

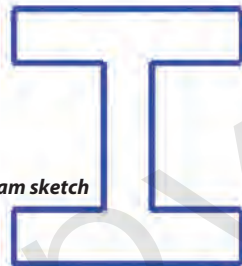
Open and Closed Sketches

In Pro/ENGINEER sketch mode, users can create open or closed sketches. Closed sketches are sketches with a definite inside and outside.

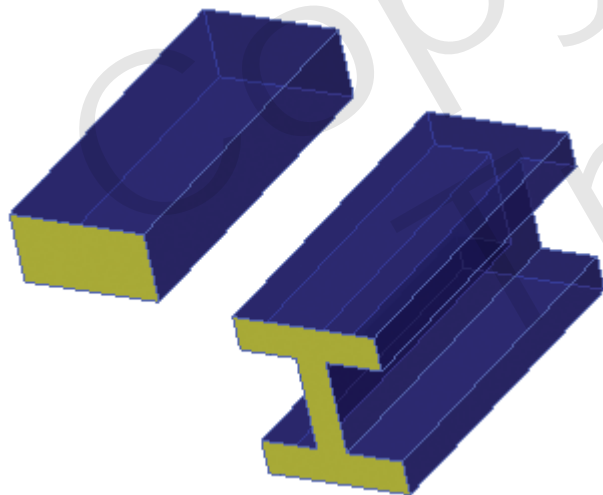
Below are both a rectangle and an I-beam that are closed sketches. Closed sketches are preferred because they behave the way we would expect, and they create a feature that is not prone to failure and does not put you in resolve mode if it is moved off the solid part.



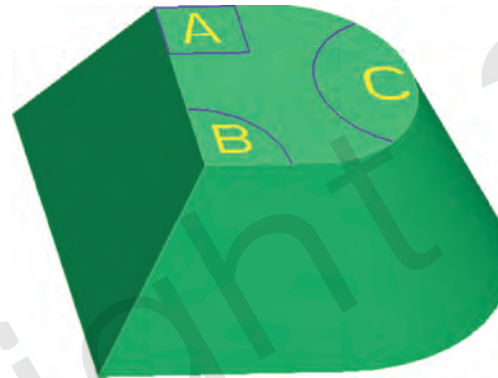
Closed rectangular sketch



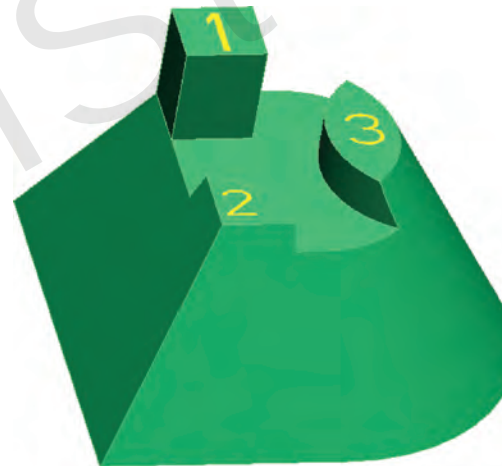
Closed I-beam sketch



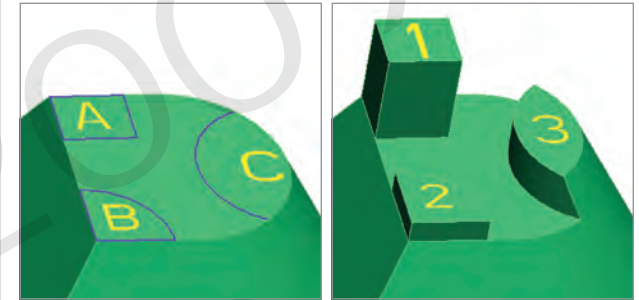
Open sketches, on the other hand, are a bit rebellious. They behave in ways that you may or may not expect. For example, look at the part below. It has one closed section (A) and two open sections (B and C). What do you think will happen when these are extruded?



In the next figure, extruding section (A) results in feature (1); extruding section (B) results in feature (2); extruding section (C) results in feature (3). Notice how open sections use the part adjacent surfaces, when extruded, to define a closed volume so the system can calculate a definite amount of material to add or remove. See how section (A) is extruded independently of adjacent surfaces.

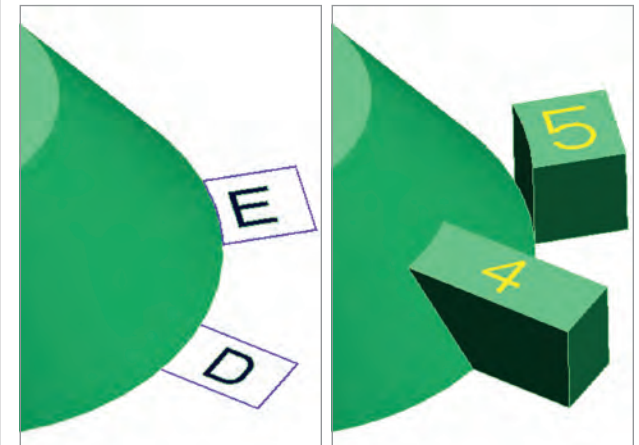


Let's see what happens when we redefine section (B) to be closed.



Can you see how feature (2) has changed?

Sometimes an open section is needed. In the following figures, section (D) is open and section (E) is closed. Extruding section (D) results in feature (4); extruding section (E) results in feature (5). Do you see how feature (4) is conforming to the part surface and feature (5) is extruded deviating from the surface, causing a gap?



The Sketcher diagnostics toolbar allows visual check for closed sections using shade closed loops as shown below.

