Introduction to Creating Geometry Using 3D Parametric CAD Modeling

Three Feature Types

- Datum Features & Non-Solid Features
- Sketched Features
  - 2D Sketch or section created first
  - 2D section(s) used to define geometry
- Predefined Features

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Datum Features & Non-Solid Features

- Purpose
  - To create and edit the size, location and orientation of other geometry
- Default Datum Features
  - Datum Planes, Datum Axes & Datum Point(s)
- User Defined, Non-Solid Features
  - Curves, planes, axes, points
    - Offset planes, angled planes, et cetera

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Sketched Features

- 2D shapes used for
  - Extrude
  - Revolve
  - Sweep
  - Blend
  - More

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Predefined Features

- Shape of the feature is predefined:
  - Round/Fillet, Chamfer
  - Holes
  - Draft
  - Shell
  - Others

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Guidelines for Creating Sketched Features

- Use default datum features
- Define flexibility into your model
- Feature type and order is important

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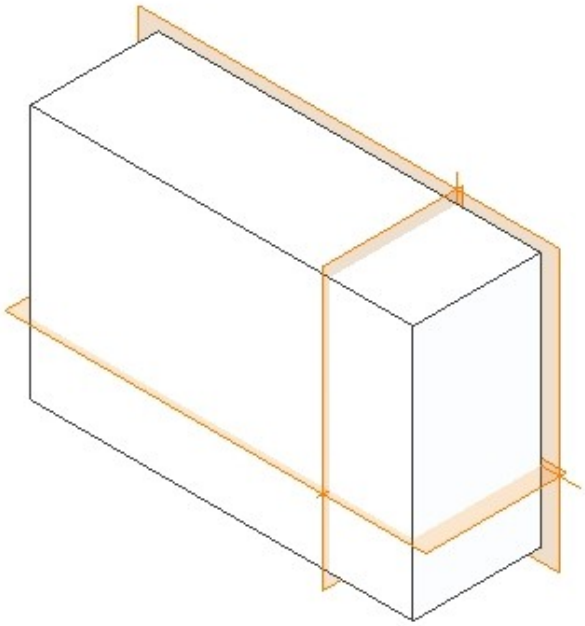
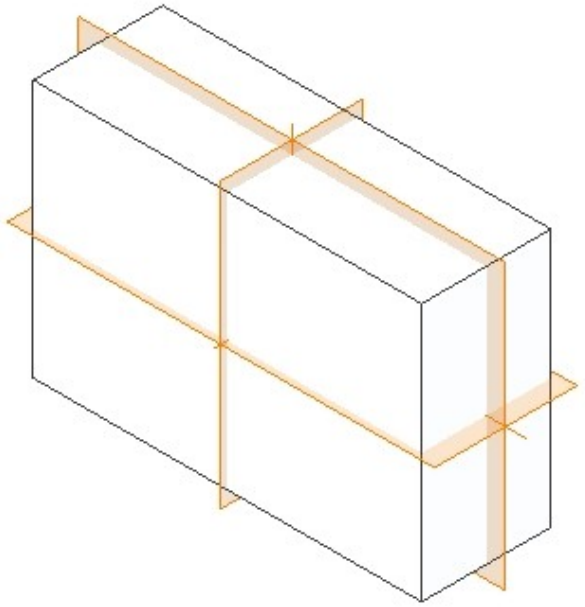
Guidelines for Creating Sketched Features

- Use default datum features
  - Define your feature so the default datum planes or axes are symmetrically or logically positioned





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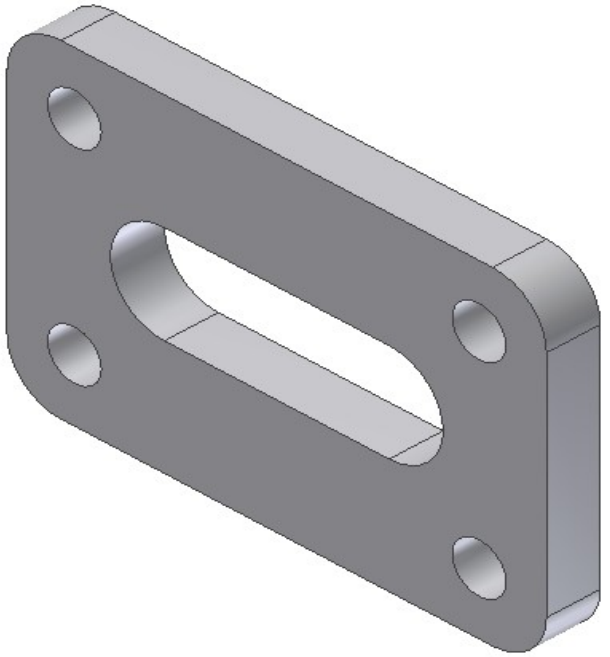
Guidelines for Creating Sketched Features

- Define flexibility into your model
  - Don't try to create “everything” into one sketch
  - Add flexibility, geometric integrity and design variation options by using separate features
  - This part could have been extruded from just one 2D sketch:
    - This would not allow the center cut to be temporarily suppressed
    - This would not allow the hole types to be changed quickly



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Guidelines for Creating Sketched Features

- Feature type and order is important
  - Define geometry in logical order
    - Example: Add rounds before using shell
    - Example: Apply draft before adding rounds

- In the example above, the shell feature is used before creating the radial pattern of holes
- In the example below, the shell feature is added after the radial pattern of holes was created