

## Technical Education Services

### Autodesk Fusion 360: Introduction to Parametric Modeling Course Length: 3 days



The Autodesk® Fusion 360™ Introduction to Parametric Modeling training course provides you with an understanding of the parametric design philosophy using the Autodesk® Fusion 360™ software. Through a hands-on, practice-intensive curriculum, you will learn the key skills and knowledge required to design models using the Autodesk Fusion 360 software.

#### Topics covered:

- Understanding the Autodesk Fusion 360 interface
- Creating, constraining, and dimensioning 2D sketches
- Creating and editing solid 3D features
- Creating and using construction features
- Creating equations and working with parameters
- Manipulating the feature history of a design
- Duplicating geometry in a design
- Placing and constraining/connecting components in a single design file
- Defining motion in a multi-component design
- Creating components and features in a multi-component design

#### Prerequisites:

- Basic computer skills



For the current course  
schedule and to register  
for this course:  
Web: [redstack.com.au](http://redstack.com.au)  
Phone: 1300 667 263

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### Autodesk Fusion 360: Introduction to Parametric Modeling

#### DAY 1

##### Chapter 1: Introduction to Autodesk Fusion 360

- Identify the key areas of the Autodesk Fusion 360 interface.
- Create a new project for use in the Autodesk Fusion 360 software.
- Load files into an Autodesk Fusion 360 project.
- Open files in the Autodesk Fusion 360 software.
- Use the design orientation commands to pan, zoom, rotate, and view a design.
- Change the Visual Style, Environment, and Effects settings to customize the display of a design.

##### Chapter 2: Creating the First Feature with Quick Shapes

- Identify and change the units for the design.
- Identify and locate the origin features of a new design.
- Create geometry using the Box, Cylinder, Sphere, Torus, and Coil quick shape modeling tools

##### Chapter 3: Creating Sketched Geometry

- Describe the general workflow to create a new sketch in an Autodesk Fusion 360 design.
- Use the sketch entity types to create lines, splines, points, rectangles, circles, and arcs.
- Sketch entities so that the required dimensions and constraints are assigned as entities are sketched.
- Add dimensions to sketch entities.
- Assign constraints to a sketch to control the required relationships between sketch entities.
- Use a sketch to create extruded or revolved geometry in a design.

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### Chapter 4: Additional Sketching Tools

- Create 2D sketch entities so that they capture design intent.
- Dimension 2D sketch entities so that they capture design intent.
- Edit existing 2D sketches.
- Move and copy sketch geometry.
- Pattern sketch geometry.

### Chapter 5: Sketched Secondary Features

- Create an extruded and revolved secondary feature.
- Create offset entities that reference existing features.
- Project geometry to create references between sketched entities and existing features.
- Create a shared version of an existing sketch so that it can be used again to create design geometry.

### Chapter 6: Pick and Place Features

- Create constant, variable, chordal, and rule-based fillets in a design.
- Create chamfers in a design
- Create simple, counterbore, and countersink holes in a design

### Chapter 7: Construction Features

- Create new construction planes, axes, and points in a design.
- Describe how using construction planes, axes, or points in a design can help you to create geometry that could not be created using the existing features or geometry.

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## DAY 2

### Chapter 8: Equations and Parameters

- Create equations between dimensions to incorporate design intent into the design.
- Create user-defined parameters in a design.

### Chapter 9: Additional Features and Operations

- Create a draft where the draft pull direction is normal to a selected plane or face.
- Create a shell feature that removes faces and assigns uniform wall thickness to the remaining faces in a design.
- Create a rib feature from a sketched section.
- Use the Split tool to split a face based on a split reference.
- Use the Scale tool to resize geometry in a design.
- Use the Thread tool to add threaded geometry to faces in a design.
- Use the Press Pull tool to efficiently create new features in a design.

### Chapter 10: Design and Display Manipulation

- Change the order of features in the BROWSER.
- Use the History Marker to change the order in which new features are added to a design.
- Temporarily remove a feature from being included as part of the design geometry.
- Create half-section views in a design to help visualize the interior of a 3D design.
- Use direct modeling techniques to edit geometry

### Chapter 11: Single Path Sweeps

- Create swept geometry using appropriate path and profile entities.
- Edit a Sweep feature.design.

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### Chapter 12: Loft Features

- Create a loft feature using appropriate profile and reference entities.
- Control the shape and weight of how lofted geometry transitions from adjacent solid geometry.
- Manipulate how the control points on a section maps to the control points on adjacent sections.

### Chapter 13: Feature Duplication Tools

- Mirror faces, bodies, and features in a design.
- Create a rectangular pattern of geometry.
- Create a circular pattern of geometry.
- Create a pattern of geometry that follows a sketched path or adjacent edges.

### Chapter 14: Distributed Design

- Describe the methods used to create an assembly in the Autodesk Fusion 360 software.
- Insert components into a design.
- Create joint origins in a model for use in assigning joint references.
- Use the Joint command to connect components in a design while maintaining the defined degrees of freedom.
- Edit a joint connection so that the joint type, references, or its values can be changed.

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### DAY 3

#### Chapter 15: Component Design Tools

- Fully constrain components relative to one another using a rigid group.
- Conduct an interference analysis between components in a design.
- Assign limits to a joint to further control its range of motion.
- Define the location of a component within its range of motion.
- Incorporate Contact Analysis into a design to limit a component's range of motion.
- Establish relationships between components that move relative to one another using the Motion Linking tool.
- Conduct a motion study. Create a new project for use in the Autodesk Fusion 360 software.

#### Chapter 16: Multi Body Design

- Design multiple bodies in a single design file.
- Create components in a single design file.
- Duplicate components in a design file.
- Add as-built joints between components in a design.

#### Chapter 17: Sculpting Geometry

- Enable the sculpt environment and understand how T-Spline surface geometry can be displayed in a model.
- Create sculpted T-Splines using the Box, Plane, Cylinder, Sphere, Torus, and Quadball quick shape modeling tools.
- Create T-Spline surfaces by referencing 2D sketch geometry that will define the shape of the surface.
- Fill the gaps in T-Spline surfaces using the Face and Fill Hole options.

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### Chapter 18: Editing Sculpted Geometry

- Manipulate points, edges, and faces in a T-Spline model to define its shape using the Edit Form command.
- Delete points, edges, and faces from a T-Spline body.
- Use specific edge, point, and face commands to manipulate the control frame of a T-Spline body.
- Assign symmetry to a T-Spline body.
- Use the Thicken command to offset a duplicate of the body.

### Chapter 19: Drawing Basics

- Create a new drawing based on a drawing template.
- Identify and place view types on a drawing file to appropriately document model geometry.
- Create exploded views of multi-component models.
- Delete and suppress drawing views.
- Edit drawing views to change the properties that were defined during view creation.
- Include views of multiple independent components in a single drawing.
- Modify the title block and border of a drawing.

### Chapter 20: Detailing Drawings

- Create dimensions to detail a drawing view.
- Create annotations to detail a drawing, such as notes, centerlines, center marks, and symbols.
- Add a parts list to a drawing.
- Create balloons that identify the components in a drawing's parts list.
- Review and edit the style options for drawing annotations.
- Output a drawing to a .PDF or a .DWG file format.

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### Chapter 21: Static Analysis Using the Simulation Environment

- Activate the Simulation environment and set up a static analysis.
- Accurately assign the material, constraints, loads, and contacts required to conduct a static analysis.
- Mesh a design using the default mesh settings.
- Solve a static analysis.
- Visualize the results of a static analysis.

### Appendix A: Outputting for 3D Printing

- Generate .STL files for 3D printing.



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### Cancellation Policy

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