

## Parametric Modeling With Autodesk Inventor 2014

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All modelling in Inventor is parametric. Any dimensions you add to your model, or feature parameters can be controlled through the parameters edito 'fx' symbol. Is this a dumb solid from Fusion or AutoCAD that you want to make parametric? Please mark as "Accept as Solution" if it answers your question or "Kudos" if you found it useful.

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