

ArcGIS® 9

Geoprocessing in ArcGIS Tutorial



Copyright © 2001–2006 ESRI
All rights reserved.
Printed in the United States of America.

The information contained in this document is the exclusive property of ESRI. This work is protected under United States copyright law and other international copyright treaties and conventions. No part of this work may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, or by any information storage or retrieval system, except as expressly permitted in writing by ESRI. All requests should be sent to Attention: Contracts Manager, ESRI, 380 New York Street, Redlands, CA 92373-8100, USA.

The information contained in this document is subject to change without notice.

DATA CREDITS

Geographic data used in the tutorial provided courtesy of San Diego Association of Governments (SANDAG) and is used herein with permission.

Some of the illustrations in this work were made from data supplied by Collins Bartholomew Ltd.; IHS Energy; Riley County, Kansas, GIS; and SANDAG. They are used herein with permission.

AUTHOR

Catherine Jones, Jill McCoy

U.S. GOVERNMENT RESTRICTED/LIMITED RIGHTS

Any software, documentation, and/or data delivered hereunder is subject to the terms of the License Agreement. In no event shall the U.S. Government acquire greater than RESTRICTED/LIMITED RIGHTS. At a minimum, use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in FAR §52.227-14 Alternates I, II, and III (JUN 1987); FAR §52.227-19 (JUN 1987) and/or FAR §12.211/12.212 (Commercial Technical Data/Computer Software); and DFARS §252.227-7015 (NOV 1995) (Technical Data) and/or DFARS §227.7202 (Computer Software), as applicable. Contractor/Manufacturer is ESRI, 380 New York Street, Redlands, CA 92373-8100, USA.

ESRI, the ESRI globe logo, 3D Analyst, AML, ArcCatalog, ArcEditor, ArcGIS, ArcGlobe, ArcIMS, ArcInfo, ArcMap, ArcReader, ArcScene, ArcSDE, ArcToolbox, ArcView, BusinessMAP, ModelBuilder, RouteMAP, StreetMap, GIS by ESRI, and www.esri.com are trademarks, registered trademarks, or service marks of ESRI in the United States, the European Community, or certain other jurisdictions.

Other companies and products mentioned herein are trademarks or registered trademarks of their respective trademark owners.

Geoprocessing in ArcGIS Tutorial

IN THIS TUTORIAL

- **Exercise 1: Finding coastal sage scrub near proposed roads**
- **Exercise 2: Joining fields to the vegetation data**
- **Exercise 3: Clipping data to the study area**
- **Exercise 4: Finding high-quality habitat patches**
- **Exercise 5: Finding habitat patches impacted by proposed roads**

This tutorial will take you on a tour of the geoprocessing functionality within ArcGIS as you find areas in San Diego County with the best potential to support the California gnatcatcher (*Poliophtila californica*).

The California gnatcatcher is a small nonmigratory songbird that resides predominantly in southwestern California and northwestern Baja. The gnatcatcher population has been in decline over several decades because of extensive habitat loss due predominantly to urbanization. Most habitats are fragmented by roads; building tracts; and other barriers insurmountable to the small, weak-flying bird. The species is listed as threatened and is dependent upon certain vegetation types for its survival.

The goal of this tutorial is to find patches of potentially high-quality habitat—then to identify which patches would be impacted by proposed roads. New roads will only serve to further fragment these fragile regions, so those high-quality habitat patches that will be impacted need to be identified.

In achieving the above goal, you will learn how to:

- Run geoprocessing tools through dialog boxes and the command line.
- Create models inside, and add scripts to, toolboxes.

By following the tutorial, you'll quickly learn how to perform your own geoprocessing tasks with ArcGIS. Included is an estimate of the time it will take to complete each exercise. You can work through the entire tutorial or complete each exercise one at a time. Although recommended, you don't have to complete the exercises in sequence.

Tutorial requirements

- It is assumed that you have installed ArcGIS Desktop before beginning this tutorial.
- You can perform Exercise 1 with an ArcView, ArcEditor, or ArcInfo license. The rest of the exercises require an ArcInfo license.
- The data required is included on the ArcGIS Desktop CD. After running the ArcGIS setup, on the Additional Installation Components dialog box, check to install the ArcGIS Tutorial Data. In the ArcGIS Tutorial Data Setup wizard, select to install the Geoprocessing data (the default installation path is `arcgis\ArcTutor\Geoprocessing`) on the drive where the tutorial data is installed.

Climate.shp, futrds, majorrds.shp, study_quads.shp, vegtype, and vegtype.dbf were provided courtesy of the San Diego Association of Governments (SANDAG) for educational purposes only. The data has been manipulated for the purpose of the tutorial. Assumptions applied and actual outcomes may vary. ESRI is not inviting reliance on this data or the methodologies followed. You should always verify actual data and exercise your own professional judgment when interpreting any outcomes.

Data	Description
climate.shp	General climate zones.
elevlt250.shp	Areas where the elevation is less than 250 m. This shapefile was derived from a United States Geological Survey (USGS) digital elevation model (DEM).
futrds	A geodatabase feature class of all proposed freeways and roads.
habitat_analysis.gdb	A file geodatabase containing the feature classes futrds and vegtype.
majorrds.shp	All major roads and freeways.
multi_clip.py	A Python script used to clip multiple feature classes and place them into a personal geodatabase.
query.txt	Includes an expression that can be copied and pasted into the expression box on the Select tool's dialog box used in Exercise 4. It can be used as an alternative to the Query Builder or typing the expression.
slopelt40.shp	Areas where the slope of the land is less than 40 percent. This shapefile was derived from a USGS DEM.
study_quads.shp	The study area. USGS 7.5 quad boundaries covering the towns of La Jolla and La Mesa in San Diego County.
vegtype	A geodatabase feature class of vegetation type information.

vegetable.dbf A table containing fields that will be joined to a layer you'll create from vegtype. The fields include a vegetation description field and a field indicating whether the vegetation type is suitable for habitation by the gnatcatcher.

The *ArcCatalog*[™] *tree* provides access to datasets stored on disk. In the ArcCatalog tree, click a dataset, then click the Metadata tab to explore the *metadata* (information) of each dataset in detail. Explore metadata for the tutorial folder to gain an overview of the data contained in the folder and how it was created.

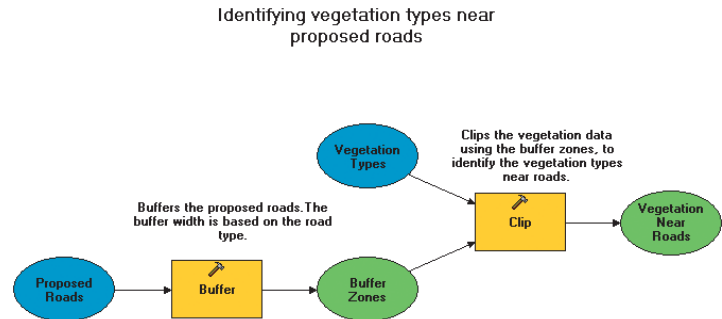
Exercise 1: Finding coastal sage scrub near proposed roads

When working with ArcGIS you'll likely need to perform geoprocessing tasks at some stage in your work flow. In ArcGIS geoprocessing tasks are performed by running *tools*. Many times your work flow takes multiple steps to produce the desired result. You can perform each geoprocessing task individually by running a system tool via its dialog box or the command line, or you can string tasks together to build a model or a script.

In this exercise you'll copy the tutorial data locally using ArcCatalog, then in ArcMap™ you'll review and run an existing model that runs a sequence of system tools. The model identifies the vegetation types near proposed roads.

The interface you use to build and edit models is the *ModelBuilder window*. By adding a tool to a ModelBuilder window and supplying values for the tool's *parameters*, you construct a *process*. A model is built by connecting processes together. The model can be run with a single click. A model increases your efficiency because you can easily alter input data or other parameter values, then reexecute the model to produce different results.

The diagram that follows shows the model you'll run. In the model, the roads are first buffered using values from the Distance field in the roads attribute table. The value for the Distance field of each road segment depends on the width of the road. The wider the road, the farther the distance (buffer), due to the impact on the surrounding area. The output from the Buffer tool (Buffer Zones) is used to clip the vegetation data (Vegetation Types) to identify the vegetation types within the buffer zones (Vegetation Near Roads).



The above diagram shows the model you'll run to identify vegetation types near proposed roads.

After running the model, you'll select all features in the final *output data* that represent coastal sage scrub vegetation to identify the patches of this vegetation type that would be impacted by the addition of the proposed roads.

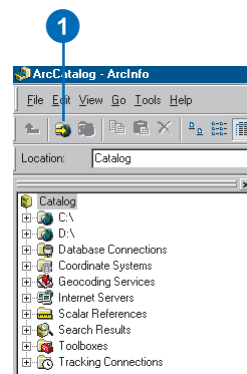
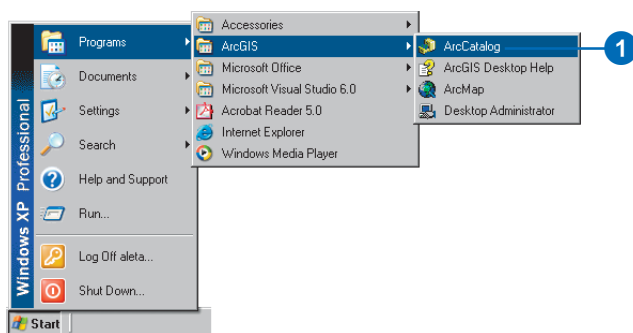
This exercise will take approximately 15 minutes to complete.

Organizing your data

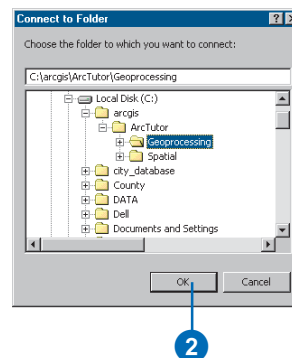
Before working with geoprocessing tools you'll first organize your tutorial data using ArcCatalog.

Starting ArcCatalog

1. Start ArcCatalog by either double-clicking a shortcut installed on your desktop or using the Programs list in your Start menu.



2. Type the path or navigate to the location where you installed the tutorial data. For example, if you installed ArcGIS on your C:\ drive, type "C:\arcgis\ArcTutor\Geoprocessing", then click OK to establish a folder connection.



Locating the tutorial data

By connecting to a folder in ArcCatalog you can quickly see the folders and data sources it contains. You'll now begin organizing your tutorial data by establishing a folder connection to its location.

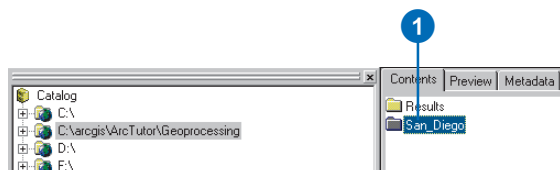
1. Click the Connect to Folder button.

Your new folder connection to the tutorial data is listed in the ArcCatalog tree.

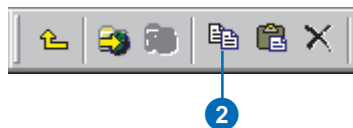
Creating a working copy of the tutorial data

You'll now copy the tutorial data into a folder on a local disk to maintain the integrity of the original data. Once it has been copied, you will then create a connection to the folder containing the data.

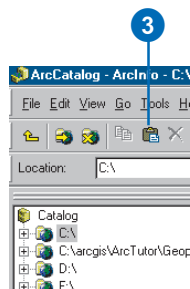
1. Click the connection to the tutorial data and click the San_Diego folder in the Contents tab.



2. Click the Copy button on the Standard toolbar.

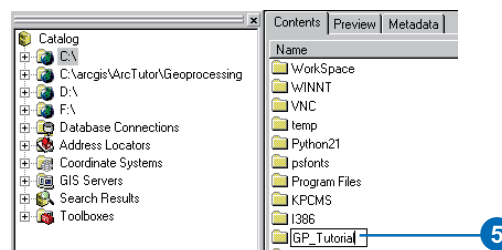


3. Click the C:\ (or an alternative drive) folder connection in the ArcCatalog tree, then click the Paste button.



A new folder called San_Diego will appear in the Contents list.

4. Click the new San_Diego folder in the Contents tab to select it, then click again to rename the folder.
5. Type "GP_Tutorial" and press Enter.



6. Click the Connect to Folder button again, create a connection to your GP_Tutorial folder, and click OK.

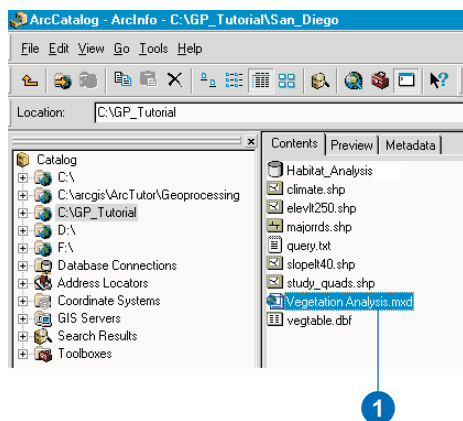
Your new folder connection, for example, C:\GP_Tutorial, is now listed in the ArcCatalog tree. You will access this connection many times during this tutorial.

Reviewing an existing model

Models can be run within any ArcGIS Desktop application. In this exercise you'll work within ArcMap to identify vegetation types near proposed roads.

Opening the Vegetation Analysis.mxd

1. Click the connection to your GP_Tutorial folder in the ArcCatalog tree, click the Contents tab, then double-click the map document Vegetation Analysis to open it.



As you have finished using ArcCatalog in this exercise, close your ArcCatalog session.

2. Click ArcCatalog on the desktop taskbar to maximize it.
3. Click File, then Exit to exit the ArcCatalog session.

Opening the ArcToolbox window

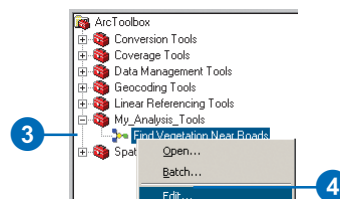
The *ArcToolbox™* window is dockable in any ArcGIS Desktop application. It provides access to tools you have stored on disk.

1. Click the Show/Hide ArcToolbox Window button on the Standard toolbar to open the ArcToolbox window.

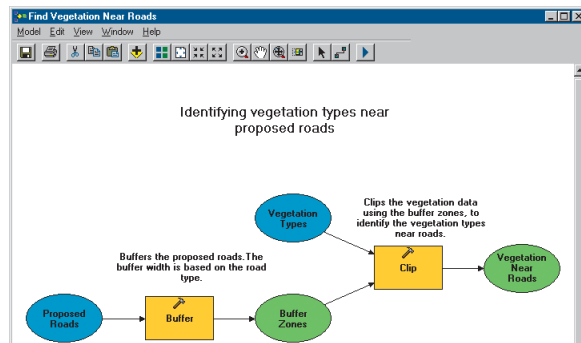


Toolboxes can contain system tools—tools installed by default—or they can contain custom tools that you have created, such as models or scripts, that might run a number of tools at one time. You'll now review a model that has already been built using two system tools.

2. Right-click the ArcToolbox folder in the ArcToolbox window and click Add Toolbox. Click the Look in dropdown arrow and click the connection to your local copy of the tutorial data—for example, C:\GP_Tutorial. Double-click Habitat_Analysis.gdb, then click the My_Analysis_Tools toolbox and click Open.
3. Expand the My_Analysis_Tools toolbox.
4. Right-click the model Find Vegetation Near Roads and click Edit.

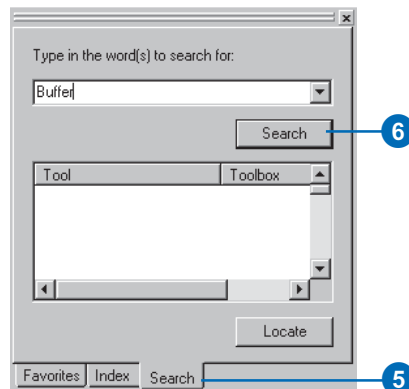


This opens a previously created model in its ModelBuilder window so you can review it.



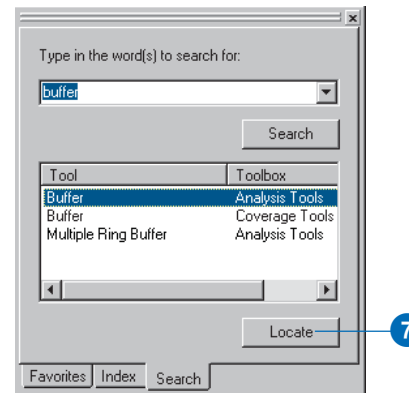
Input data elements that represent existing data appear as blue ovals in the model. *Tool elements* that represent tools stored on disk appear as yellow rectangles. *Derived data* elements, which represent data that will be created when the model is run, appear as green ovals.

5. Click the Search tab on the ArcToolbox window.
6. Type “Buffer”, then click Search to locate the Buffer tool in the ArcToolbox window.

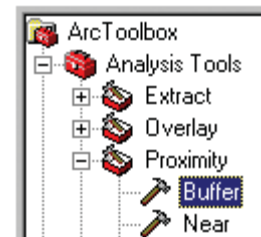


If you have the Analysis Tools toolbox and the Coverage Tools toolbox added to the ArcToolbox window, you’ll see two Buffer tools in the list of found tools. The Buffer tool in the Coverage Tools toolbox only works with coverages as inputs. The tool in the Analysis Tools toolbox works with feature classes as inputs.

7. Click the Analysis Tools Buffer and click Locate.



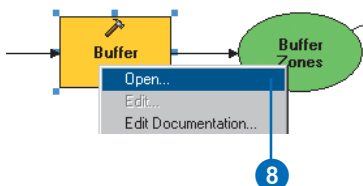
The tool is located in the ArcToolbox window.



Rather than opening the Buffer tool’s dialog box from the ArcToolbox window, you’ll open it within the ModelBuilder window to view the parameter values set for the tool.

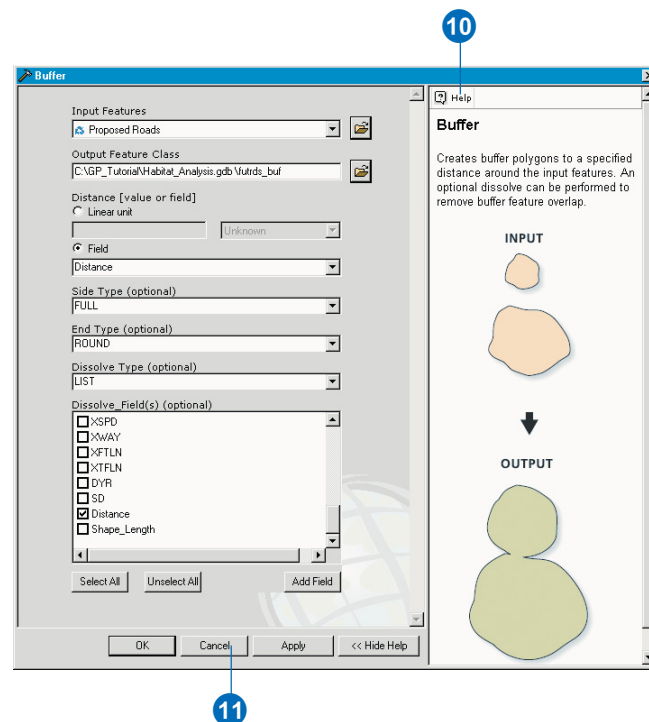
8. Right-click the Buffer tool element in the model and click Open to open the tool's dialog box.

Buffers the proposed roads. The buffer width is based on the road type.



A description for the tool is displayed in the Help panel by default.

9. Click each parameter in the dialog box to view help in the Help panel for each.
10. Click Help to get more detailed help for the tool if desired.
11. Click Cancel on the Buffer dialog box.



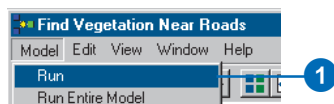
12. Right-click the Clip tool element and click Open to open the tool's dialog box.
13. Examine the parameter values set for the tool and view the help for each parameter as you did with the Buffer tool, then click Cancel.

In this model there are two processes, each consisting of a tool and its parameter values—one using the Buffer tool and the other using the Clip tool. The two processes are connected together to build the model.

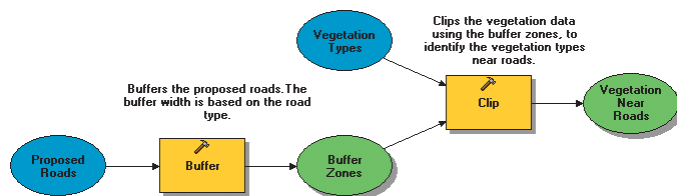
Running the model

The model is colored in, meaning it is in a ready-to-run state, as all necessary parameter values have been supplied in each tool's dialog box.

1. Click Model, then Run.



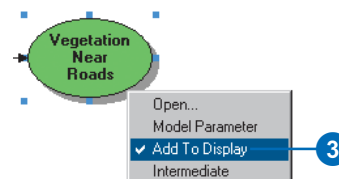
When the model runs, the tool of the process that is currently executing is highlighted in red. When each process has executed, a dropshadow appears behind the tool and its derived data element, indicating that the process has run and the derived data has been created on disk.



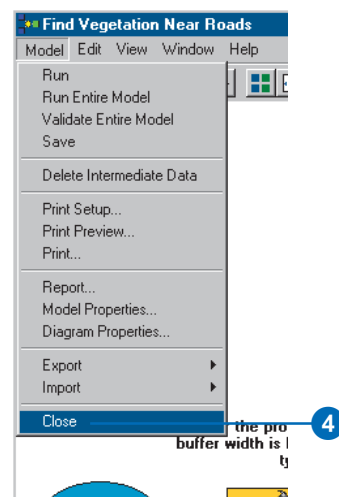
2. Click Close on the progress dialog box.

The final result is added to the ArcMap display by default, because the option to add derived data to the display is checked on the context menu of the derived data element Vegetation Near Roads.

3. Right-click the Vegetation Near Roads derived data element in the model to see that Add To Display is checked.

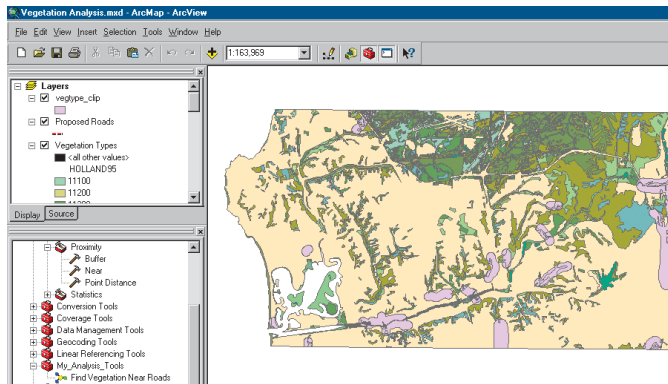


4. Click Model, then Close, then click Yes to save the changes you have made to the model.



- Examine the final result (vegtype_clip) in the ArcMap display. In the graphic below, the color used to symbolize vegtype_clip is purple. The color is randomly generated for the output feature class, so the color used in symbolizing your feature class might be different.

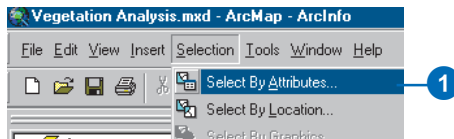
All vegetation types near proposed roads have been identified.



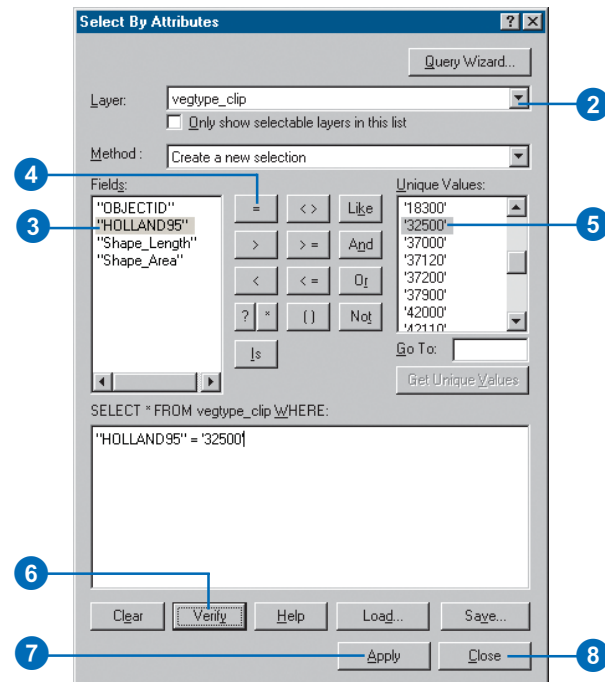
Selecting areas of coastal sage scrub

You'll now select areas where the vegetation type is San Diego coastal sage scrub to see how much of this particular vegetation type would be impacted by the proposed roads.

- Click Selection on the Main menu and click Select By Attributes.



- Click the Layer dropdown arrow and click vegtype_clip.
 - Double-click the field "HOLLAND95" to add it to the expression box.
 - Click = to add the equals operator to the expression box.
 - Click Get Unique Values then double-click the unique value '32500' to add it to the expression box.
- HOLLAND95 is a code for vegetation type. The value 32500 corresponds to San Diego coastal sage scrub.
- Click Verify to check the syntax of the expression.
 - Click Apply to select the features.
 - Click Close.



9. Examine the selection in the ArcMap display. All areas near proposed roads where the vegetation is San Diego coastal sage scrub are identified.



This brings you to the end of Exercise 1. This exercise has given a brief introduction to running models. The rest of this tutorial goes into more detail about geoprocessing in ArcGIS. You'll use more of the system tools, run a script to batch process multiple inputs, and build more complex models. You can continue on to Exercise 2 or stop and complete the tutorial at a later time. If you do move on to Exercise 2 now, do not delete your working copy of the tutorial data, which is your GP_Tutorial folder, and do not remove the folder connection that accesses it from ArcCatalog.

10. Click File on the Main menu of ArcMap and click Exit.
Click Yes to save your changes.

Exercise 2: Joining fields to the vegetation data

There are multiple system tools that you can access, such as dissolve or union, through *system toolboxes* that are installed with ArcGIS.

In ArcCatalog you can choose whether to access system tools via the ArcCatalog tree or the ArcToolbox window. If your toolboxes are located in different locations on disk, it is recommended that you use the ArcToolbox window to centralize the location of the toolboxes you use most frequently. In applications in which the ArcCatalog tree is not accessible, such as in ArcMap, you'll always work with tools using the ArcToolbox window.

Some manipulations you make to your data should be done as preparation steps, so you can avoid the repetition of unnecessary steps in your main work flow. In this exercise you'll first locate the system toolboxes on disk, then you'll open the ArcToolbox window. Once you have specified appropriate environment settings, you'll join two fields to a layer you'll create from the vegetation data (vegtype). This exercise will take approximately 20 minutes to complete.

Setting up

1. Start ArcCatalog by using the Programs list in your Start menu.

If you did the previous exercise, skip the next step and go to the next section, 'Locating the system toolboxes'.

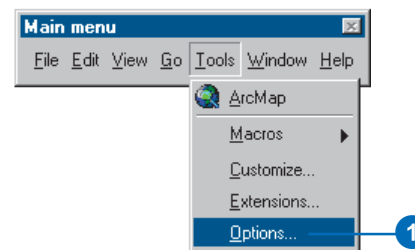
2. Copy the GP_Tutorial folder from `arcgis\ArcTutor\Geoprocessing\Results\Ex1` on the drive where you installed ArcGIS to a local drive, such as your C:\ drive.

Locating the system toolboxes

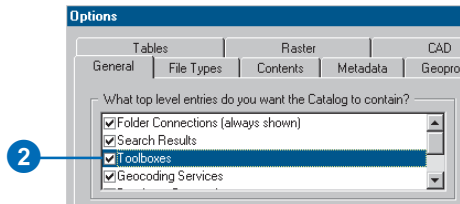
The Toolboxes folder in the ArcCatalog tree contains two folders: a My Toolboxes folder and a System Toolboxes folder. The *My Toolboxes folder* points to a location on disk that can be changed in the Geoprocessing tab of the Options dialog box. It is the location used for new toolboxes created inside the ArcToolbox window. The My Toolboxes folder also contains a History toolbox within which *history models* are created automatically for each session, recording the tools and parameter values used. For more information on the My Toolboxes folder, see 'Changing the default location of the My Toolboxes folder' in Chapter 4. For more information on history tools, see 'Keeping track of geoprocessing operations' in Chapter 3.

The System Toolboxes folder contains the toolboxes that come with ArcGIS. The Toolboxes folder is not added to the ArcCatalog tree by default.

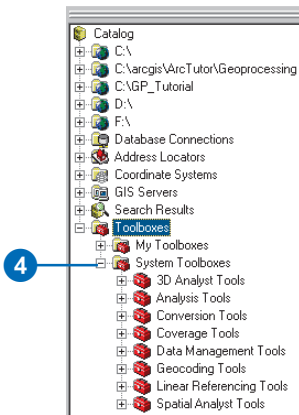
1. Click the Tools menu and click Options.



- Click the General tab and check Toolboxes if it is not already checked.



- Click OK on the Options dialog box.
- Expand the Toolboxes folder, then expand the System Toolboxes folder to view the toolboxes that are available.



Each toolbox contains a set of system tools that can be used to geoprocess your data.

Working with the ArcToolbox window

The ArcToolbox window provides a way to centralize the location of the tools you use most frequently. You can remove toolboxes you don't use often from the ArcToolbox

window and add them back at any time. The ArcToolbox window is dockable inside any ArcGIS Desktop application.

- Click the Show/Hide ArcToolbox Window button on the Standard toolbar of ArcCatalog to open the ArcToolbox window.

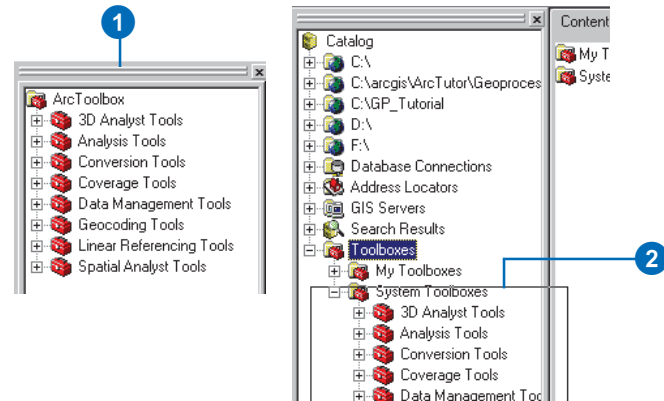


The window is placed within the application. Its position may vary depending on the windows you have open.

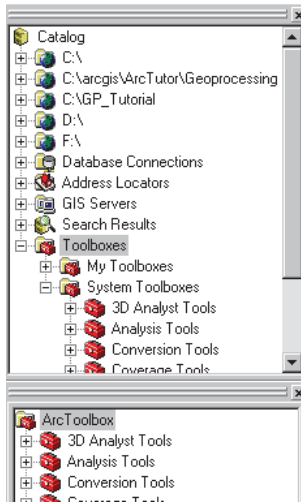
Docking the ArcToolbox window

The ArcToolbox window can be placed anywhere in the application or on the desktop.

- Click and drag the bar at the top of the ArcToolbox window.
- Place the ArcToolbox window over the ArcCatalog tree, then drop the panel. The outline of the ArcToolbox window helps you to place it in the desired position.



Your ArcToolbox window should now display under the ArcCatalog tree, as the following diagram shows.



Note: If you hold down the Ctrl key and drag the ArcToolbox window, it will not dock itself to the application.

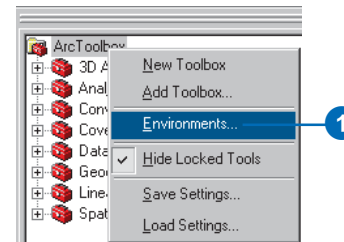
Setting the geoprocessing environment

Before you start to perform geoprocessing tasks on your data, you should set up any relevant *environment settings*. The application Environment Settings dialog box allows you to change the default application settings that are used when you run geoprocessing tools.

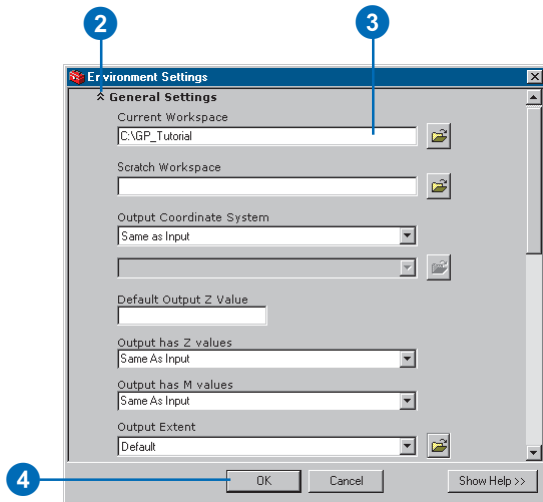
Note: You can specify environment settings for the application that will apply to all tools. Alternatively, you can set environment settings for a specific model or for a process within a model. Environment settings for the application are used by default unless they are specified for

a model or for a process within a model. Settings for a process will override settings for the model and for the application. Settings for a model will override settings for the application.

1. Right-click the ArcToolbox window and click **Environments...** to open the Environment Settings dialog box for the application.



2. Click General Settings to expand its contents.
3. Click the Current Workspace parameter and type the path to your GP_Tutorial folder—for example, “C:\GP_Tutorial”—for its value. Alternatively, click the Browse button to the right of the parameter and navigate to this location.
4. Click OK.



Now that the *current workspace* is set, you can simply type the name for the input and output parameter values. Data will be taken from, and placed in, this location, saving you time typing in or browsing to the location of data each time you run a tool.

Adding fields to the vegetation data

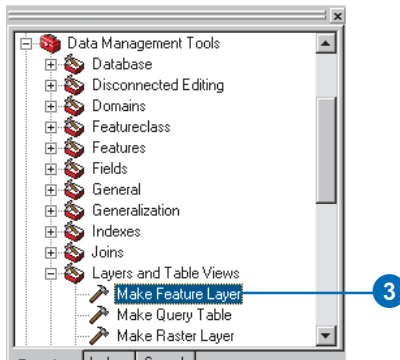
If you performed Exercise 1, you selected areas where the vegetation type was San Diego coastal sage scrub—polygons with a value of 32500 for the HOLLAND95 field

in the attribute table of vegtype.shp. This vegetation type is preferred by the California gnatcatcher. However, there are other vegetation types that are also suitable. You'll now join two fields from a dBASE® Format (DBF) table (vegtable.dbf) to a layer you'll create from your vegetation feature class, vegtype, located inside Habitat_Analysis.gdb. The fields you'll join to the layer are HABITAT—a Boolean field indicating whether or not the vegetation type is suitable for habitation by the gnatcatcher—and VEG_TYPE—a description for the HOLLAND95 code.

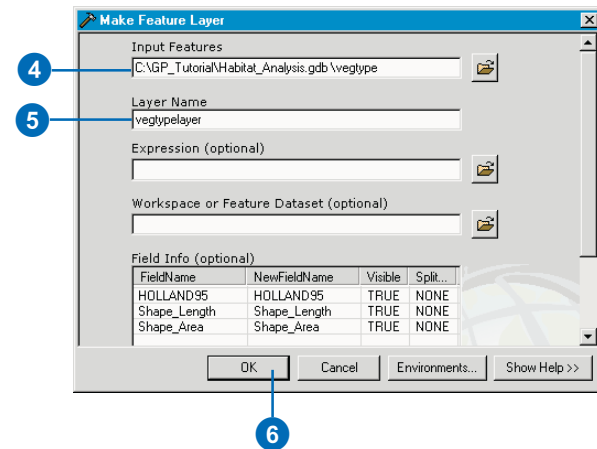
As when joining fields in ArcMap, the Add Join tool only accepts in-memory *layers* that reference geographic data stored in a data source. You'll create a layer—vegtypelayer—and join vegtable.dbf to it. Then you'll create a new layer from the result of the Add Join tool, so you can rename the fields to be more meaningful and make visible those fields that you want to see in the attribute table of the output layer. You'll then copy the features in the layer to a new feature class that you'll save inside Habitat_Analysis.gdb.

Creating a layer

1. Expand your GP_Tutorial connection, then click the plus sign next to the Habitat_Analysis.gdb to view its contents.
2. In the ArcToolbox window, expand the Data Management Tools toolbox, then expand the Layers and Table Views toolset.
3. Right-click Make Feature Layer and click Open or double-click the tool.



4. Drag the feature class vegtype from within the Habitat_Analysis geodatabase into the Input Features text box.
5. Type vegtypelayer for the name of the in-memory layer to create.
6. Click OK.

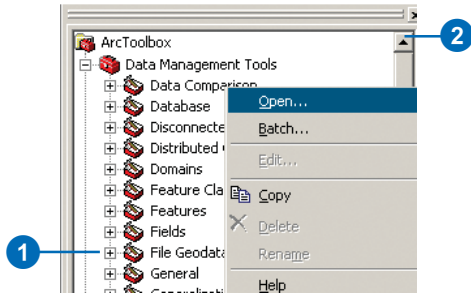


An in-memory layer is created. You cannot see this layer in the ArcCatalog tree, but it can be used as input to geoprocessing tools that support it.

7. Click Details on the progress dialog box to view the execution messages produced.
8. Check Close this dialog when completed successfully on the progress dialog box, then click Close.

Joining the DBF table to vegtypelayer

1. In the ArcToolbox window, expand the Data Management Tools toolbox, then expand the Joins toolset.
2. Right-click Add Join and click Open.



Notice that all required parameters in the Add Join dialog box need values set for them. There is a green circle next to these parameters, indicating that a value needs to be specified.

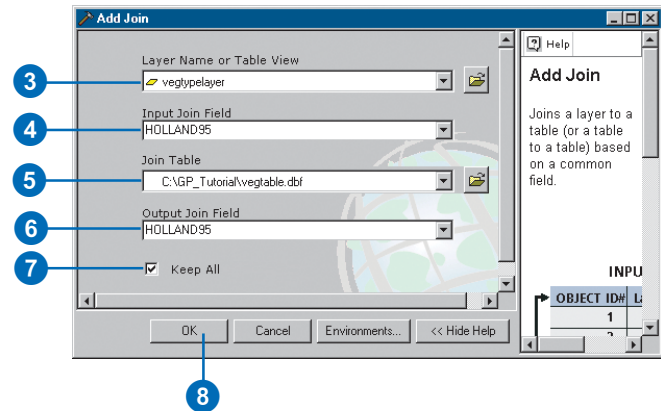
3. Click the dropdown arrow and click vegtypelayer to use the created in-memory layer as the value for the Layer Name or Table View parameter.

The Add Join tool appends attributes from one table to another using a common field. Here, the common field is named the same in the layer and in the table (HOLLAND95).

4. Click the dropdown arrow for the Input Join Field parameter and click HOLLAND95.
5. Type “vegetable.dbf” for the value of the Join Table parameter.

The path to the current workspace is supplied for you when you click another parameter.

6. Click the dropdown arrow for the Output Join Field parameter and click HOLLAND95.
7. Leave the default Keep All to retain all records in the result.
8. Click OK.

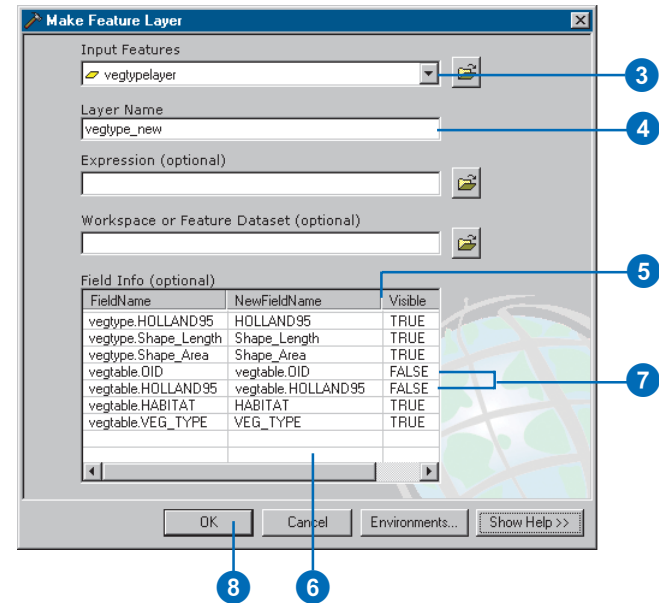


Vegtypelayer will now contain two fields: HABITAT and VEG_TYPE.

Renaming and choosing visible fields

You'll now create an in-memory layer from vegtypelayer with a subset of fields. You'll also rename the fields to remove the concatenated text introduced by joining fields.

1. In the ArcToolbox window, expand the Layers and Table Views toolset located inside the Data Management Tools toolbox.
2. Right-click Make Feature Layer and click Open.
3. Click the Input Features parameter dropdown arrow and click the in-memory layer vegtypelayer.
4. Type "vegtype_new" for the value of the parameter Layer Name. This is the name of the output layer to create. The output layer is an in-memory layer that references the data source of the layer set for the Input Features parameter.
5. In the Field Info table, click and widen the NewFieldName column.
6. In the Field Info table, click vegtype.HOLLAND95 in the NewFieldName column and remove the text vegtype.. Remove the text vegtype. from vegtype.Shape_Length and vegtype.Shape_Area. Remove the text vegetable. from vegetable.HABITAT and vegetable.VEG_TYPE.
7. Click TRUE in the Visible column for both vegetable.OID and vegetable.HOLLAND95 and change it to FALSE. You don't need the OID field, and you have two HOLLAND95 fields, so one can be removed from view.
8. Click OK.

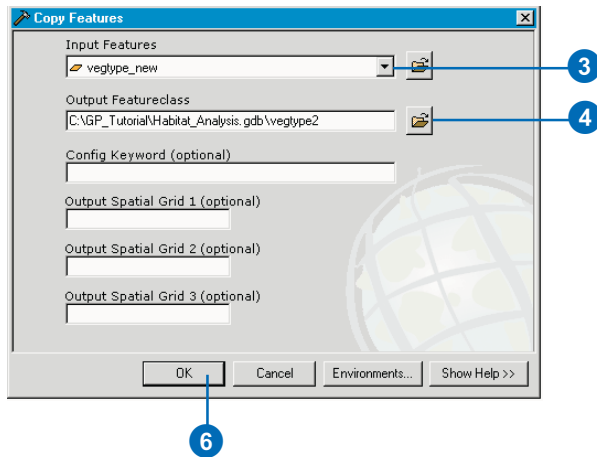


Creating a feature class from the layer

You'll now copy the features in the layer to a feature class using the Copy Features tool.

1. In the ArcToolbox window, expand the Features toolset located inside the Data Management Tools toolbox.
2. Right-click Copy Features and click Open.

3. Click the dropdown arrow for the Input Features parameter and click the vegtype_new layer you created.
4. Click the Browse button next to the parameter Output Feature Class.
5. Click the Look in dropdown arrow and click the connection to the tutorial data (C:\GP_Tutorial). Double-click Habitat_Analysis.gdb, then type “vegtype2” for the name of the output feature class to create inside the geodatabase.
6. Click Save on the Output Feature Class dialog box, then click OK.



You'll now delete the feature class vegtype from your Habitat_Analysis geodatabase, as it is no longer needed.

7. In the ArcCatalog tree, right-click vegtype inside the Habitat_Analysis geodatabase and click Delete.

8. Right-click vegtype2 and click Rename. Type “vegtype” and press Enter.

You can continue on to Exercise 3 or stop and complete the tutorial at a later time. If you do not move on to Exercise 3 now, do not delete your working copy of the tutorial data or the folder connection that accesses it in ArcCatalog.

Exercise 3: Clipping data to the study area

You can complete many tasks using the system tools as you have learned in Exercise 2; however, the process of running single tools can be time consuming. When you want to perform *batch processing* (running the same tool on multiple inputs), such as when converting multiple datasets to a different format, running the same tool multiple times is not ideal. To automate your work flow so you can, for instance, perform a task on multiple inputs in one step, you can add your own scripts to a toolbox, or execute the tool in batch mode.

Scripts you add to a toolbox work as any system tool. They can be run via a dialog box, at the command line, inside a model, or inside another script.

Any geoprocessing tool can be run in batch mode. When you right-click any tool in ArcCatalog or ArcToolbox you will see Batch on the context menu. When selected, the tool dialog box opens and displays a batch grid that you will fill out.

This exercise will show you both methods for running the same tool on multiple inputs and will take approximately 20 minutes to complete. First, you'll add a previously created script to a toolbox that will clip the tutorial datasets located in your GP_Tutorial folder and place them inside your Habitat_Analysis file geodatabase. Second, you will open the Clip tool in batch mode and clip several of the tutorial datasets located in your GP_Tutorial folder and place them inside your Habitat_Analysis file geodatabase.

Setting up

1. If you don't have ArcCatalog open, start it by using the Programs list in your Start menu.

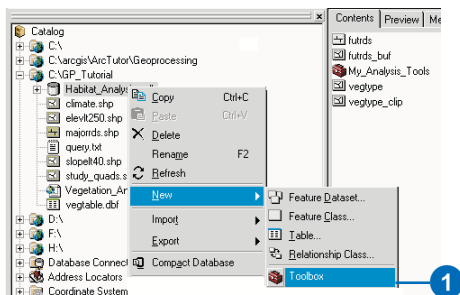
If you did the previous exercise, skip the next step and go to the next section, 'Adding a new toolbox to Habitat_Analysis.gdb'.

2. Copy the GP_Tutorial folder from `arcgis\ArcTutor\Geoprocessing\Results\Ex2` on the drive where you installed ArcGIS to a local drive—for example, your C:\ drive.

Adding a new toolbox to Habitat_Analysis.gdb

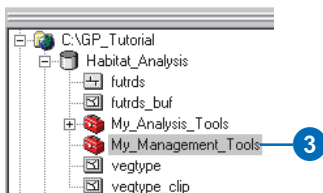
You'll first create a new toolbox to hold the tools you'll create in this exercise.

1. Right-click Habitat_Analysis.gdb in your GP_Tutorial folder, point to New, and click Toolbox.



A new toolbox is created inside the geodatabase.

2. Expand your Habitat_Analysis geodatabase to see the created toolbox in the ArcCatalog tree.
3. Right-click the toolbox and click Rename. Type "My_Management_Tools" and press Enter.

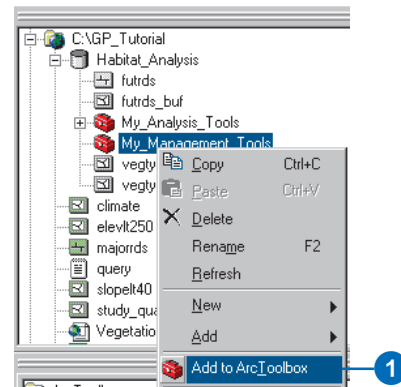


4. Right-click the toolbox and click Properties. Change the label to My_Management_Tools. Click OK.

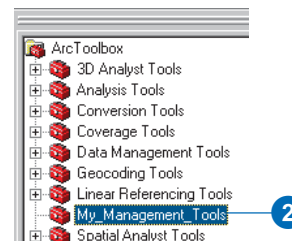
Creating a shortcut to the toolbox

You'll create a shortcut to your My_Management_Tools toolbox in the ArcToolbox window. The toolbox shortcut will point to your My_Management_Tools toolbox stored inside your Habitat_Analysis geodatabase on disk.

1. Right-click your My_Management_Tools toolbox and click Add to ArcToolbox.

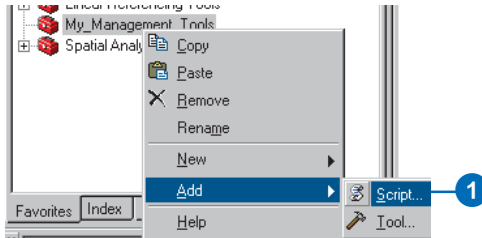


2. If the ArcToolbox window is not displayed, click the Show/Hide ArcToolbox Window button on the Standard toolbar, then collapse all the toolboxes inside the ArcToolbox window so you can see your My_Management_Tools toolbox.



Adding a script to your toolbox

1. Right-click your My_Management_Tools toolbox in the ArcToolbox window, point to Add, and click Script.



2. Type “Multi_Clip” for the name of the script and type “Clip Feature Classes” for the label.
3. Optionally, add a description to explain what the script will do.

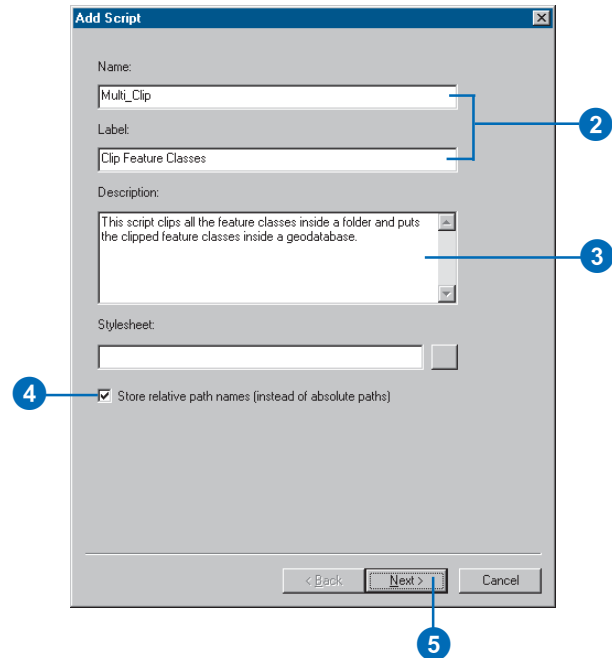
The script’s name can be used when running the script inside another script or at the command line. The label is used as the script’s display name in the user interface.

The default stylesheet will be applied to the dialog box of the script unless specified otherwise. Here you’ll use the default, so leave the Stylesheet text box blank.

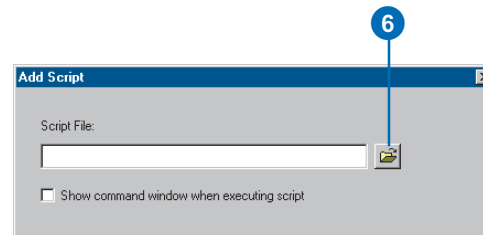
4. Check Store *relative path* names.

All pathnames referenced by the script will be stored relative to the location of the toolbox containing the script. Storing relative paths avoids the hassle of repairing pathnames to data sources if the toolbox and associated data sources are moved to a new location. Provided the same directory structure is used at the new location, the tools contained within the toolbox will still be able to find their data sources by traversing the relative paths.

5. Click Next.



6. Click the Browse button to the right of the Script File text box and navigate to your GP_Tutorial folder.



- Click multi_clip.py and click Open to add the script.

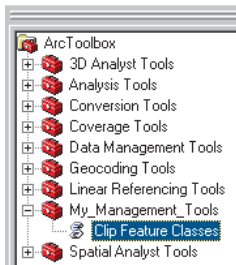
A *script* is a simple text file that can be created in any text editor. It holds a loosely typed (meaning variable declaration is not required) scripting language, such as JScript, VBScript, or Python. Refer to *Writing Geoprocessing Scripts With ArcGIS* to learn more about writing geoprocessing scripts.

- Click Next.

Leave the parameters and their properties as they are at this point.

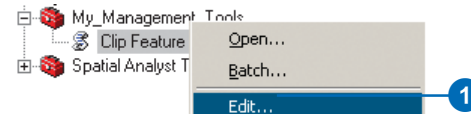
- Click Finish.

The script Clip Feature Classes has been added to your My_Management_Tools toolbox.



Editing a script

- Right-click the script in the ArcToolbox window and click Edit.



This opens the Python script in the PythonWin application so you can view or modify the contents of the script.

- Examine the code. Notice that the values set for the variables gp.workspace, clipFeatures, out_workspace, and XYTolerance are set as system arguments (sys.argv []). Four parameters need to be defined for these system arguments in the Properties dialog box of the script so the parameters will display on the script's dialog box.

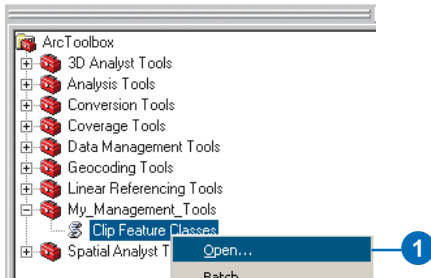
For more information on the contents of this particular script, see 'Creating a New Script Module' in Chapter 2 of *Writing Geoprocessing Scripts With ArcGIS*.

Note: Instead of using system arguments for the values of variables, you could hard code the value. However, hard coding the value means that only the value specified in the script can be used—for example, the path to a workspace containing input data.

- Close the PythonWin application.

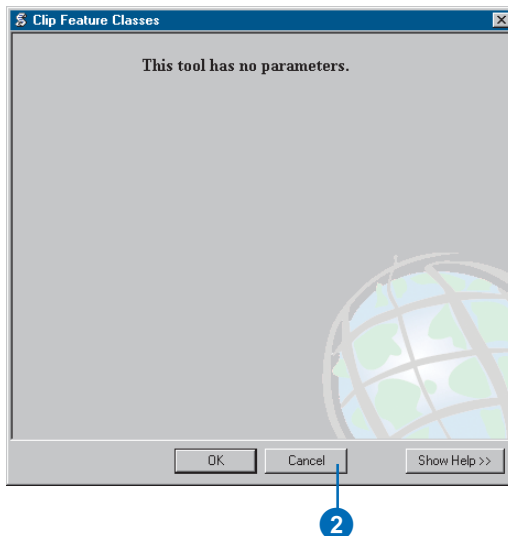
Setting script parameters

1. Right-click your Clip Feature Classes script in the ArcToolbox window and click Open to open its dialog box.

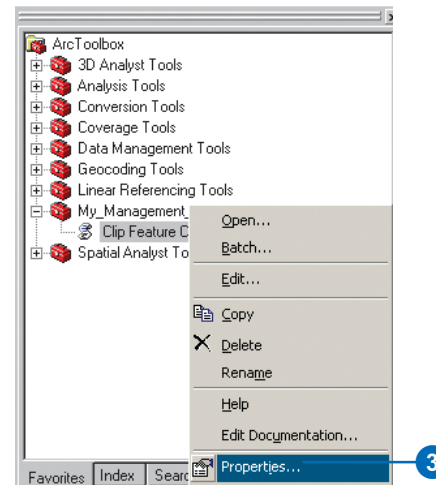


Notice that there are no parameters on the dialog box. This is because they have not been defined for the four system arguments (`sys.argv []`) you saw in the code.

2. Click Cancel.



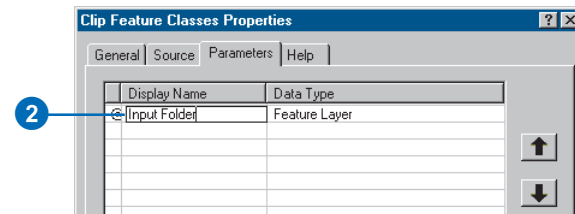
3. Right-click the Clip Feature Classes script and click Properties.



4. Click the Parameters tab.

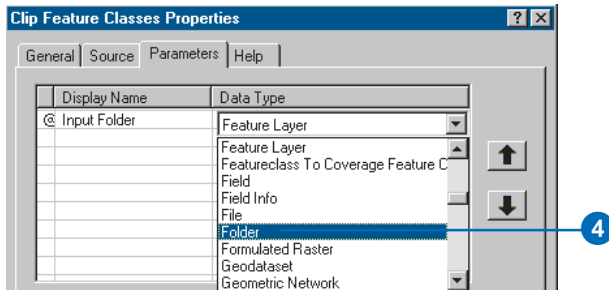
Setting the Input Folder parameter

1. In the Parameters section, click in the first row under the column heading Display Name.
2. Type "Input Folder" for the name of the parameter.



You must specify the type of data a parameter can store.

3. Click the same row under the column heading Data Type to specify the *data type* for the parameter.
4. Scroll through the list of data types and click Folder.



5. Examine the property values set for the Input Folder parameter in the Parameter Properties section of the dialog box.

The values that could be set for the property Type are either Required or Optional. As it is required to set an input folder, leave the default value, Required.

The values that can be set for the property Direction are either Input or Output. As this parameter is an input parameter—that is, the folder contains the input feature classes that will be clipped—leave the default value, Input.

If the input parameter accepted multiple values, such as multiple folders, you would set the MultiValue property value to Yes. However, here you can only process one folder at a time, so leave the default value, No.

You can specify a default value that will display for the parameter on the tool's dialog box. For the input folder, you want the default value to be the path to your GP_Tutorial folder.

6. Type “C:\GP_Tutorial” for the value of the property Default if your GP_Tutorial folder is on your C:\ drive. Alternatively, type the path to your GP_Tutorial folder.

This value will be displayed and used by this parameter by default.

Don't set a value for the Environment property; instead, accept the default. This property will be explained when the XY Tolerance parameter property values are defined.

Attribute domains are used to constrain the values allowed for the parameter. There are two different types of attribute domains: range domains or coded value domains. A range domain specifies a valid range of values, and a coded value domain specifies a valid set of values for a parameter. For this parameter, an attribute domain is not necessary.

Some parameters are dependent on the value of other parameters for information, such as a parameter that lists the fields in an input table. The fields can only be displayed once the input data parameter has a value. The input folder is not dependent on any other parameter, so you'll use the default.

The values for the properties of the parameter Input Folder should now be set up as shown in the graphic that follows.

Display Name	Data Type
@ Input Folder	Folder
	Feature Layer

Click any parameter above to see its properties below.

Parameter Properties

Property	Value
Type	Required
Direction	Input
MultiValue	No
Default	C:\GP_Tutorial
Environment	
Domain	
Dependency	

Setting the Input Clip Feature Class parameter

1. Click in the second row under the column heading Display Name.
2. Type “Input Clip Feature Class”.
3. Click the same row under the column heading Data Type to specify the data type for the parameter.
4. Scroll up through the list of data types and click Feature Class.
5. Leave the defaults for the values of the properties Type, Direction, and MultiValue, then type the location for GP_Tutorial\study_quads.shp, such as “C:\GP_Tutorial\study_quads.shp”, for the value of the property Default. This shapefile will be displayed and used as the value for the parameter by default.

Your Properties dialog box should now resemble the graphic that follows.

Display Name	Data Type
Input Folder	Folder
@ Input Clip Feature Class	Feature Class
	Feature Layer

Click any parameter above to see its properties below.

Parameter Properties

Property	Value
Type	Required
Direction	Input
MultiValue	No
Default	C:\GP_Tutorial\study_quads.shp
Environment	
Domain	

Setting the Output Workspace parameter

1. Click in the third row under the column heading Display Name.
2. Type “Output Workspace”.
3. Click the same row under the column heading Data Type to specify the data type for the parameter.
4. Scroll through the list of data types and click Workspace.
5. As this parameter is an input parameter (an existing workspace into which the outputs will be placed), accept the default value of input for the property Direction.
6. For the value of the property Default, type the location for GP_Tutorial\Habitat_Analysis.gdb, such as “C:\GP_Tutorial\Habitat_Analysis.gdb”. This personal geodatabase will be displayed and used as the value for the parameter by default. All clipped outputs will, by default, be placed in this location.

The values set for the properties of the parameter Output Workspace should resemble those in the graphic that follows.

Display Name	Data Type
Input Folder	Folder
Input Clip Feature Cl...	Feature Class
Output Workspace	Workspace
	Feature Layer

Click any parameter above to see its properties below.

Property	Value
Type	Required
Direction	Input
MultiValue	No
Default	C:\GP_Tutorial\Habitat_Analysis.gdb
Environment	
Domain	
Dependency	

Setting the XY Tolerance parameter

1. Click in the fourth row under the column heading Display Name.
2. Type "XY Tolerance".
3. Click the same row under the column heading Data Type to specify the data type for the parameter.
4. Scroll through the list of data types and click Linear unit.
5. Click the row for the property Type in the Value column to open the dropdown list of values. Click Optional. It is not required to specify a value for the XY Tolerance parameter. A default value will be used if there is no value set.

Leave the defaults for the properties Direction, MultiValue, and Default. You could type a value to use as the default. Instead, you'll take the value set for XY Tolerance from the Environment Settings dialog box to mimic the behavior of the system tools.

6. Click the row for the property Environment in the Value column to open the dropdown list of environment setting values. Click XY Tolerance. The value set for the XY Tolerance in the Environment Settings dialog box will be displayed as the value for the parameter by default. If the value set in the Environment Settings dialog box is changed, the value specified there will be used by the XY Tolerance parameter.

Leave the default for the Domain property. If you wanted to limit the range of values that could be set for the parameter, you could set up a domain. The XY Tolerance depends on the input data, so you won't set a domain for this parameter.

The values set for the properties of the XY Tolerance parameter should resemble those in the graphic that follows.

Display Name	Data Type
Input Folder	Folder
Input Clip Feature Class	Feature Class
Output Workspace	Workspace
@ XY Tolerance	Linear unit Feature Layer

Click any parameter above to see its properties below.

Property	Value
Type	Optional
Direction	Input
MultiValue	No
Default	
Environment	XY Tolerance [xy_tolerance]
Domain	
Dependency	

7. Click OK.

Running the script

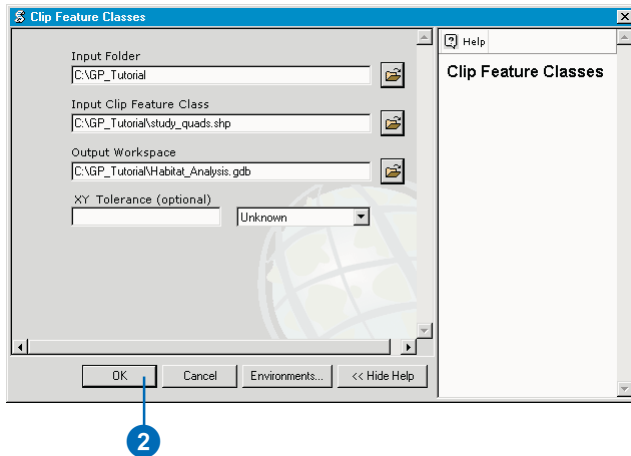
1. Double-click the Clip Feature Classes script in the ArcToolbox window to open it. The parameters you defined are displayed on its dialog box.

Notice how each parameter contains a default value. For the Input Folder, Input Clip Feature Class, and Output Workspace parameters, these are the values you set up for the Default property of each parameter.

The *XY Tolerance* is the distance that determines the range in which features are made coincident. You'll leave the default (blank) and the units Unknown for the XY Tolerance parameter. You set the value for the XY Tolerance parameter to be taken from the Environment Settings dialog box when you set up the properties for this parameter. As you haven't specified a value for this parameter in the Environment Settings dialog box, a default value will be calculated. The default value calculated will ensure the precision of the data, as it always integrates the data at the smallest possible XY Tolerance value.

The units for the XY Tolerance parameter are set to Unknown. The units given to the spatial reference of the output feature classes will be used by the XY Tolerance parameter. If you were to set the units to be anything other than Unknown, they would be converted to the same units as the output. The output feature classes get their units from the spatial reference set for the first input, that is, the first shapefile in the GP_Tutorial folder to be clipped, provided you haven't specified otherwise in the Spatial Reference section of the Environment Settings dialog box.

2. Click OK.

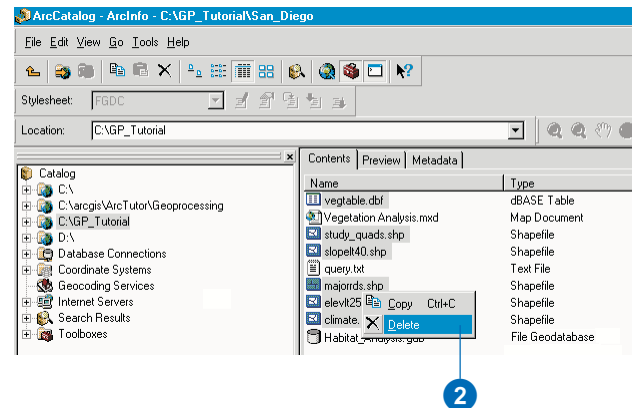


The clipped feature classes are added to the Habitat_Analysis geodatabase. You will use these feature classes in the next exercise to locate potential high-quality habitat areas for the California gnatcatcher.

Deleting unnecessary datasets

You can now delete the shapefiles and the DBF table you copied locally to your GP_Tutorial folder as they are not needed for the rest of this tutorial.

1. In the ArcCatalog tree, click the connection to the tutorial data, which is your GP_Tutorial folder. In the Contents tab, click each shapefile and the DBF file; press Ctrl to select more than one item.
2. Right-click the selection and click Delete.



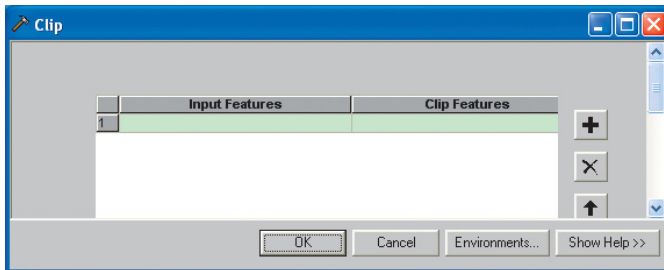
Running the Clip tool in batch mode

For geoprocessing, batch means to execute a single tool multiple times with different inputs without your intervention. In this portion of the tutorial you will run the Clip tool once for several of the input datasets located in your GP_Tutorial folder and place them inside your Habitat_Analysis file geodatabase.

Opening the Clip tool in batch mode

1. In the ArcToolbox window, double-click the Analysis Tools, double-click the Extract toolset, then right-click the Clip tool and click Batch.

The batch window will open, where you can set up multiple runs for the tool. When you open the batch window for a tool, one row is automatically added, ready to accept input.



A required parameter must contain a value for the tool to execute successfully. Required parameters are represented as green cells in the batch grid. In the batch grid, color and icons give you the status of the parameters.

Cell color	Corresponding icon in tool dialog	Color	Meaning
		Green	Required parameter.
e:\schools.shp		Red	Error: process will not execute.
E:\Data\DocDat		Yellow	Warning: output exists but results may not be what you expect.
	(none)	Gray	Unavailable: parameter has no use given values of other parameters.
	(none)	White	Parameter is correct.
	(none)	Blue	Selected row.

Creating a batch process

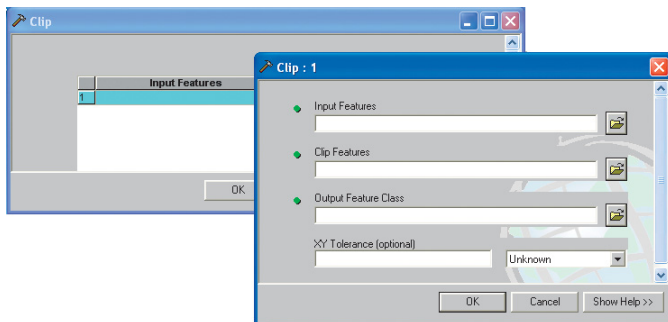
First, resize the window to show as many columns as possible in the batch grid. This is because most tools have more parameters than the number of columns visible in the window.

1. Click the outside edge of the window and drag to resize.

You can maximize the window using the controls in the upper right of the window. You can also change the width of individual columns by clicking and dragging the column separators.

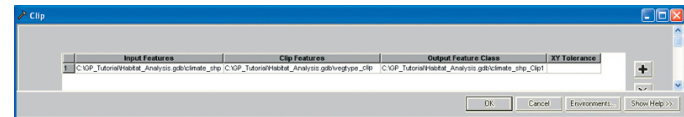
In the batch grid, each process (a tool plus its data) is represented by a row. Within each row, each cell represents one of the tool's parameters.

2. Double-click the row header (the column containing the numbers) to open the tool dialog containing all parameters.
3. Click Browse and navigate to your Habitat_Analysis geodatabase.



4. Double-click the Habitat_Analysis geodatabase and select Climate_shp.
5. Click Add.
You will notice that the Output feature Class parameter is automatically filled, the same as when you run the tool dialog in single mode.
6. Click Browse and click the vegtype_clip feature class for the Clip Features parameter.
7. Click Add.
8. Click OK to close the Clip tool dialog box.

The first row of the batch grid shows the values you just selected.

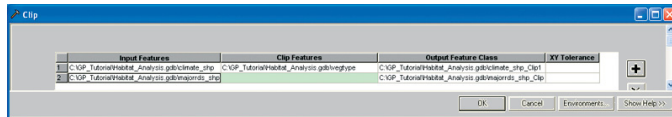


Adding a second process

You will now add a second process (row) to the batch grid.

1. Click the Add button **+**. A new row (process) appears.
2. Right-click the Input Features cell and click Browse.
3. Navigate to the Habitat_Analysis geodatabase.
4. Double-click the Habitat_Analysis geodatabase and select majorrds_shp.
5. Click Add.
6. Click OK to close the dialog box.

You will notice that the Output feature Class parameter is automatically filled, the same as when you run the tool dialog in single mode. The Clip Features parameter remains empty and is green to remind you to enter a value before you execute the tool. Leave it blank.



Adding a third process

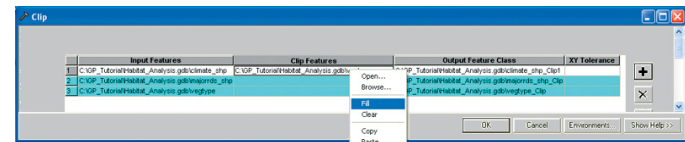
1. Click Add to add another process.
2. Navigate to the Habitat_Analysis geodatabase in ArcCatalog.
3. Double-click the Habitat_Analysis geodatabase and select vegtype.
4. Drag and drop the dataset on the Input Features cell of the third process.

Again, you will notice that the Output feature Class parameter is automatically filled, the same as when you run the tool dialog in single mode. The Clip Features parameter remains empty and is green to remind you to enter a value before you execute the tool. Leave it blank.

Using the Fill option

Since each input dataset will be clipped with the same clip features, you can use one of the batch grid features to automatically fill in the Clip Features column. In instances where there are many rows, Fill is the preferred method.

1. Select the second and third rows.
2. Right-click the Clip Features cell in the first row (the one that contains a value) and click Fill. The value of the clicked cell will fill the empty cell in each selected row.

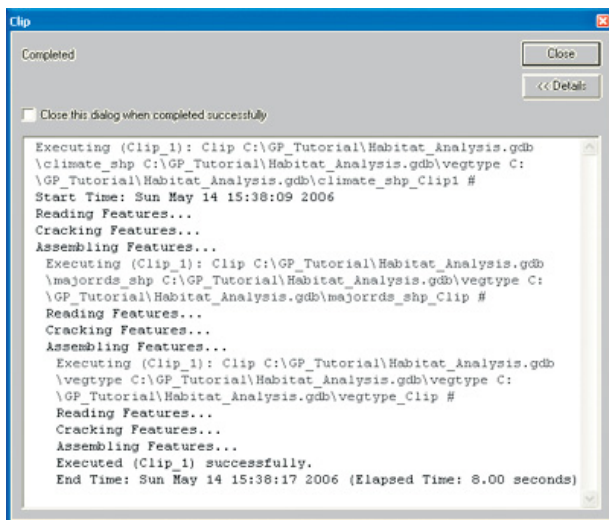


Executing the Clip tool

When you have filled the batch grid with all your processes you can execute the Clip tool. All the cells should appear white (colorless) and there should be no warning or error icons.

1. Click Ok.

The tool will execute once for each row, starting with the first row. The tool's progress dialog box will show the messages for each execution.



Closing ArcCatalog

1. Click the File menu and click Exit to close ArcCatalog.

You can continue on to Exercise 4 or stop and complete the tutorial at a later time. If you do not move on to Exercise 4 now, do not delete your GP_Tutorial folder, and do not remove the folder connection that accesses it.

Exercise 4: Finding high-quality habitat patches

Creating a model provides a way to run a sequence of geoprocessing tools by enabling you to string your tools together in a visual diagram. A model can be run again and again, and you can alter parameter values to experiment with different outcomes.

Your task in this exercise is to locate high-quality habitat patches—areas that have the best potential to support the California gnatcatcher. You will add data from your Habitat_Analysis geodatabase to the table of contents of your ArcMap session. You'll then create a new model to find potential habitat areas, incorporating the following five criteria:

1. Roads form a barrier for the gnatcatcher. The impact of roads on gnatcatcher habitat increases proportionately with the road size.
2. The gnatcatcher prefers to live in areas where the vegetation is of a particular type, most preferably, San Diego coastal sage scrub, though other vegetation types are also suitable.
3. Parent gnatcatchers drive fledglings off their territory once they are ready. Fragmentation of existing habitat has meant young gnatcatchers cannot move very far away. For the species to survive long term, they need sizable patches of suitable vegetation, though home ranges tend to be smaller nearer the coast and larger in the drier, sparser inland areas. Nearer the coast—within the Maritime (zone 1) and Coastal (zone 2) climate zones—vegetation patches greater than and equal to 25 acres (1,089,000 ft²) will be considered suitable. In the inland areas—within the Transitional climate zone

(zone 3)—vegetation patches greater than and equal to 50 acres (2,178,000 ft²) will be considered suitable.

4. Gnatcatcher sightings have been highly correlated with elevation. Areas below 250 m in elevation have contained the majority of all documented sightings in the past. Only areas where the elevation is less than 250 m above sea level will be considered as suitable habitat locations.
5. The gnatcatcher prefers to nest in areas where the slope of the terrain is minimal. Habitat on slopes that are steeper than 40 percent have low potential for nesting. Only areas where the slope of the terrain is less than 40 percent will be considered as suitable habitat locations.

Setting up

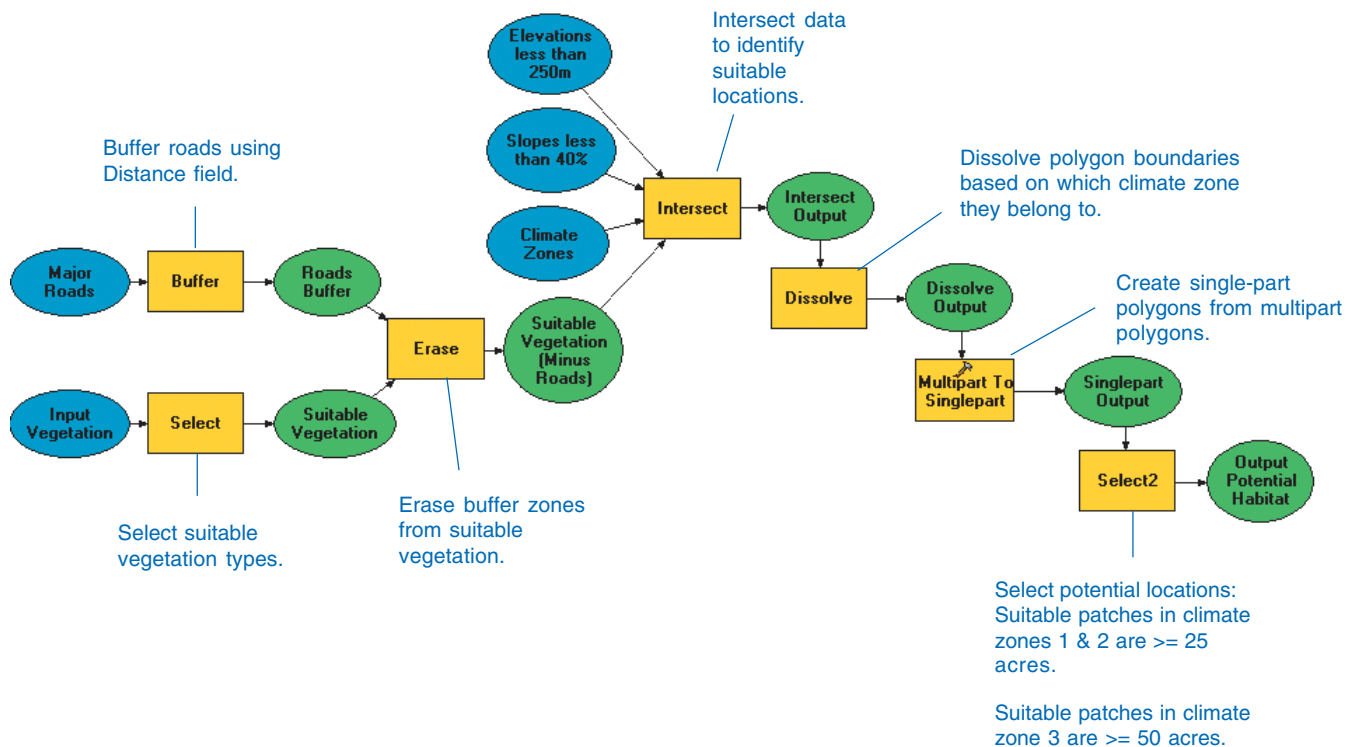
If you did the previous exercise, skip the next step and go to the next section, 'Starting ArcMap and adding data'.

1. Copy the GP_Tutorial folder from C:\arcgis\ArcTutor\Geoprocessing\Results\Ex3 to your C:\ drive or an alternative drive.

This exercise will take approximately 40 minutes to complete.

The following diagram shows the work flow you'll go through in this exercise to identify high-quality habitat patches.

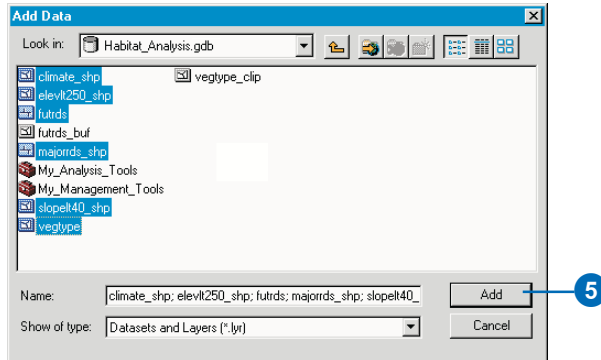
Gnatcatcher Habitat Suitability



Starting ArcMap and adding data

You'll start by opening ArcMap and adding the feature classes you created in the Habitat_Analysis geodatabase.

1. Start ArcMap by using the Programs list in your Start menu.
2. Click OK to open a new, empty map.
3. Click the Add Data button on the Standard toolbar, then double-click the folder connection to your local copy of the tutorial data, for example, C:\GP_Tutorial.
4. Double-click the Habitat_Analysis geodatabase to open it and select all feature classes except vegtype_clip and futrds_buf; use the Ctrl key to select multiple feature classes.
5. Click Add.



The feature classes are added to your ArcMap session as layers.

Opening the ArcToolbox window

When working in ArcMap, you access toolboxes through the ArcToolbox window. By saving the map document you save any changes made to the ArcToolbox window. This allows you to have different versions of the ArcToolbox window with each map document you create, depending on which tools you want to use for each project.

1. Click the Show/Hide ArcToolbox Window button on the Standard toolbar to open the ArcToolbox window if it is not already open.



2. If the ArcToolbox window does not display under the ArcMap table of contents, click and drag the bar at the top of the ArcToolbox window and place it over the table of contents, then drop the panel.

Creating a personal geodatabase to hold intermediate results

The *Command Line window* consists of a command line and a message section. All tools added to the ArcToolbox window can be executed from the command line as well as from their dialog boxes. The *command line* provides a quick way to execute tools if you are familiar with their parameters.

You'll now create a file geodatabase (Temp_Results.gdb) by running a tool at the command line. This file geodatabase will hold intermediate data that you'll create to produce the

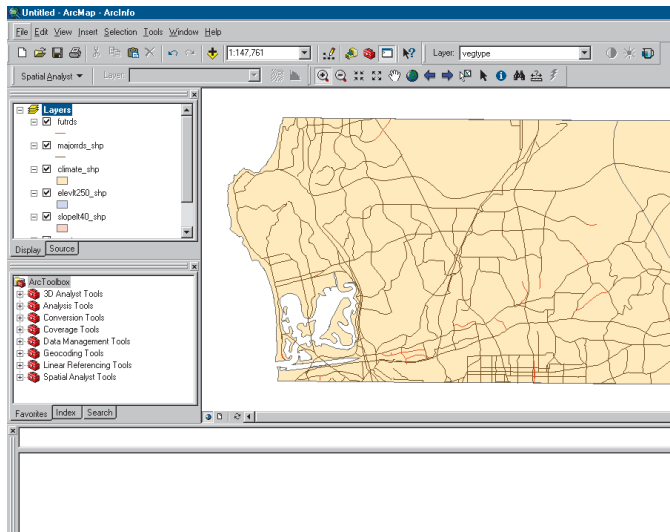
final result. Results you want to keep will be placed inside the existing geodatabase, Habitat_Analysis.gdb.

1. Click Show/Hide Command Line Window on the Standard toolbar to open the window.



As with the ArcToolbox window, the Command Line window can be docked anywhere inside any ArcGIS Desktop application.

2. Press and hold the Ctrl key while you click and drag the bar at the top of the Command Line window and position it at the lower-left corner of your ArcCatalog application. Holding the Ctrl key prevents the window from docking to the application.
3. Release the Ctrl key and drop the panel.

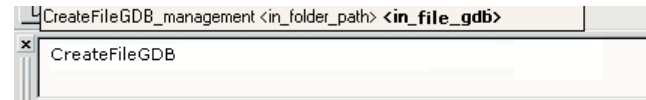


The window will dock at the bottom of your application. If you find that it is not docked in the position shown, click the Command Line title bar to snap the window into position. If the Draw toolbar is present, click View, then Toolbars, and uncheck Draw.

The Command Line window contains a prompt (a command line) where you can run tools and a message window, showing all the messages associated with running tools—at the command line, through a tool’s dialog box, by running a model from within a ModelBuilder window or any messages returned after running a standalone script.

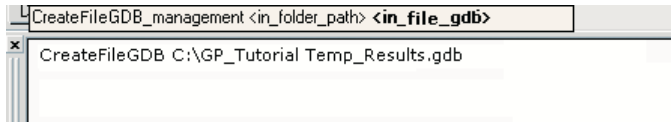
AutoComplete enables you to type part of the tool name until you see the name of the tool displayed in the dropdown list. You can then press the Space key to automatically complete the rest of the tool name. Once the tool name is entered, the tool’s usage appears as a ToolTip. It prompts you for the next parameter value that must be specified by highlighting the parameter in bold.

4. Type “CreateFileGDB”, then press the Space key.



Note: The Data Management Tools toolbox must be added to the ArcToolbox window to run tools from this toolbox at the command line. If the toolbox is not added, right-click the ArcToolbox window and click Add Toolbox. Navigate to the Toolboxes folder at the root level of the ArcCatalog tree and locate the Data Management Tools toolbox, then add it to the ArcToolbox window.

5. Type “C:\GP_Tutorial”, or your alternative drive, for the value of the <in_folder_path> parameter, then type a space.
6. Type “Temp_Results.gdb” for the value of the <in_file_gdb> parameter. This is the file geodatabase to create inside your GP_Tutorial folder.



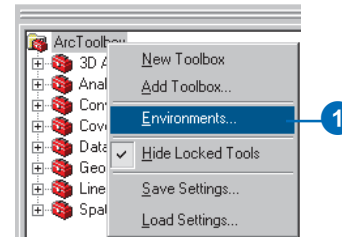
7. Press Enter to run the CreateFileGDB tool.

A new personal geodatabase called Temp_Results.gdb is placed in your GP_Tutorial folder.

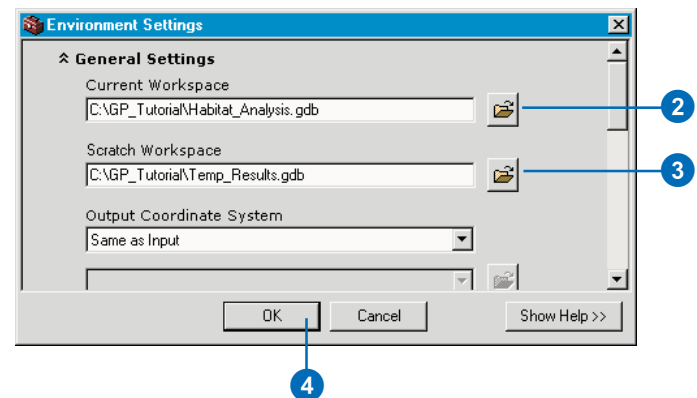
Setting workspaces

You'll set the current workspace to point to your Habitat_Analysis file geodatabase. You'll also set up a *scratch workspace* to point to your Temp_Results file geodatabase. When building a model it is useful to set up your workspaces in this manner. In each tool's dialog box, you can simply leave the default path provided (to your scratch workspace) for most outputs. For those outputs you want to keep separate, such as the final result from running the model, you can blank out the default path and type a name for the output from the tool. Such outputs will be placed in the location set for your current workspace. You'll see this as you build the model in this exercise.

1. Right-click the ArcToolbox folder and click Environments.



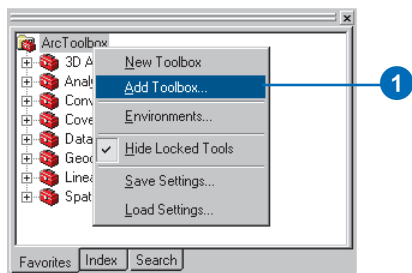
2. Click the General Settings dropdown arrow to expand its contents. For the value of the Current Workspace environment setting, type “C:\GP_Tutorial\Habitat_Analysis.gdb”, or type your alternative location. Alternatively, click the Browse button to navigate to this location.
3. For the value of the Scratch Workspace environment setting, type “C:\GP_Tutorial\Temp_Results.gdb”, or type your alternative location. Alternatively, click the Browse button to navigate to this location.
4. Click OK.



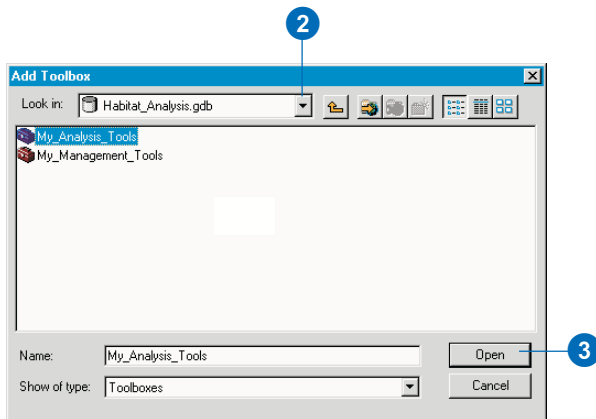
Adding the My_Analysis_Tools toolbox

You'll first add your My_Analysis_Tools toolbox to the ArcToolbox window within your ArcMap session.

1. Right-click the ArcToolbox window and click Add Toolbox.



2. Click the Look in dropdown arrow and navigate to Habitat_Analysis.gdb in your GP_Tutorial folder.
3. Click My_Analysis_Tools and click Open.



The toolbox is added to the ArcToolbox window.

Saving the map document

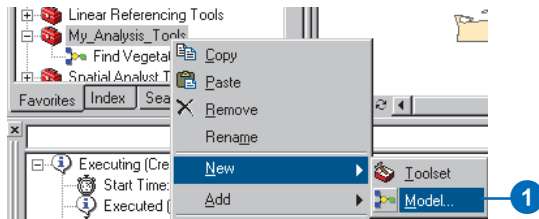
Geoprocessing settings include environment settings, the state of the ArcToolbox window, and variables you have created at the command line. These settings are saved in ArcMap when you save a map document. There are times when you want to switch between applications, have different settings for different projects, or use the default settings. In these cases you can save the geoprocessing settings by right-clicking the ArcToolbox window and clicking Save Settings. The default settings or saved settings in a file can be loaded at any time by right-clicking the ArcToolbox window and clicking Load Settings.

1. Click File on the Main menu and click Save As.
2. Click the Save in dropdown arrow and navigate to your GP_Tutorial folder.
3. Type "Habitat_Analysis.mxd" for the filename and click Save.

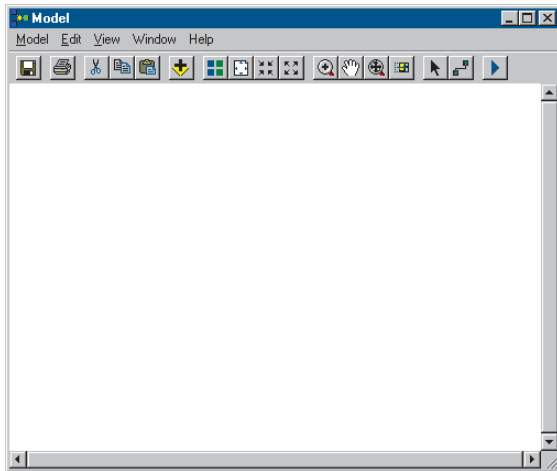
Creating a new model

You will now create a model by building processes and connecting them together to find areas with the best potential to support the gnatcatcher.

1. Right-click your My_Analysis_Tools toolbox in the ArcToolbox window, point to New, and click Model.



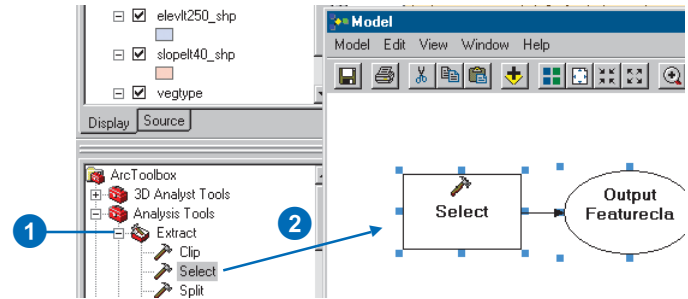
An empty ModelBuilder window will open.



Selecting suitable vegetation types

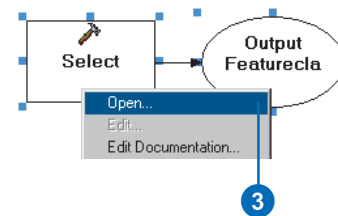
You'll select from the vegetation feature class (vegtype) those vegetation types preferred by the gnatcatcher. You'll use the Select tool to select all suitable vegetation types (polygons with a Habitat value of 1).

1. Expand the Extract toolset located inside the Analysis Tools toolbox.
2. Click and drag the Select tool into the display window.



An element that references the Select tool is created in the display window.

3. Right-click the Select tool element and click Open.



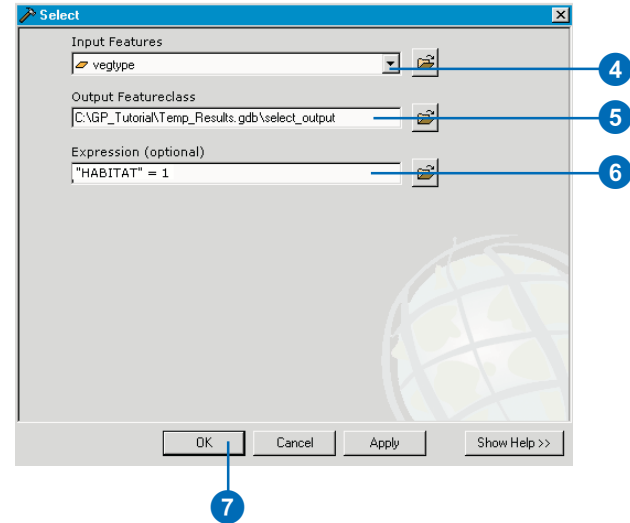
- Click the Input Features dropdown arrow and click the layer vegtype.

Notice that layers from the ArcMap table of contents have a yellow layer icon to the left of them to identify them as layers.

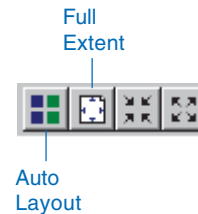
- Since the Scratch Workspace was set earlier, the default path given for the Output Feature Class parameter is set to this location. Leave the default location and type “select_output” for the name.
- Type “HABITAT” = 1” for the value of the Expression parameter, or click the Browse button next to the Expression parameter and use the Query Builder to build the expression.

When all parameters have valid values, the green circles to the left of each parameter will be removed, indicating that a valid value has been set for each parameter. If you have entered an invalid value for a parameter, a red cross icon will display to the left of the parameter. By placing the cursor over such icons, you can obtain information to help solve the problem with the current parameter value that is set.

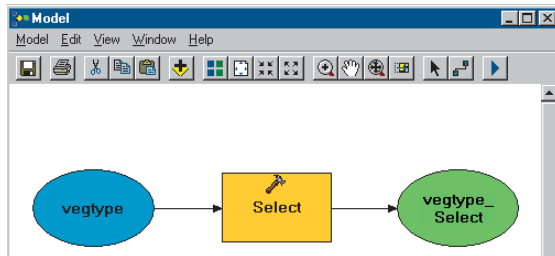
- Click OK.



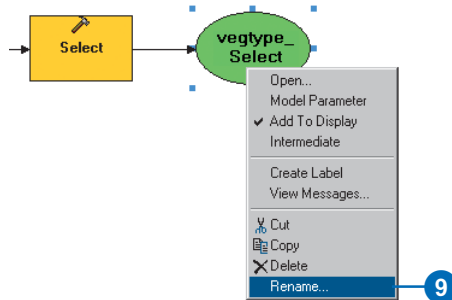
- Click Auto Layout then click Full Extent to apply the current diagram properties to the elements and to place the elements within the display window.



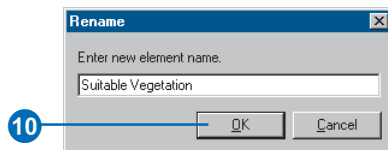
Notice that the process is now colored in, meaning it is ready to run.



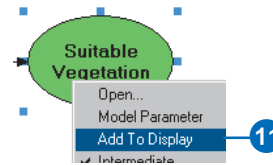
9. Right-click the vegtype_Select derived data element and click Rename.



10. Type “Suitable Vegetation” and click OK.

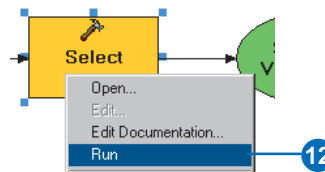


11. Right-click the Suitable Vegetation data element and click Add To Display.



With the Add To Display property checked, the derived data referenced by this element will be added to the display each time the model is run.

12. Right-click the Select tool element and click Run to run the process.



Notice that as the process runs, its progress is documented in the message section of the Command Line window, and the tool element that references the tool is highlighted in red. When the process has finished, the tool and its derived data element become shaded, indicating that the process has run and the derived data has been created on disk.

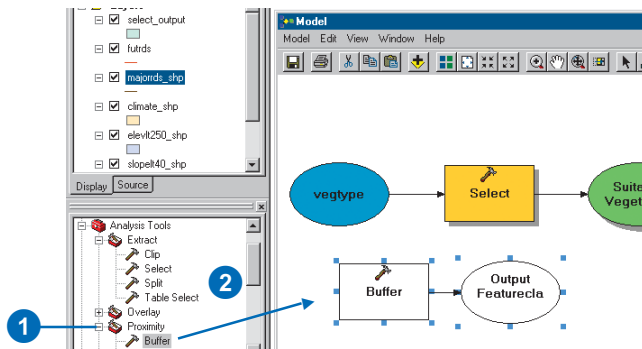
13. Check Close this dialog when completed processing if the Select progress dialog box is present, then click Close.
14. Examine the layer select_output in your ArcMap display. All vegetation types that are preferable for the gnatcatcher, including areas of San Diego coastal sage scrub, have been selected from the vegetation data.

Excluding the roads

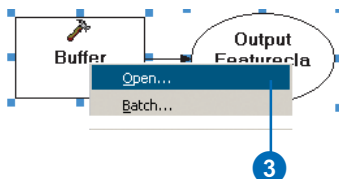
You'll now exclude the areas near roads and freeways as potential habitat locations because roads form a barrier for the gnatcatcher, and there are impacts on the habitat patches near roads. You'll buffer the roads using the Distance field in the majorrds_shp attribute table, then erase the resultant buffer zones from the suitable vegetation patches.

Buffering the roads

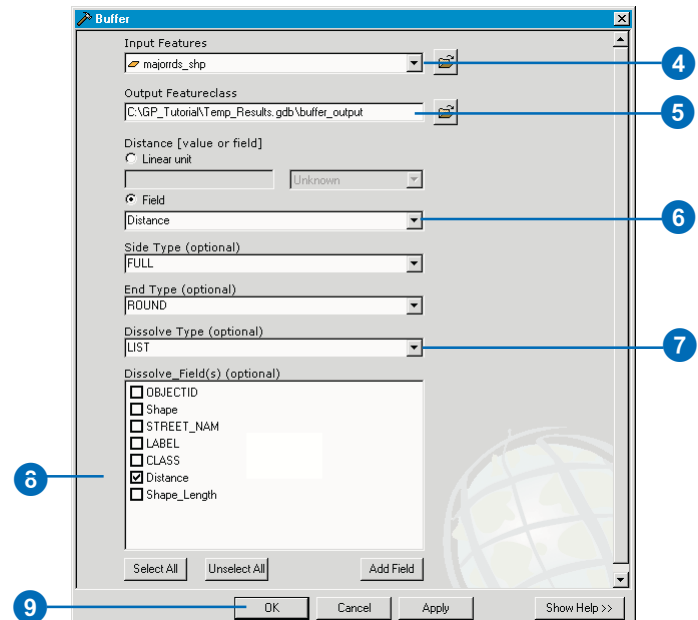
1. Expand the Proximity toolset located inside the Analysis Tools toolbox.
2. Click and drag the Buffer tool into the display window.



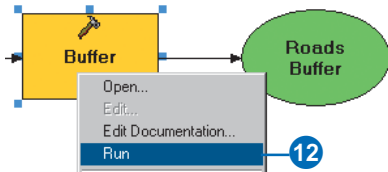
3. Right-click the Buffer tool element that represents the Buffer tool in the model and click Open.



4. Click the Input Features dropdown arrow and click the layer majorrds_shp.
5. Leave the default location for the Output Feature Class parameter and type "buffer_output" for the name.
6. Click Field, click the Field dropdown arrow, and click Distance. The values in this field will be used for the width of the buffer.
7. Click the Dissolve Type dropdown arrow and click LIST to list the dissolve fields.
8. Check Distance in the Dissolve_Field(s) list. All polygons that share the same distance value will be dissolved.
9. Leave the defaults for the rest of the parameter values and click OK.



10. Click Auto Layout then click Full Extent. Right-click the buffer_output derived data element and click Rename. Type “Roads Buffer” and click OK.
11. Right-click Roads Buffer and click Add To Display.
12. Right-click the Buffer tool element and click Run to run the process.

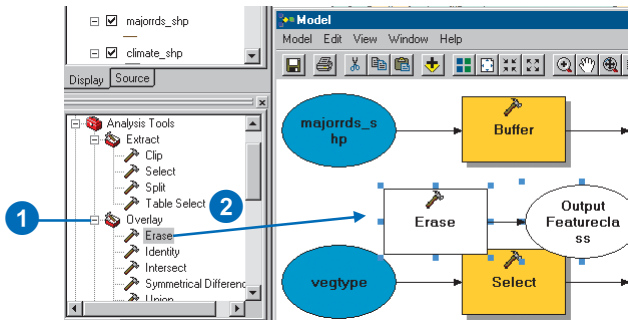


13. Examine the layer buffer_output in your ArcMap display.

Erasing the roads

You'll now erase the buffer zones from the suitable vegetation patches to exclude these areas from the analysis.

1. Expand the Overlay toolset located inside the Analysis Tools toolbox.
2. Click and drag the Erase tool into the display window.



3. Right-click the Erase tool in the model and click Open.

4. Click the Input Features dropdown arrow.

Notice that you now have four variables at the top of the list of inputs as well as all the layers in the table of contents to choose from.

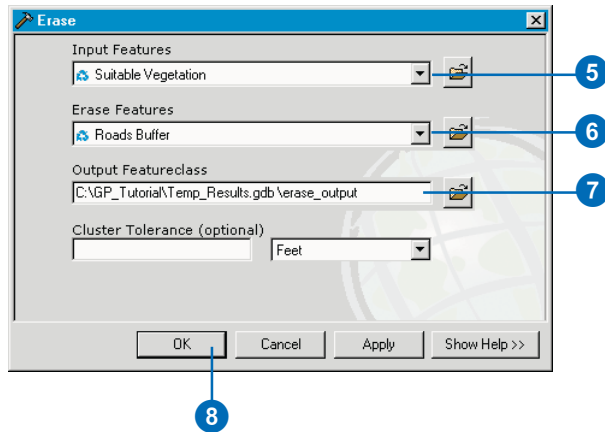
When you drag data into a ModelBuilder window or when you set the data referenced by input or derived data elements within a tool's dialog box, the elements created in the ModelBuilder window are variables that can be shared between processes.

5. Click the variable Suitable Vegetation from the dropdown list. Note the blue icon indicating that it is a variable.
6. Click the Erase Features dropdown arrow and click the variable Roads Buffer.
7. Leave the default location for the value of the Output Feature Class parameter and type “erase_output” for the name.

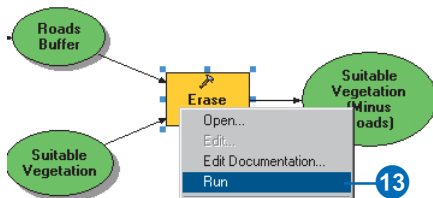
Leave the default value (blank) for the XY Tolerance parameter and leave the units as Feet. An appropriate default value will be calculated using the units of the spatial reference set for the output—in this case, in feet, as the input spatial reference units are in feet.

All lines within the default XY Tolerance calculated will be considered coincident.

8. Click OK.

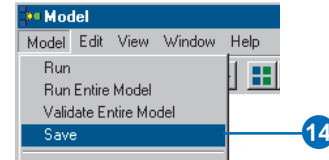


9. Click Auto Layout then Full Extent to display all model elements in their default position within the display window.
10. Right-click the erase_output derived data element and click Rename. Type “Suitable Vegetation (Minus Roads)” and click OK.
11. Click the element Suitable Vegetation (Minus Roads) to select it, then click the blue handles and drag them to resize the element so all the text is displayed.
12. Right-click the Suitable Vegetation (Minus Roads) derived data element and click Add To Display.
13. Right-click the Erase tool and click Run.



It is good practice to save your changes to the model as you are building it.

14. Click the Model menu and click Save.



The model is saved with its default name. You’ll rename the model later.

15. Examine the layer erase_output in your ArcMap display. It shows all vegetation patches that are away from existing roads and are suitable for habitation by the gnatcatcher.

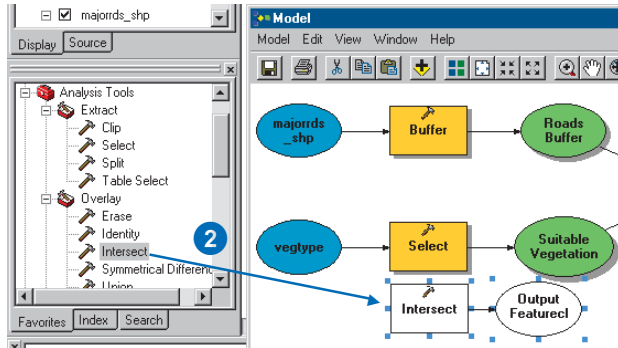
Selecting potential habitat locations

You’ll now use four tools to find potential habitat locations, incorporating the data referenced by the Suitable Vegetation (Minus Roads) element you have derived. You want to find locations where the terrain is less than 40 percent in slope and less than 250 m in elevation as well as where the area of the suitable locations is greater than or equal to 25 acres in climate zones 1 and 2 and greater than or equal to 50 acres in climate zone 3.

Intersecting feature classes

You’ll first intersect `slopetlt40_shp`, `elevlt250_shp`, `climate_shp`, and Suitable Vegetation (Minus Roads) to derive a dataset of suitable patches.

1. Expand the Overlay toolset located inside the Analysis Tools toolbox.
2. Click and drag the Intersect tool into the display window.



3. Right-click the Intersect tool and click Open.
4. Click the Input Features dropdown arrow and click the layer climate_shp. The layer is added to the features list.
5. Click the Input Features dropdown arrow again and click the layer slopel40_shp.
6. Click the Input Features dropdown arrow again and click the layer elevlt250_shp.
7. Click the Input Features dropdown arrow again and click the variable Suitable Vegetation (Minus Roads).

You'll use the default and will not assign any ranks to the input feature classes. The default takes an average if the distance between feature classes is less than the XY Tolerance. You would assign ranks if you knew that some of your feature classes were more integrally sound than others. Features in the feature classes with lower ranks will snap to features in the feature classes with higher ranks; the highest rank is 1, and lower ranks go up in value.

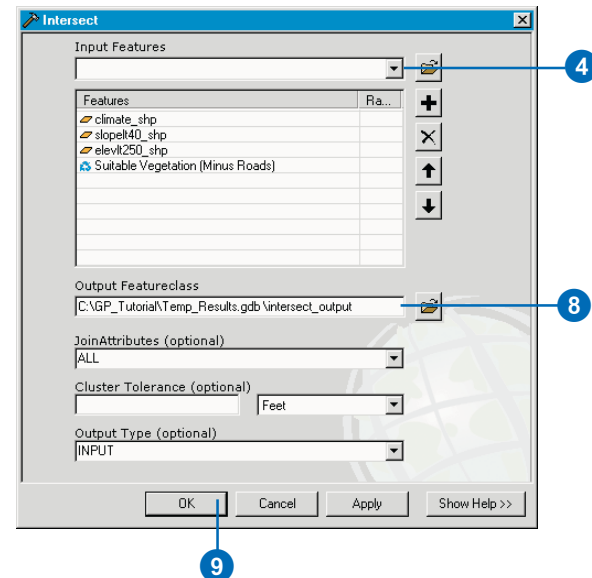
8. Leave the default location for the parameter Output Feature Class and type "intersect_output" for the name. Leave the default to join all attributes.

Leave the default for the value of the XY Tolerance parameter and the units as Feet. The default (blank) means that an appropriate default value will be calculated, using the units of the spatial reference set for the output, in this case, in feet, as the input spatial reference units are in feet.

All lines within the default XY Tolerance will be considered coincident.

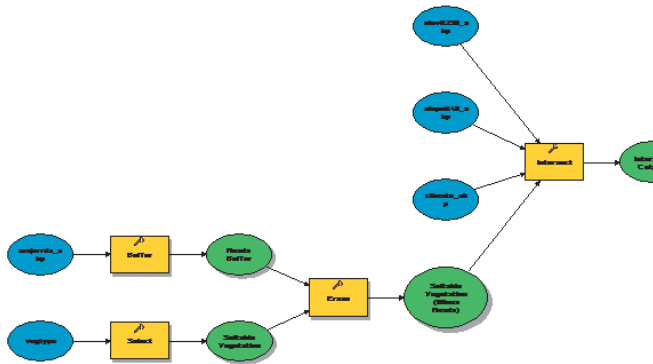
Leave the default for the Output Type so the geometry of the output feature class will be the same as the input with the lowest geometry type (polygon in this case).

9. Click OK.



10. Click Auto Layout, then Full Extent to display all the model elements in their default position in the display window.
11. Right-click the intersect_output derived data element and click Rename. Type “Intersect Output” and click OK.

Your model should now resemble the following graphic.



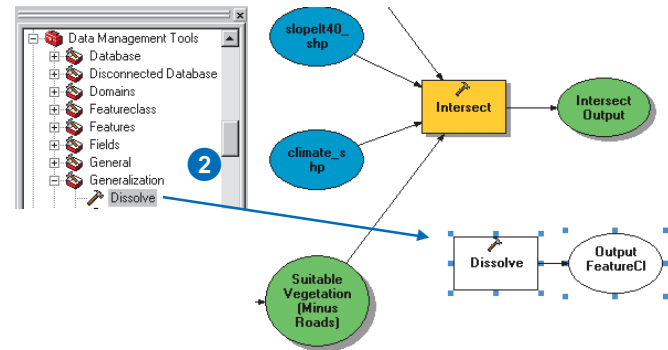
12. Right-click the Intersect Output derived data element and click Add To Display.
13. Right-click the Intersect tool and click Run.
14. Examine the layer intersect_output in your ArcMap display.
15. Click the Model menu and click Save.

Dissolving polygons

You'll now dissolve the polygons of intersect_output to remove unnecessary boundaries between polygons. In the final tool you'll select polygons based on the climate zone they belong to. Zones 1 and 2 both have a CLIMATE_ID

of 2 as they are both close to the coast, and zone 3 has a CLIMATE_ID of 3, representing areas farther inland. You'll dissolve all polygons with the same CLIMATE_ID.

1. Expand the Generalization toolset located inside the Data Management Tools toolbox.
2. Click and drag the Dissolve tool into the ModelBuilder window.

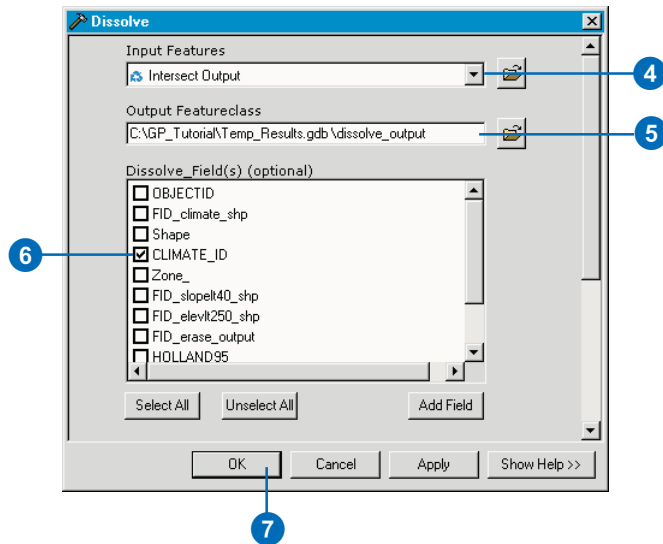


3. Right-click the Dissolve tool and click Open.

4. Click the Input Features dropdown arrow and click the variable Intersect Output.
5. Leave the default location for the Output Feature Class parameter and type “dissolve_output” for the name.
6. Check CLIMATE_ID in the Dissolve_Field(s) list to use CLIMATE_ID as the dissolve field.

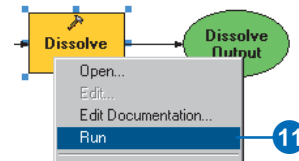
The Dissolve tool gives you the option to create a statistics field, but you don’t need to create one at this time, so leave the Statistics Field(s) parameter blank.

7. Click OK.



8. Click Auto Layout, then Full Extent to display all the model elements in their default position in the display window.
9. Right-click the dissolve_output derived data element and click Rename. Type “Dissolve Output” and click OK.

10. Right-click the Dissolve Output derived data element and click Add To Display.
11. Right-click the Dissolve tool and click Run.

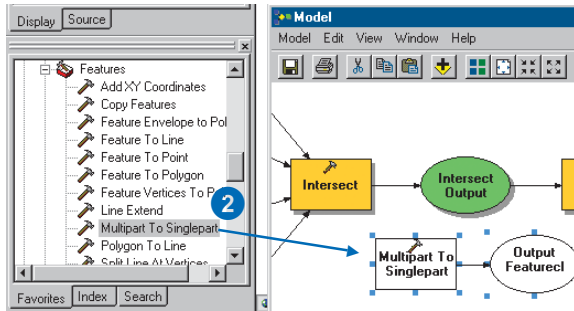


12. Examine the layer dissolve_output in your ArcMap display.
13. Right-click this layer in the table of contents of ArcMap and click Open Attribute Table. There are now only two rows in the table. The polygons have been dissolved into two groups, those with a CLIMATE_ID of 2—climate zones 1 and 2—and those with a CLIMATE_ID of 3, which includes climate zone 3.
14. Close the table.
15. Click the Model menu and click Save.

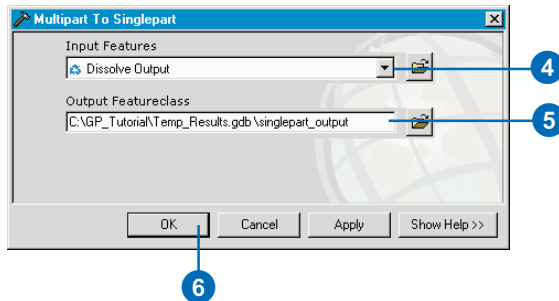
Creating single-part polygons

The Dissolve tool creates multipart polygons—separate polygons sharing the same attributes. Before selecting the required information using the Select tool, you’ll first convert the multipart polygons to single-part polygons in order to extract information about each polygon.

1. Expand the Features toolset located inside the Data Management Tools toolbox.
2. Click and drag the Multipart To Singlepart tool into the ModelBuilder window.



3. Right-click the Multipart To Singlepart tool and click Open.
4. Click the Input Features dropdown arrow and click the variable Dissolve Output.
5. Leave the default for the location of the Output Feature Class parameter and type “singlepart_output” for the name.
6. Click OK.



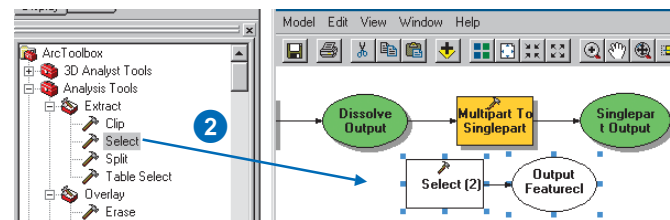
7. Click Auto Layout, then Full Extent to display all the model elements in their default positions in the display window.
8. Right-click the singlepart_output derived data element and click Rename. Type “Singlepart Output” and click OK.
9. Right-click the Singlepart Output derived data element and click Add To Display.
10. Right-click the Multipart To Singlepart tool and click Run.
11. Examine the output layer in your ArcMap display and the layer’s attribute table.
12. Click the Identify tool and click a polygon.

Notice that each polygon now contains its own information for climate zone, length, and area.

Selecting suitable locations

You’ll now use the Select tool to extract the suitable locations.

1. Expand the Extract toolset located in the Analysis Tools toolbox.
2. Click and drag the Select tool into the ModelBuilder window.



Note that the tool added will be called Select (2) as you have already added a Select tool.

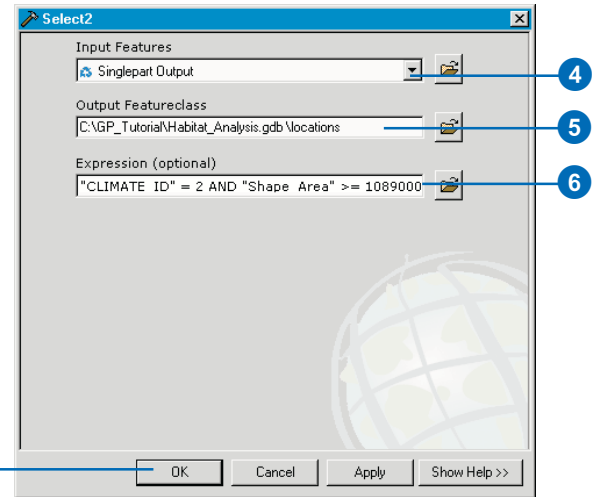
3. Right-click the Select (2) tool element and click Open.
4. Click the Input Features dropdown arrow and click the variable Singlepart Output.
5. You'll put the final output inside your Habitat_Analysis geodatabase. Clear the default path and type "locations" for the value of the Output Feature Class parameter. The path to your current workspace will automatically be added when you click the next parameter in the dialog box.
6. Type the following expression for the value of the Expression parameter or click the browse button to the right of the Expression parameter and build the expression using the Query Builder dialog box.

"CLIMATE_ID" = 2 AND "Shape_Area" >= 1089000
OR "CLIMATE_ID" = 3 AND "Shape_Area" >= 2178000

Alternatively, browse in your operating system to query.txt located in your GP_Tutorial folder, copy the query, and paste it into the Expression parameter's input box.

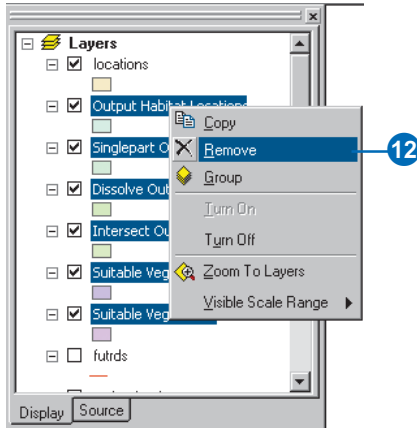
The expression extracts all areas where the climate zone is 1 or 2 (polygons with a CLIMATE_ID of 2) and the area of the vegetation patch is greater than or equal to 25 acres (1,089,000 ft²), or where the climate zone is 3 (polygons with a CLIMATE_ID of 3) and the area of the vegetation patch is greater than or equal to 50 acres (2,178,000 ft²).

7. Click OK.



8. Click Auto Layout, then Full Extent to display all the model elements in their default position in the display window.
9. Right-click the element locations and click Rename. Type "Output Habitat Locations", then click OK.
10. Right-click the element Output Habitat Locations and click Add To Display.
11. Right-click the Select (2) tool element and click Run.

- Click all results except locations in the table of contents—press the Ctrl key to select more than one layer—right-click, and click Remove.



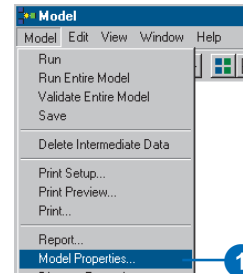
- Examine the layer locations in your ArcMap display.

It shows areas capable of supporting the California gnatcatcher based on the criteria set. Potential patches contain suitable vegetation types that are far enough away from roads, where the elevation is lower than 250 m, and where the slope of the terrain is less than 40 percent. The size of the vegetation patch is greater than or equal to 25 acres in climate zones 1 and 2 and greater than or equal to 50 acres in climate zone 3.

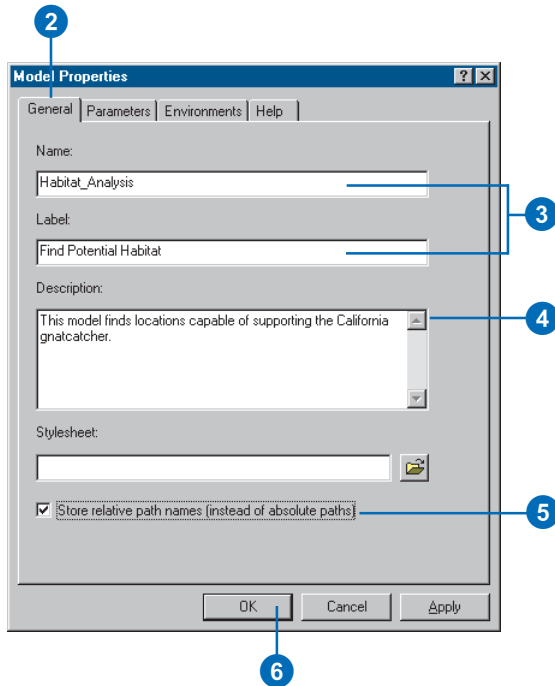
Saving and renaming the model

Any new model you create has a default name (Model). You can change the name of the model in the Model Properties dialog box.

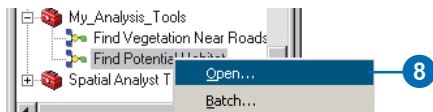
- Click Model and click Model Properties.



- Click the General tab.
- Specify a new name for your model, “Habitat_Analysis”, and a label, “Find Potential Habitat”. You would use the name if you were running the model at the command line or inside a script. The label is the display name for the model.
- Add a description for the model, describing the model’s contents.
This description will appear in the Help panel of the model’s dialog box. You’ll view it later in this tutorial.
- Check Store relative path names so that all paths for sources of information referenced by the tool are saved relative to the location of the toolbox. If the toolbox and its data are moved, paths to data sources will be altered accordingly.
- Click OK.



7. Click Model, Close, and Yes to save your changes.
8. Right-click your Find Potential Habitat model in the ArcToolbox window and click Open.



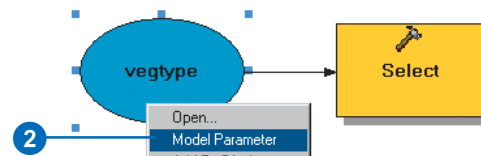
Notice that there are no parameters available in the model's dialog box. This is because you have not set any variables as model parameters.

9. Click Cancel.

Setting model parameters

In a similar way to setting variables as parameters for the dialog box of a script, as you did in Exercise 3, you must set the variables in the model as model parameters that you want to display in your model's dialog box.

1. Right-click the Find Potential Habitat tool and click Edit to open the model.
2. Right-click the element vegtype and click Model Parameter.



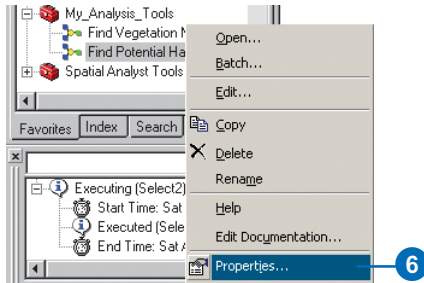
A P icon will appear next to the element indicating it is set as a model parameter.

3. Right-click vegtype and click Rename. Type "Input Vegetation" and click OK. This is the name that will be displayed for the input data parameter in the model's dialog box.
4. Right-click Output Habitat Locations and click Model Parameter.
5. Click the Model menu and click Save, then click the Model menu again and click Close.

The Properties dialog box provides a way to change the name of the model, set parameters (an alternative way to setting them within the model), change their order, and set the tool's environment settings. You'll now examine the

parameters that are set to display in the dialog box of your new Find Potential Habitat model.

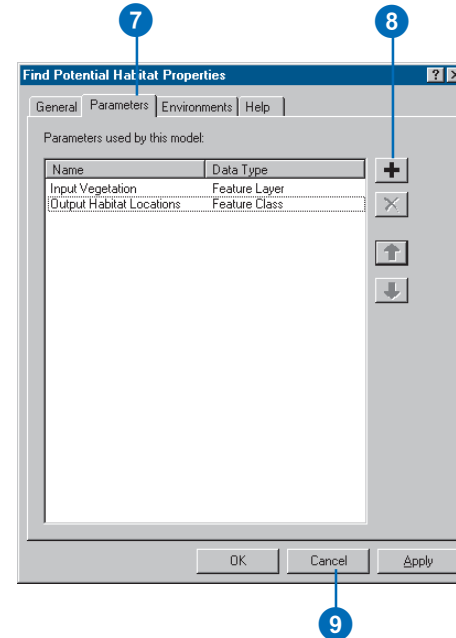
6. Right-click your Find Potential Habitat model and click Properties.



7. Click the Parameters tab.

Notice the two variables set as model parameters for the user to specify values in the model's dialog box. The order of the parameters can be changed by clicking a parameter, then clicking Move Up or Move Down. The order for the parameters in the model's dialog box mimics the order seen in this list.

8. Click Add to see the variables that could also be set as parameters, then click Cancel on the Add Model Parameter dialog box.
9. Click Cancel.



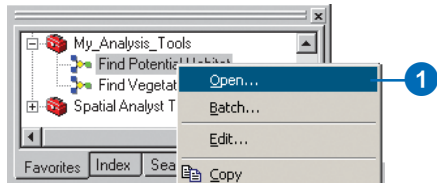
Overwriting outputs

By default, results from running tools are not overwritten. You'll specify to overwrite results from running tools—via their dialog boxes—so the existing locations output will be overwritten.

1. Click Tools on the Main menu and click Options.
2. Click the Geoprocessing tab and check to overwrite the outputs of geoprocessing operations.
3. Click OK.

Running the model from its dialog box

1. Right-click the Find Potential Habitat model in the ArcToolbox window and click Open.



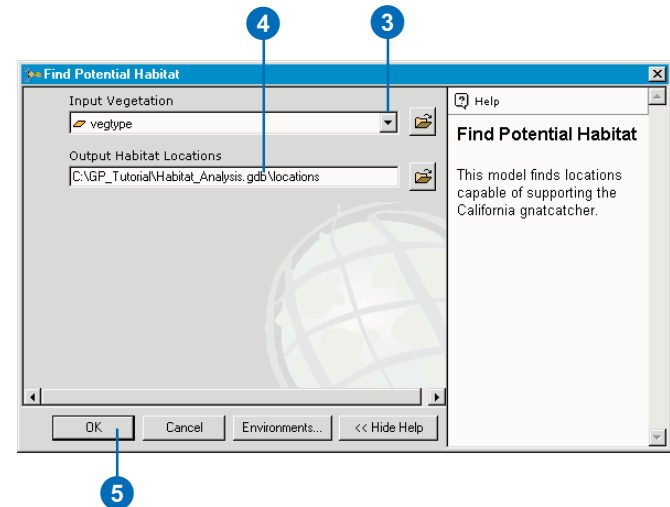
You now have two parameters displayed on the model's dialog box.

2. Click the Show Help button to see the description for the model that you wrote earlier.
3. Leave the default value for the Input Vegetation parameter to use vegtype.
4. Leave the default value for the Output Habitat Locations parameter to create an output called locations. This output will replace the output inside your current workspace, Habitat_Analysis.gdb.

Although you already executed each individual tool while building the model, these tools will be executed again, and the results will be overwritten.

Note: When you run a model from its dialog box, unlike when running the model within the ModelBuilder window, the intermediate data—data referenced by derived data elements that are flagged as intermediate—will be deleted after the model has executed.

5. Click OK to run the model.



6. Examine the status messages in the message section of the Command Line window and examine the result that is added to your display.

Note: The final result is added to the display by default. This is because the option to add results of geoprocessing operations to the display is checked on by default in the Geoprocessing tab of the Options dialog box, accessed via the Tools menu on the Main menu.

Note also that the 'locations' result created earlier by running the model from within the ModelBuilder window has been overwritten because the option to overwrite the outputs from geoprocessing operations is turned on in the Options dialog box. Also, the result produced is permanent, because the option to create temporary results from tools is turned off by default in the Options dialog box.

7. Click File on the Main menu of ArcMap and click Save.

In this exercise you created a new model that finds potential habitat locations for the California gnatcatcher. In the next exercise you will find out which of these locations will be impacted by proposed roads.

You can continue on to Exercise 5 or stop and complete the tutorial at a later time. If you do not move on to Exercise 5 now, click File, then Exit to close your ArcMap session, but do not delete your GP_Tutorial folder.

Exercise 5: Finding habitat patches impacted by proposed roads

Once you have created models within, or added scripts to, a toolbox, they can be used just like any of the system tools. They can be added to the ModelBuilder window of another model, run from the command line, or run from another script.

In this exercise you'll create a new model and add to it the Find Potential Habitat model you created in Exercise 4. You'll clip the result from the Find Potential Habitat model (locations) by using buffer zones you'll create around the proposed roads to find potential habitats that may be impacted by the proposed roads.

This exercise will take approximately 16 minutes to complete.

Setting up

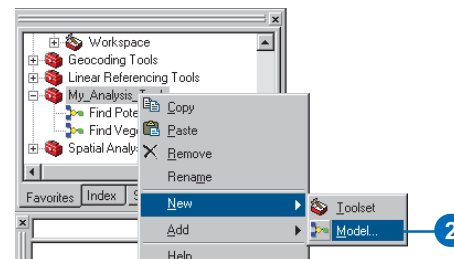
If you didn't do the previous exercises, follow the next eight steps. If you did Exercise 4, go to the next section, 'Creating a new model'.

1. Open ArcCatalog. In the ArcCatalog tree copy the folder GP_Tutorial from C:\arcgis\ArcTutor\Geoprocessing\Results\Ex4 to your C:\ drive, or an alternative drive, then close ArcCatalog.
2. Open ArcMap and open the map document Habitat_Analysis.mxd from your GP_Tutorial folder. Right-click the ArcToolbox window and click Add Toolbox. Navigate to your My_Analysis_Tools toolbox and click Open.
3. Right-click the ArcToolbox window and click Environments.

4. Click the General Settings dropdown arrow to expand its contents and type "C:\GP_Tutorial\Habitat_Analysis.gdb", or type your alternative location, for the value of the Current Workspace environment setting. Alternatively, click the Browse button to navigate to this location.
5. Type "C:\GP_Tutorial\Temp_Results.gdb", or type your alternative location, for the value of the Scratch Workspace environment setting. Alternatively, click the Browse button to navigate to this location, then click OK.
6. Click Tools on the Main menu and click Options.
7. Click the Geoprocessing tab.
8. Check Overwrite the outputs of geoprocessing operations. Then click OK on the Options dialog box.

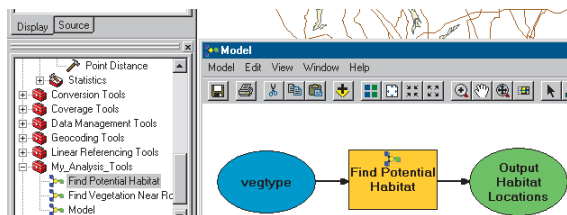
Creating a new model

1. If you have closed ArcMap, open it again and open the map document Habitat_Analysis.mxd from your GP_Tutorial folder.
2. Right-click your My_Analysis_Tools toolbox in the ArcToolbox window, point to New, and click Model.



Adding the Find Potential Habitat model

1. Click the Find Potential Habitat model inside your My_Analysis_Tools toolbox and drag it into the ModelBuilder window of the new model.



Notice the icon on the Find Potential Habitat tool element indicates that the tool is another model. You can right-click this element and click Edit to open the model inside its ModelBuilder window. You hide the complexity of a larger model by breaking it down into smaller submodels.

Buffering the proposed roads

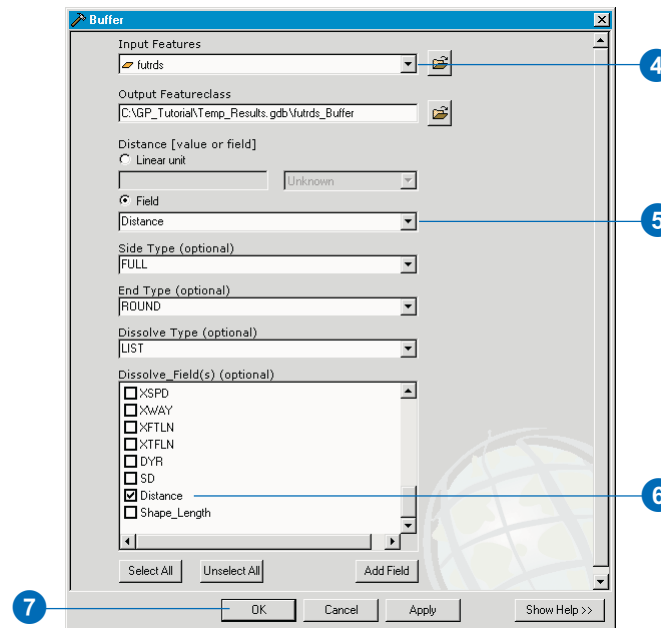
1. Expand the Proximity toolset located inside the Analysis Tools toolbox.
2. Click and drag the Buffer tool into the display window.
3. Right-click the Buffer tool element that represents the Buffer tool in the model and click Open.

4. Click the Input Features dropdown arrow and click the layer futrds.

Leave the default location and name for the Output Feature Class parameter.

5. Click Field, click the dropdown arrow, and click Distance. The values in this field will be used for the width of the buffer.

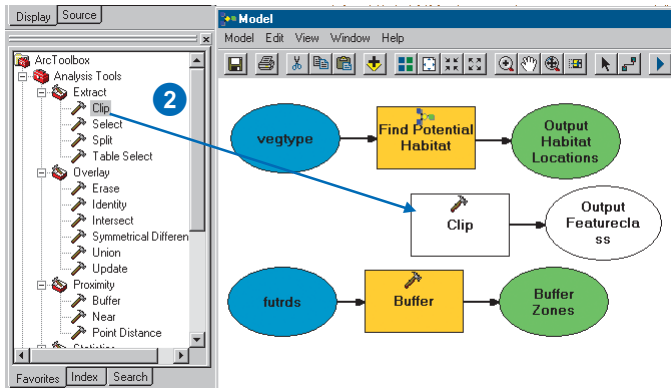
6. Click the Dissolve Type dropdown arrow and click LIST to list the dissolve fields. Check Distance in the Dissolve_Field(s) list. All polygons that share the same distance value will be dissolved.
7. Click OK.



8. Click the Full Extent tool to display all elements in the display window.
9. Right-click the derived data element futrds_Buffer and click Rename.
10. Type "Buffer Zones" and click OK.

Clipping habitat patches using buffer zones

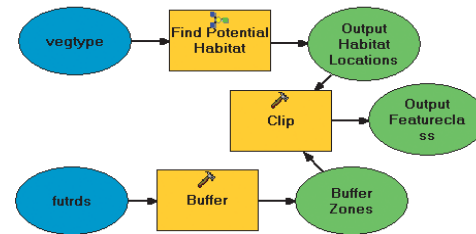
1. Expand the Extract toolset located inside the Analysis Tools toolbox.
2. Click the Clip tool and drag it into the ModelBuilder window.



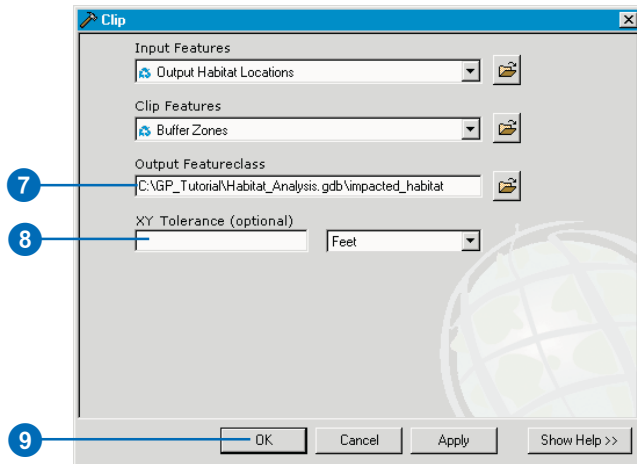
3. Click the Add Connection tool.



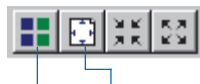
4. Click the element Output Habitat Locations, then click the Clip tool element.
A *connector* is drawn between the two elements, connecting the output from the Find Potential Habitat tool as input for the Clip tool.
5. Click the element Buffer Zones, then click the Clip tool.



6. Double-click the Clip tool element to view the parameter values set in the tool's dialog box.
Notice that by connecting the input data elements to the tool element, the values for the Input Features parameter and the Clip Features parameter are set inside the tool's dialog box.
7. As the result from the Clip tool is the final output from the model, clear the default path and type "impacted_habitat" for the name of the Output Feature Class. The path to the current workspace will be supplied by default.
8. Leave the default value (blank) for the XY Tolerance parameter and the units as Feet. An appropriate default value will be calculated using the units of the spatial reference set for the output, in this case, in feet, as the input data spatial reference units are in feet.
9. Click OK.



- Click Auto Layout, then click Full Extent to display all the model elements in the display window in their default position.



Auto
Layout Full
Extent

Renaming elements

- Right-click the element impacted_habitat and click Rename.
- Type "Output Impacted Habitat" and click OK.
- Right-click the element vegtype and click Rename.
- Type "Input Vegetation" and click OK.
- Right-click the element futrds and click Rename.
- Type "Input Proposed Roads" and click OK.

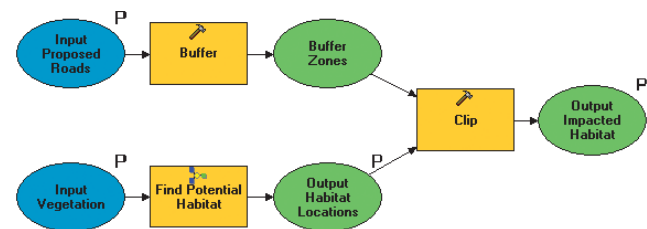
Setting model parameters

You want the user of this model to be able to specify the following inputs: Input Vegetation and Input Proposed Roads, as both of these could change over time. You also want the user to be able to set the path and name for the Output Habitat Locations output and the final output, Output Impacted Habitat.

So the user can specify values for these parameters, you'll make them model parameters.

- Right-click the element Output Impacted Habitat and click Model Parameter.
- Right-click the element Output Habitat Locations and click Model Parameter.
- Right-click the element Input Vegetation and click Model Parameter.
- Right-click the element Input Proposed Roads and click Model Parameter.

Your model should resemble the graphic that follows.

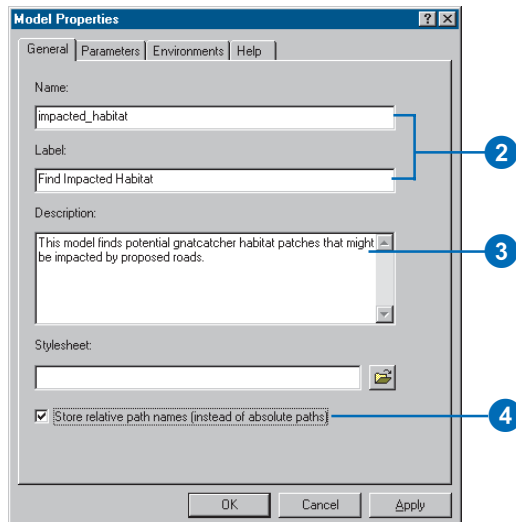


- Click Model and Save, then click Model, then Close.

Renaming the model

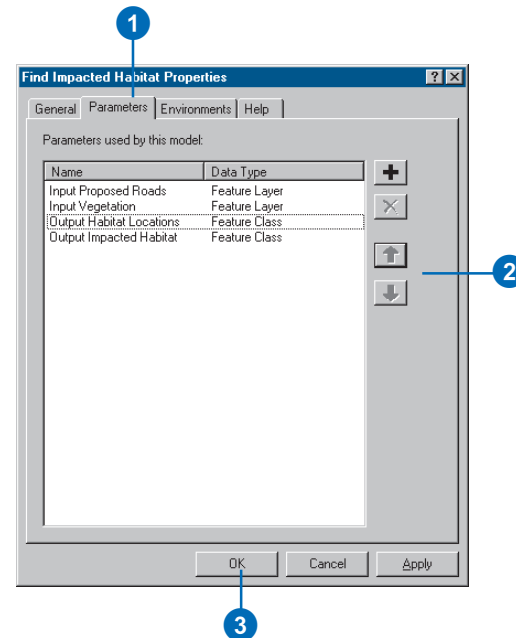
Models can be renamed in the Model Properties dialog box within the ModelBuilder window, as you saw earlier, or from the Properties option on the model's context menu.

1. In the ArcToolbox window, right-click Model inside your My_Analysis_Tools toolbox and click Properties.
2. Click the General tab and type "impacted_habitat" for the name and "Find Impacted Habitat" for the label.
3. For the description of the model, type, "This model finds potential gnatcatcher habitat patches that might be impacted by proposed roads."
4. Check Store relative path names to save all paths for sources of information referenced by the tool relative to the location of the toolbox. If the toolbox and its data are moved, the model will still work without having to repair the paths to derived data.



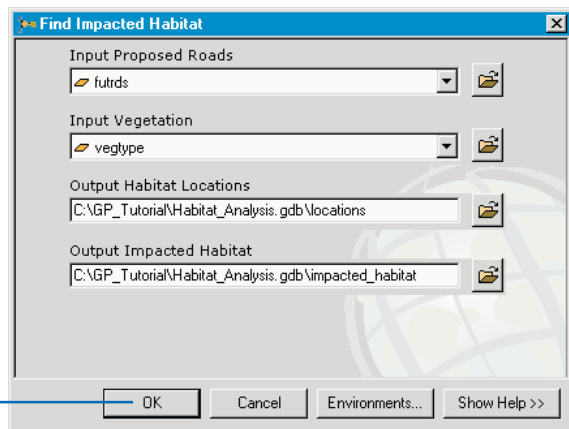
Changing the order of parameters

1. Click the Parameters tab.
2. Click Input Proposed Roads and click the Move Up arrow to place this parameter at the top of the list. Use the Move Up or Move Down arrows to place Input Vegetation after Input Proposed Roads, then Output Habitat Locations, and Output Impacted Habitat last.
3. Click OK.



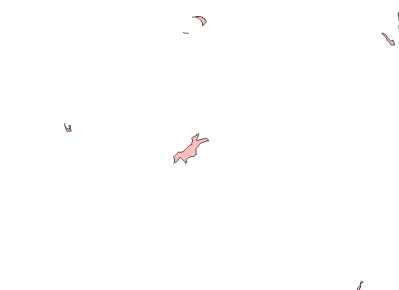
Running the model

1. In the ArcToolbox window, double-click the Find Impacted Habitat model to open its dialog box.
2. Click OK to run the model.



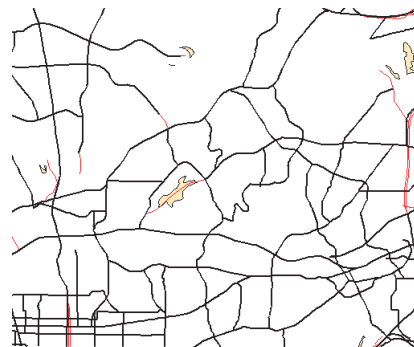
Displaying the results

1. Examine the status messages in the Command Line window.
2. Turn off all layers in the table of contents except impacted_habitat.
3. Examine the result in the ArcMap display.



These polygons represent habitat locations that would be impacted by the introduction of new roads.

4. Turn on the layers majorrds_shp and futrds in the table of contents and zoom in on the impacted areas.
5. Change the symbology of the layers if desired.



There is actually only one area of impacted habitat that a proposed road will go directly through. The rest of the impacted areas are close to new roads, but the roads will not travel directly through them. These areas fall within a certain distance from the proposed roads (the buffer distance assigned). They will be impacted by proposed roads, but only by noise and fume pollution.

6. Turn on the locations layer.
7. Change the order of the layers in the table of contents so they are in the following order from the top of the table of contents: futrds, majorrds_shp, impacted_habitat, then locations. The order of the rest of the layers does not matter.
8. Zoom to the full extent of the map to see all high-quality habitat locations, including those that are impacted by proposed roads.



After an investigation in the field to see if there are indeed gnatcatchers present in the impacted habitat locations, the proposed roads that are planned to be routed through or near these locations will need to be assessed to see if they can be rerouted away from areas of high-quality habitat.

Saving the map document

1. Click File on the Main menu of ArcMap and click Save.
2. Click File again and click Exit to close the ArcMap session.

This brings you to the end of this tutorial. You should now have the experience to start using your own data and to get your geoprocessing tasks done with greater efficiency and ease. The rest of this book explains how to use ArcGIS to perform geoprocessing in more depth, so use it as a guide as you begin performing your own specific geoprocessing tasks.

