

# ExtrudeFeatures Object

## Description

The ExtrudeFeatures collection object provides access to all of the ExtrudeFeature object in a part component definition and provides methods to create additional ExtrudeFeatures.

## Methods

Name	Description
<a href="#">Add</a>	Method that creates a new Extrude feature.
<a href="#">CreateExtrudeDefinition</a>	Method that creates a new ExtrudeDefinition object. The object created does not represent an extrude feature but instead is a representation of the information that defines an extrude feature. You can use this object as input to the ExtrudeFeatures.Add method to create the actual feature.

## Properties

Name	Description
<a href="#">Application</a>	Returns the top-level parent application object. When used in the context of Inventor, an Application object is returned. When used in the context of Apprentice, an ApprenticeServer object is returned.
<a href="#">Count</a>	Property that returns the number of items in this collection.
<a href="#">Item</a>	Returns the specified object from the collection.
<a href="#">Type</a>	Returns an ObjectTypeEnum indicating this object's type.

## Accessed From

[Features.ExtrudeFeatures](#), [FlatPatternFeatures.ExtrudeFeatures](#), [PartFeatures.ExtrudeFeatures](#), [SheetMetalFeatures.ExtrudeFeatures](#)

## Samples

Name	Description
<a href="#">SurfaceBodyCopy</a>	This sample demonstrates copying a surface body from one part to another. This is equivalent to the Promote command, but the API is much more flexible. In order for the sample to be self-contained, it creates two parts on the fly that will be used to demonstrate copying a body from one part to another. When copying a body into a part, you provide the surface body and a matrix to define its position in the new part. This sample creates a matrix based on the position of these parts within an assembly. This sample demonstrates the creation of a decal feature.

[Add a decal feature](#)

Displays information about all of the extrude features in the active document. A part document must be active when this is run.

[Using Inventor's error dialog](#)

Demonstrates using Inventor's error dialog.

[Edit profile of an extrude feature](#)

This sample demonstrates editing the profile of an extrude feature.

[Create and Edit an Extrude Feature with a pocket](#)

This sample demonstrates how to edit an extrude feature. It shows how to create a sketch plane at a specified orientation to existing geometry.

[Sketch profile control](#)

This sample demonstrates the usage of the Profiles API to control the shape of the profile. The sample creates three concentric circles and creates an extrusion of the region between the inner circles.

[Create sheet metal rip feature](#)

This sample demonstrates the creation of a rip sheet metal feature.

[Thread Feature Create](#)

This sample demonstrates the creation of a thread feature. It creates a cylinder in a new part document and creates a thread feature on the cylinder.

## Version

Introduced in version 5

# ExtrudeFeatures.Add Method

Parent Object: [ExtrudeFeatures](#)

## Description

Method that creates a new Extrude feature.

## Syntax

ExtrudeFeatures.**Add**( *Definition* As [ExtrudeDefinition](#) ) As [ExtrudeFeature](#)

## Parameters

Name	Type	Description
Definition	<a href="#"><u>ExtrudeDefinition</u></a>	Input ExtrudeDefinition object that defines the extrude feature you want to create. An ExtrudeDefinition object can be created using the ExtrudeFeatures.CreateExtrudeDefinition method. It can also be obtained from an existing ExtrudeFeature object.

## Samples

Name	Description
<a href="#">Delete Face, Boundary Patch and Stitch features</a>	Demonstrates creating Face, Boundary Patch and Stitch features.
<a href="#">SurfaceBody Copy</a>	This sample demonstrates copying a surface body from one part to another. This is equivalent to the Promote command, but the API is much more flexible. In order for the sample to be self-contained, it creates two parts on the fly that will be used to demonstrate copying a body from one part to another. When copying a body into a part, you provide the surface body and a matrix to define its position in the new part. This sample creates a matrix based on the position of these parts within an assembly.
<a href="#">Add a decal feature</a>	This sample demonstrates the creation of a decal feature.
<a href="#">Derived Parts and Assemblies</a>	This sample demonstrates the use of the API to create derived parts and assemblies.
<a href="#">Using Inventor's error dialog</a>	Demonstrates using Inventor's error dialog.
<a href="#">Extrude Feature - Create Block with Pocket</a>	This sample demonstrates creating a simple solid consisting a block with a pocket. It shows how to create a sketch plane at a specified orientation to existing geometry.
<a href="#">Edit profile of an extrude feature</a>	This sample demonstrates editing the profile of an extrude feature.
<a href="#">Extrude sketch text</a>	This sample demonstrates the creation of an extrude feature from sketch text.
<a href="#">Add iMate Definition</a>	Add iMate definitions using AddMateiMateDefinition and AddInsertiMateDefinition.
<a href="#">Create and Edit an Extrude Feature with a pocket</a>	This sample demonstrates how to edit an extrude feature. It shows how to create a sketch plane at a specified orientation to existing geometry.
<a href="#">Sketch from Face Silhouette</a>	This sample creates a cylindrical solid, creates a new sketch plane and creates some new sketch lines from the actual edges and the apparent (silhouette) edges of the cylinder.
<a href="#">Sketch profile control</a>	This sample demonstrates the usage of the Profiles API to control the shape of the profile. The sample creates three concentric circles and creates an extrusion of the region between the inner circles.
<a href="#">Create sheet metal rip feature</a>	This sample demonstrates the creation of a rip sheet metal feature.
<a href="#">Thread Feature Create</a>	This sample demonstrates the creation of a thread feature. It creates a cylinder in a new part document and creates a thread feature on the cylinder.

## Version

Introduced in version 2012

# ExtrudeFeatures.Application Property

Parent Object: [ExtrudeFeatures](#)

## Description

Returns the top-level parent application object. When used in the context of Inventor, an Application object is returned. When used in the context of Apprentice, an ApprenticeServer object is returned.

## Syntax

ExtrudeFeatures.**Application()** As Object

## Property Value

This is a read only property whose value is an Object.

## Version

Introduced in version 5

# ExtrudeFeatures.Count Property

Parent Object: [ExtrudeFeatures](#)

## Description

Property that returns the number of items in this collection.

## Syntax

ExtrudeFeatures.**Count()** As Long

## Property Value

This is a read only property whose value is a Long.

## Version

Introduced in version 5

# ExtrudeFeatures.CreateExtrudeDefinition Method

Parent Object: [ExtrudeFeatures](#)

## Description

Method that creates a new ExtrudeDefinition object. The object created does not represent an extrude feature but instead is a representation of the information that defines an extrude feature. You can use this object as input to the ExtrudeFeatures.Add method to create the actual feature.

## Remarks

The ExtrudeDefinition object returned is fully defined and can be used to create an extrude feature. However, defaults are used for extrude options (including a distance extent type with a value of “1.0 in”), so you may want to change some of the property values of the ExtrudeDefinition object before using it to create a feature.

## Syntax

ExtrudeFeatures.**CreateExtrudeDefinition(** *Profile* As [Profile](#), *Operation* As [PartFeatureOperationEnum](#) **) As** [ExtrudeDefinition](#)

## Parameters

Name	Type	Description
Profile	<a href="#">Profile</a>	Input Profile object that specifies the sketch profile to use for the extrude feature.
Operation	<a href="#">PartFeatureOperationEnum</a>	Input that specifies the type of operation used to add the feature to the model. Valid inputs are kNewBodyOperation, kJoinOperation, kCutOperation, kIntersectOperation and kSurfaceOperation.

## Samples

Name	Description
<a href="#">Delete Face, Boundary Patch</a>	Demonstrates creating Face, Boundary Patch and Stitch features.

[and Stitch  
features](#)

This sample demonstrates copying a surface body from one part to another. This is equivalent to the Promote command, but the API is much more flexible. In order for the sample to be self-contained, it creates two parts on the fly that will be used to demonstrate copying a body from one part to another. When copying a body into a part, you provide the surface body and a matrix to define its position in the new part. This sample creates a matrix based on the position of these parts within an assembly.

[Add a decal  
feature](#)

This sample demonstrates the creation of a decal feature.

[Derived Parts  
and Assemblies](#)

This sample demonstrates the use of the API to create derived parts and assemblies.

[Using Inventor's  
error dialog](#)

Demonstrates using Inventor's error dialog.

[Extrude Feature  
- Create Block  
with Pocket](#)

This sample demonstrates creating a simple solid consisting a block with a pocket. It shows how to create a sketch plane at a specified orientation to existing geometry.

[Edit profile of  
an extrude  
feature](#)

This sample demonstrates editing the profile of an extrude feature.

[Extrude sketch  
text](#)

This sample demonstrates the creation of an extrude feature from sketch text.

[Add iMate  
Definition](#)

Add iMate definitions using AddMateiMateDefinition and AddInsertiMateDefinition.

[Create and Edit  
an Extrude  
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This sample demonstrates how to edit an extrude feature. It shows how to create a sketch plane at a specified orientation to existing geometry.

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This sample creates a cylindrical solid, creates a new sketch plane and creates some new sketch lines from the actual edges and the apparent (silhouette) edges of the cylinder.

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This sample demonstrates the usage of the Profiles API to control the shape of the profile. The sample creates three concentric circles and creates an extrusion of the region between the inner circles.

[Create sheet  
metal rip feature](#)

This sample demonstrates the creation of a rip sheet metal feature.

[Thread Feature  
Create](#)

This sample demonstrates the creation of a thread feature. It creates a cylinder in a new part document and creates a thread feature on the cylinder.

## Version

Introduced in version 2012

# ExtrudeFeatures.Item Property

Parent Object: [ExtrudeFeatures](#)

## Description

Returns the specified object from the collection.

## Syntax

ExtrudeFeatures.**Item( *Index* As Variant ) As [ExtrudeFeature](#)**

## Property Value

This is a read only property whose value is an [ExtrudeFeature](#).

## Parameters

Name	Type	Description
<i>Index</i>	Variant	Input Variant value that specifies the feature to return. This can be either a numeric value indicating the index of the item in the collection or it can be a string indicating the feature name. If an out of range index or a name of a non-existent feature is provided, an error occurs.

## Samples

Name	Description
<a href="#">Sketch Share</a>	This sample demonstrates setting a sketch so it is shared.

## Version

Introduced in version 5

# ExtrudeFeatures.Type Property

Parent Object: [ExtrudeFeatures](#)

## Description

Returns an ObjectTypeEnum indicating this object's type.

## Syntax

ExtrudeFeatures.**Type() As [ObjectTypeEnum](#)**

## Property Value

This is a read only property whose value is an [ObjectTypeEnum](#).

## Version

Introduced in version 5