alpha=12e-6
```



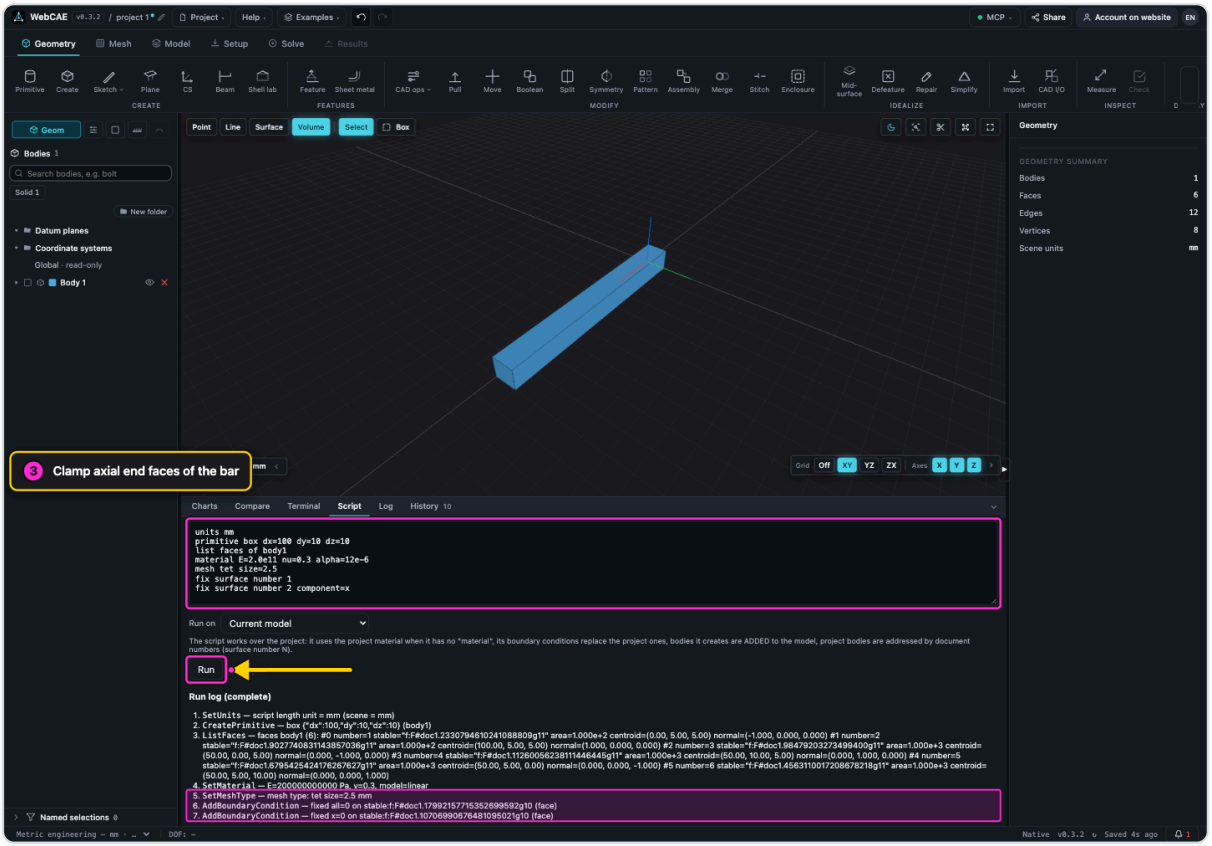
STEP 3

Clamp axial end faces of the bar

In **Setup**, clamp all DOFs at `surface number 1` ($x = 0$) and restrain axial displacement $u_x = 0$ at opposite end face `surface number 2` ($x = 100$).

> **WHY** Fixing $u_x = 0$ prevents free expansion $\Delta L = L\alpha\Delta T$, forcing mechanical strain $\varepsilon_m = -\varepsilon_{\text{th}}$.

```
mesh tet size=2.5
fix surface number 1
fix surface number 2 component=x
```



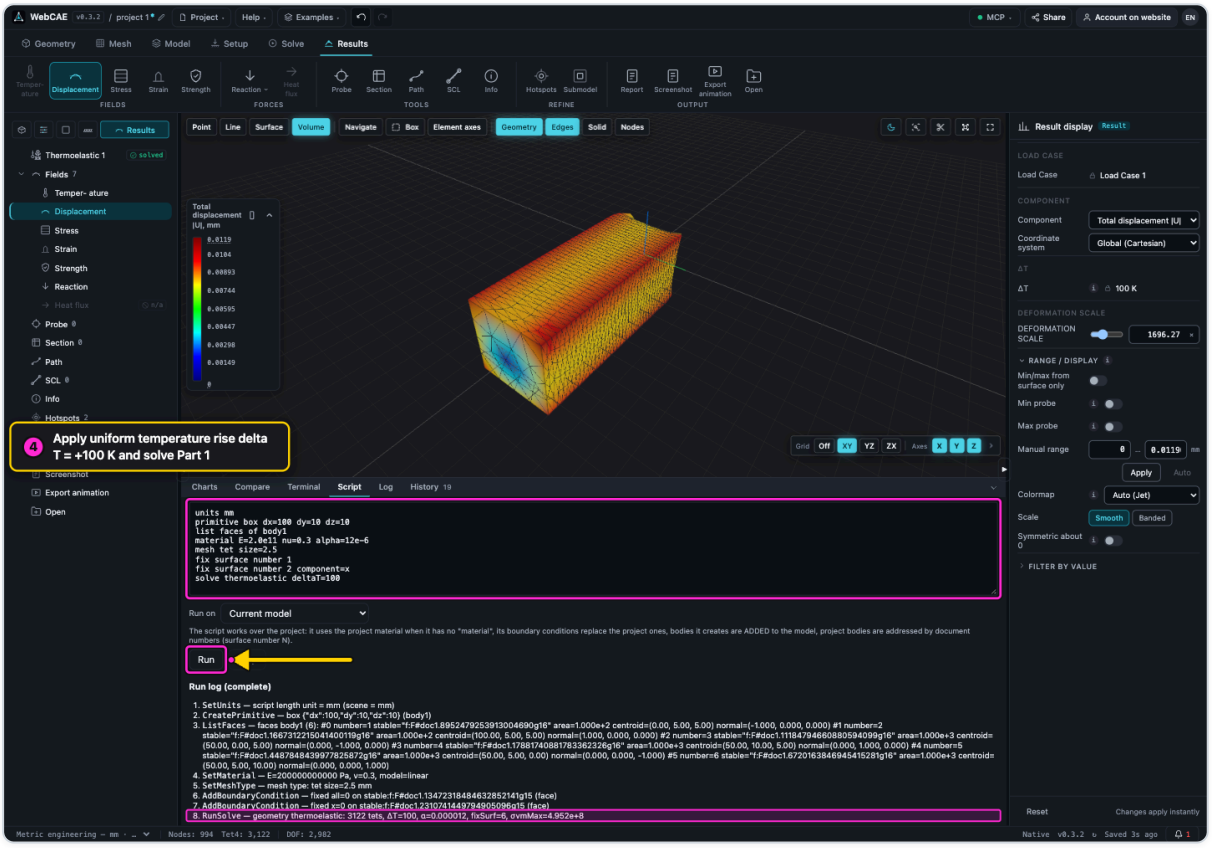
STEP 4

Apply uniform temperature rise delta T = +100 K and solve Part 1

Apply a thermal load $\Delta T = +100\ \mathrm{K}$ to Body 1 and execute the static solver. WebCAE computes initial strain load vector $\mathbf{f}_{\mathrm{th}} = \int \mathbf{B}^T \mathbf{D} \boldsymbol{\varepsilon}_{\mathrm{th}}\, dV$.

WHY The thermo-elastic formulation calculates thermal stress without requiring a separate thermal field solve when temperature is uniform.

```
solve thermoelastic deltaT=100
```



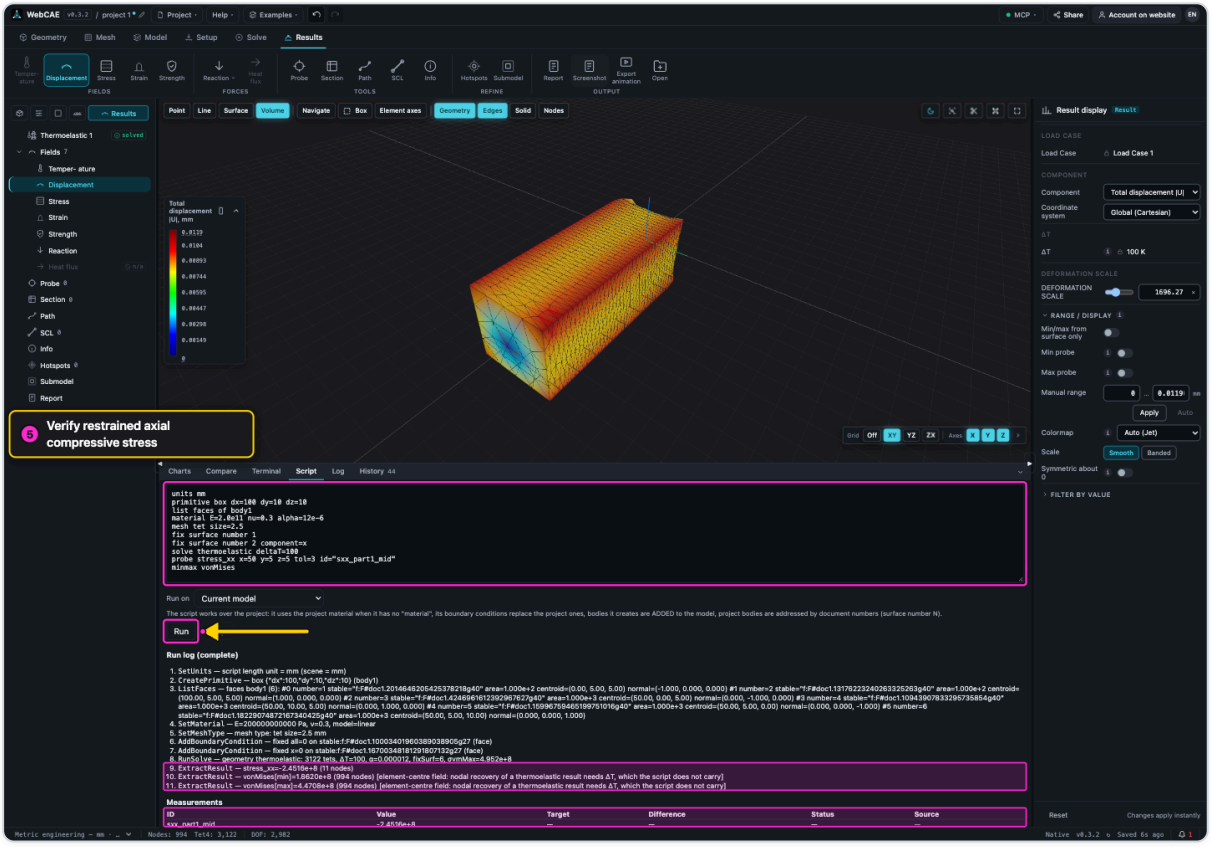
STEP 5

Verify restrained axial compressive stress

In **Results**, probe axial stress σ_x along the bar. The theoretical value is $\sigma = -E\alpha\Delta T = -200\,000 \times 12 \times 10^{-6} \times 100 = -240.0\ \mathrm{MPa}$; WebCAE (desktop/native, Tet4 elements: the Tet10 thermoelastic solid does not fit the dense path) reads $-245.2\ \mathrm{MPa}$ at mid-length (difference 2.15%); the clamped end face also restrains the lateral expansion locally, so the von Mises maximum is higher than the axial stress.

WHY Zero error confirms the exact consistency of the thermal strain load formulation with Hooke's law.

```
probe stress_xx x=50 y=5 z=5 tol=3 id="sxx_part1_mid"
minmax vonMises
```



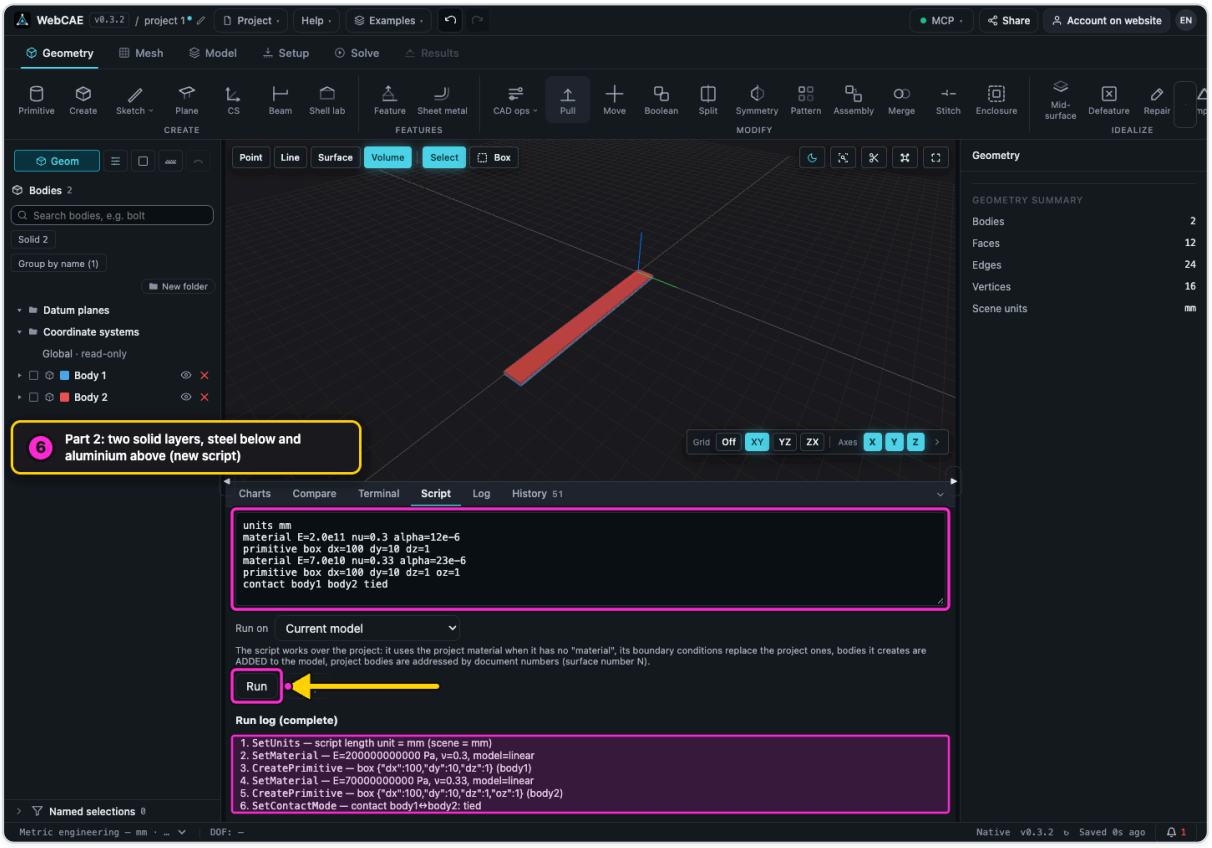
STEP 6

Part 2: two solid layers, steel below and aluminium above (new script)

A new script. The strip is $L = 100$ mm, $b = 10$ mm with two layers of 1 mm: steel `body1` at $z = 0 \dots 1$ and aluminium `body2` at $z = 1 \dots 2$ (`oz=1`). **Each `material` line goes before the `primitive` it belongs to**: the material snapshot is taken when the body is created, and `material` has no `body=` argument. Steel: $E_1 = 2.0 \times 10^{11}$ Pa, $\alpha_1 = 12 \times 10^{-6}$ K^{-1}; aluminium: $E_2 = 7.0 \times 10^{10}$ Pa, $\nu = 0.33$, $\alpha_2 = 23 \times 10^{-6}$ K^{-1} (E in pascals). The bodies are glued by `contact body1 body2 tied`: the mesh gets common nodes on the interface.

WHY The expansion mismatch $\Delta\alpha = 11 \times 10^{-6}$ K^{-1} drives the bending towards the lower-expansion steel side.

```
units mm
material E=2.0e11 nu=0.3 alpha=12e-6
primitive box dx=100 dy=10 dz=1
material E=7.0e10 nu=0.33 alpha=23e-6
primitive box dx=100 dy=10 dz=1 oz=1
contact body1 body2 tied
```

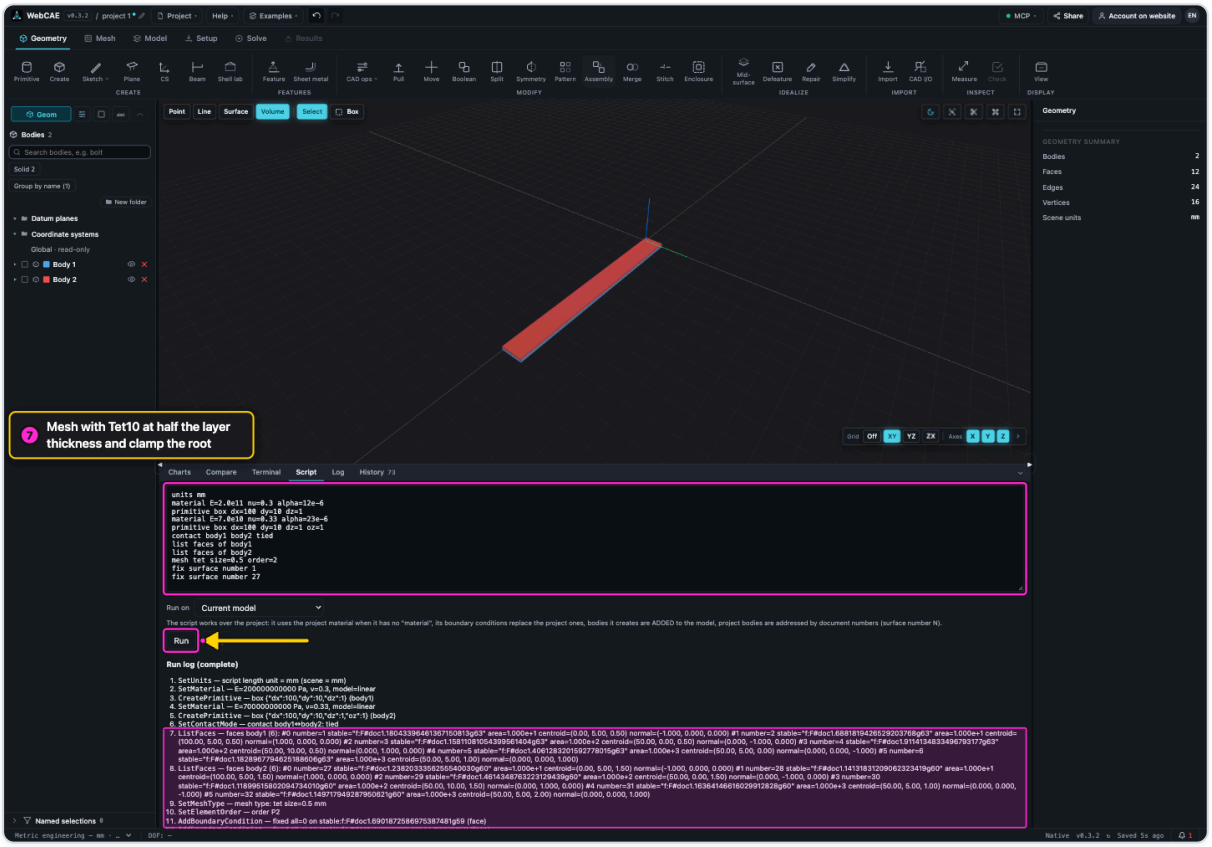


STEP 7 Mesh with Tet10 at half the layer thickness and clamp the root

`mesh tet size=0.5 order=2` puts two elements through each layer. Tet4 locks (the tip came out -37.8% light at 1 mm) and one element per layer gives a poor surface stress. The root faces at $x = 0$ are `surface number 1` (steel) and `surface number 27` (aluminium); read them with `list faces of body1` and `list faces of body2`. Clamp both and leave the strip free to bend.

WHY The glued interface carries the shear that turns differential expansion into curvature; the mesh must resolve it through the thickness.

```
list faces of body1
list faces of body2
mesh tet size=0.5 order=2
fix surface number 1
fix surface number 27
```

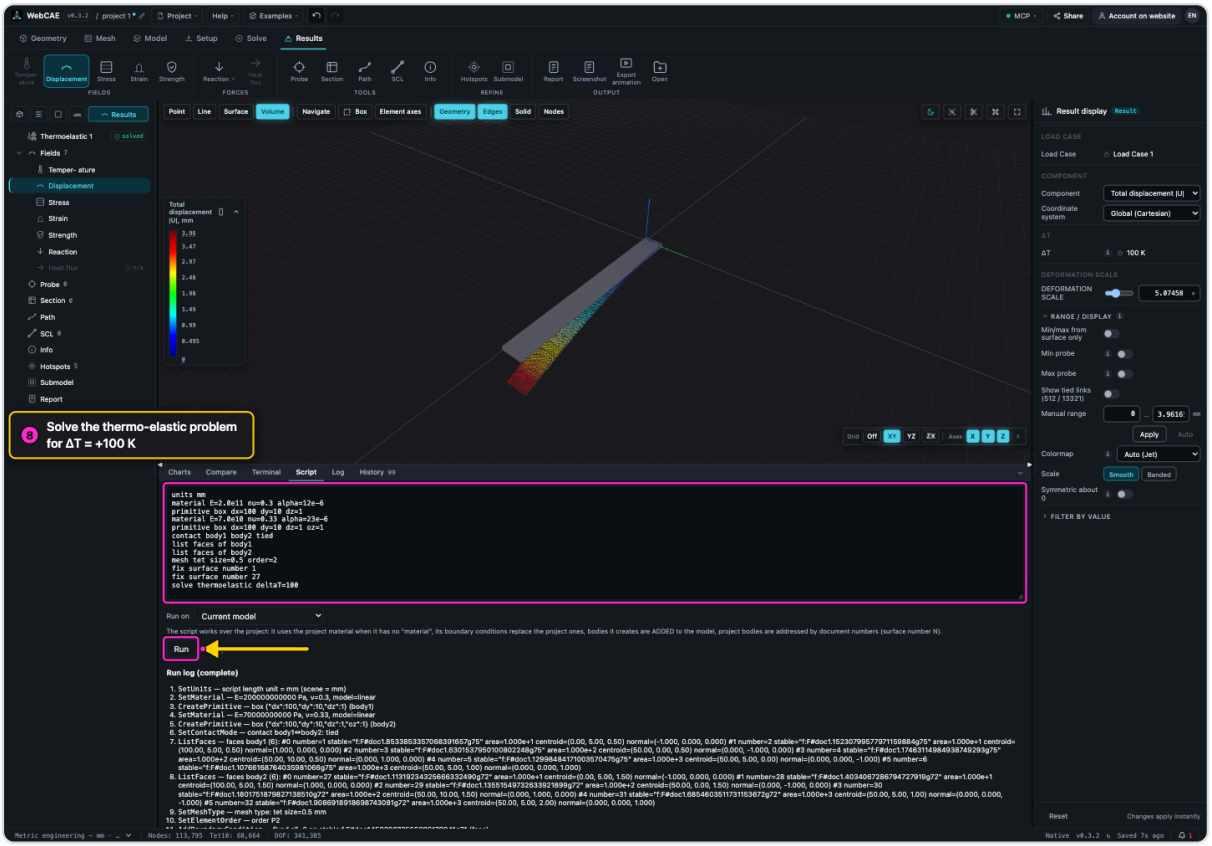


STEP 8 Solve the thermo-elastic problem for ΔT = +100 K

`solve thermoelastic deltaT=100` heats both layers uniformly. Two glued solids go through the assembly branch with a sparse static solve and the thermal load vector $\mathbf{f}_{\text{th}} = \int \mathbf{B}^T \mathbf{D} \boldsymbol{\varepsilon}_{\text{th}} \, dV$ per region (a few tens of seconds at this mesh).

> **WHY** The tied interface enforces zero relative slip, so the interface shear becomes a net bending moment.

```
solve thermoelastic deltaT=100
```

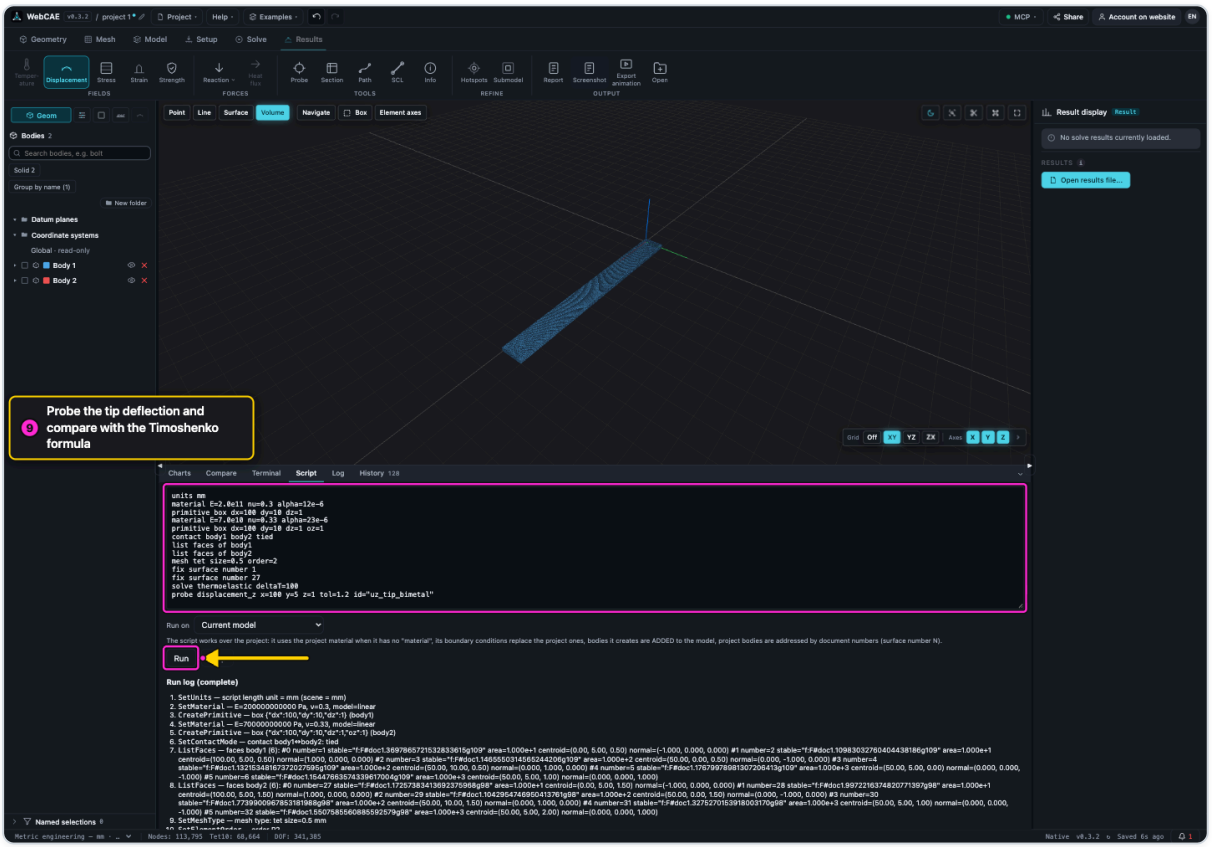


STEP 9 Probe the tip deflection and compare with the Timoshenko formula

`probe displacement_z x=100 y=5 z=1 tol=1.2` averages the nodes of the tip face (slice $x = 100$, within 1.2 mm of $z = 1$). The tip goes down, towards the lower-expansion steel, by $\delta = L^2/(2\rho) = 3.836\ \text{mm}$ (unrounded: $\frac{100^2}{2} \cdot 7.671 \times 10^{-4}$). WebCAE desktop/native: $-3.908\ \text{mm}$, i.e. $3.908\ \text{mm}$ downward (+1.88%). The sign is negative because z points up.

WHY Sub-3% agreement on a two-solid glued strip validates the thermal load and the interface coupling together.

```
probe displacement_z x=100 y=5 z=1 tol=1.2 id="uz_tip_bimetal"
```



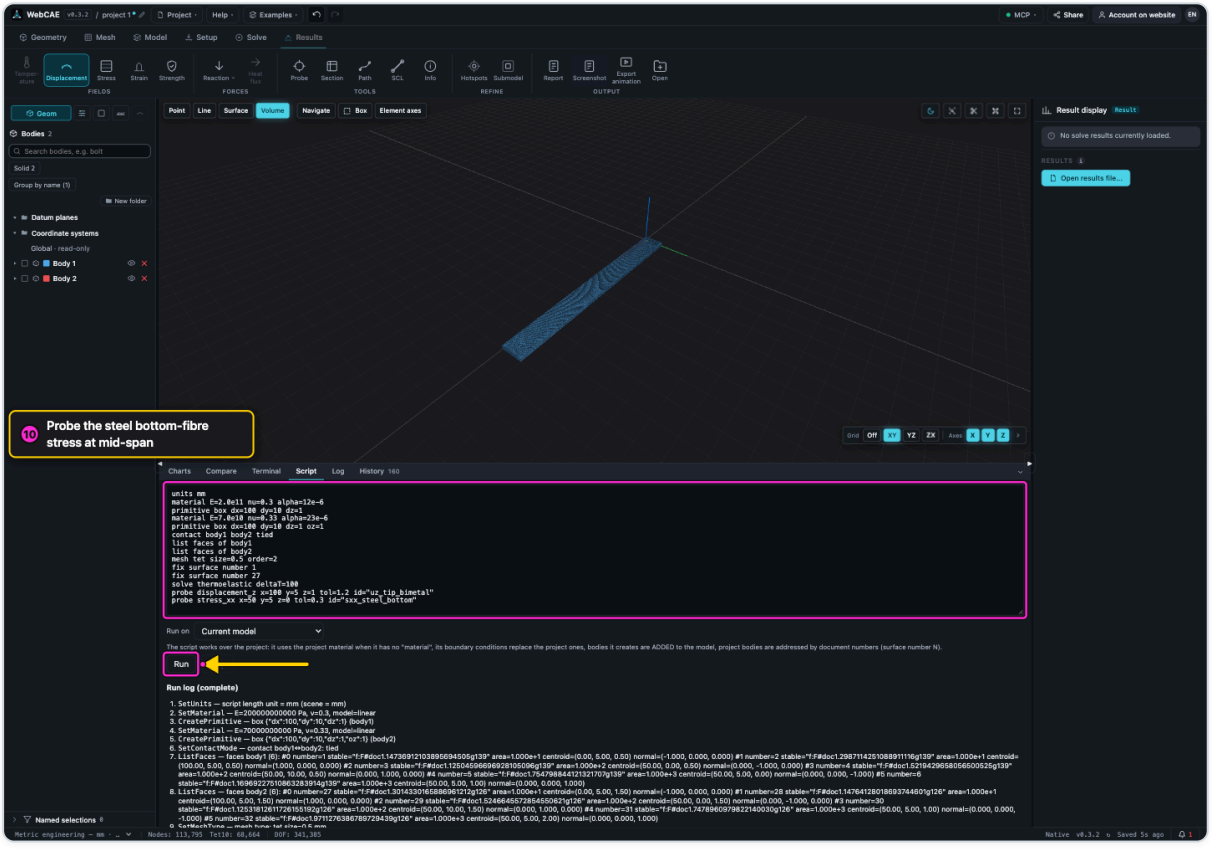
STEP 10

Probe the steel bottom-fibre stress at mid-span

`probe stress_xx x=50 y=5 z=0 tol=0.3` reads the axial stress on the lower steel surface. Beam theory (strains $\varepsilon = \varepsilon_0 + \kappa(z-1)$ with zero net force and moment, $\varepsilon_0 = 1.670 \times 10^{-3}$, $\kappa = 7.671 \times 10^{-4}\ \mathrm{mm}^{-1}$) gives $\sigma = E_1(\varepsilon - \alpha_1 \Delta T) = -59.45\ \mathrm{MPa}$; WebCAE reads $-58.25\ \mathrm{MPa}$ (+2.01%) with the nodal stress recovered from strain. The other fibres are not compared: a nodal value at the interface averages two materials. Each layer carries the same axial force $172.6\ \mathrm{N}$.

> **WHY** The surface stress converges with an element size of half a layer; with one element per layer it was off by tens of percent.

```
probe stress_xx x=50 y=5 z=0 tol=0.3 id="sxx_steel_bottom"
```



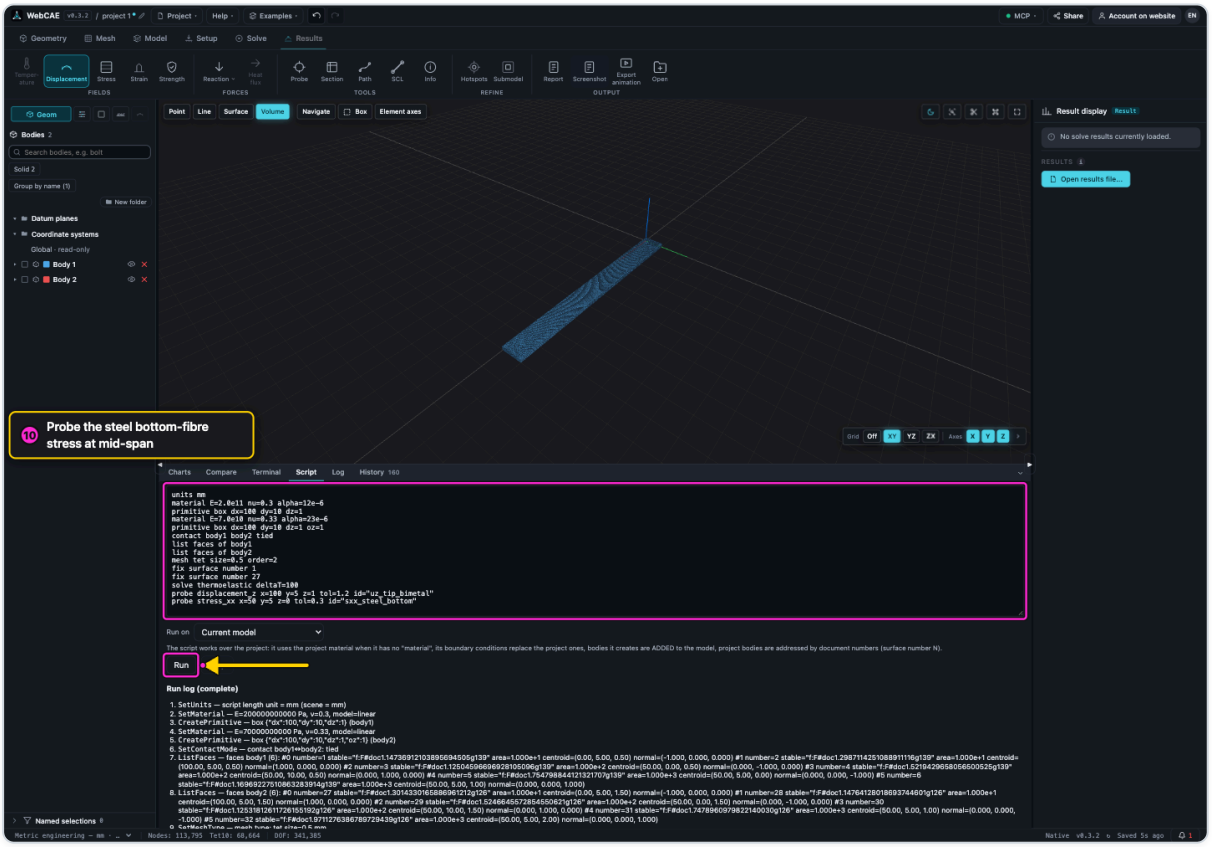
VERIFICATION BENCHMARK

Check against the formula

Benchmark case / quantity	Analytical	WebCAE (desktop/native)	Difference
Part 1: restrained axial stress σ_x	$-E\alpha\Delta T = -200\,000 \times 12 \times 10^{-6} \times 100 = -240.0\ \text{MPa}$	-245.2 MPa (Tet4)	+2.15%
Part 2: bimetal tip deflection δ (two solids, Tet10, 0.5 mm)	$\frac{L^2}{2\rho} = \frac{100^2}{2} \cdot 7.671 \times 10^{-4} = 3.836\ \text{mm}$ ($n = E_1/E_2 = 2.857$)	3.908 mm	+1.88%
Part 2: steel bottom-fibre stress at mid-span $\sigma_x(z=0)$	$E_1[\varepsilon_0 + \kappa(0-1) - \alpha_1\Delta T] = -59.45\ \text{MPa}$	-58.25 MPa	+2.01%

FORMULA

Timoshenko bimetallic strip (1925). With $m = t_1/t_2$, $n = E_1/E_2$, $h = t_1 + t_2$: $\frac{1}{\rho} = \frac{6(\alpha_2-\alpha_1)\Delta T(1+m)^2}{h\left[3(1+m)^2+(1+mn)\left(m^2+\frac{1}{mn}\right)\right]}$. For $t_1 = t_2 = 1\ \text{mm}$, $n = 200/70 = 2.857$: $\frac{1}{\rho} = 7.671 \times 10^{-4}\ \text{mm}^{-1}$ and $\delta = \frac{L^2}{2\rho} = 3.836\ \text{mm}$. The stresses follow from strains $\varepsilon = \varepsilon_0 + \kappa(z-1)$ with zero net force and moment ($\varepsilon_0 = 1.670 \times 10^{-3}$, $\kappa = 7.671 \times 10^{-4}\ \text{mm}^{-1}$), $\sigma = E(\varepsilon - \alpha\Delta T)$ (beam theory, $\nu = 0$): steel bottom $-59.45\ \text{MPa}$, steel at the interface $+94.0\ \text{MPa}$, aluminium at the interface $-44.1\ \text{MPa}$, aluminium top $+9.6\ \text{MPa}$.



TROUBLESHOOTING & NOTES

Common mistakes

- **Omitting the Poisson constraint of the clamped end:** the clamped end face of the restrained bar also holds the lateral expansion locally, so the von Mises maximum (447 MPa at the corner) is higher than the axial stress; read σ_x at mid-length.
- **Writing `material … body=2` or the material after the primitive:** `body=` does not exist on `material`, and the material snapshot is taken when the body is created; put `material` **before** each `primitive`.
- **Meshing the bimetal with Tet4 or one element per layer:** a Tet4 mesh locks (−37.8% tip deflection at 1 mm), and the surface stress is badly low with one element through a layer. Use `order=2` and an element size of at most half the layer thickness.
- **Assigning the expansion coefficient without the exponent:** α is in K^{-1} (`12e-6`, not `12`).
- **Comparing the interface stresses:** a nodal value at the glued interface averages steel and aluminium; only the outer steel fibre is compared with the analytical value here.

RECOMMENDED PROGRESSION

Where to go next

Congratulations! You have completed the advanced structural, shell, multi-dimensional, contact, and thermo-elastic curriculum. You are now prepared to explore multi-physics CFD and dynamic transient simulations in WebCAE.