

Autodesk Inventor Fusion 2013 Manual

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1. Create a rough two-dimensional sketch of the basic shape of the base feature of the design.
2. Apply/modify constraints and dimensions to the two-dimensional sketch.
3. Extrude, revolve, or sweep the parametric two-dimensional sketch to create the base solid feature of the design.
- 4.

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We are basically trying to find something that integrates with Inventor. When something changes in Inventor, you just need to open the parts manual and it updates automatically. Also, something that integrates the BOM's so they don't have to retype them or export them out, which also gets updated automatically.

Parts Manual - Autodesk Community

unabridged manual to the Fusion Team. All that proves to me, is that there is a need for such a document. There are manuals present in PDF form but they tend to cover single aspects of Fusion, in combination with the related video. Indeed I've found both the individual manuals and related videos to be of excellent quality. However

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