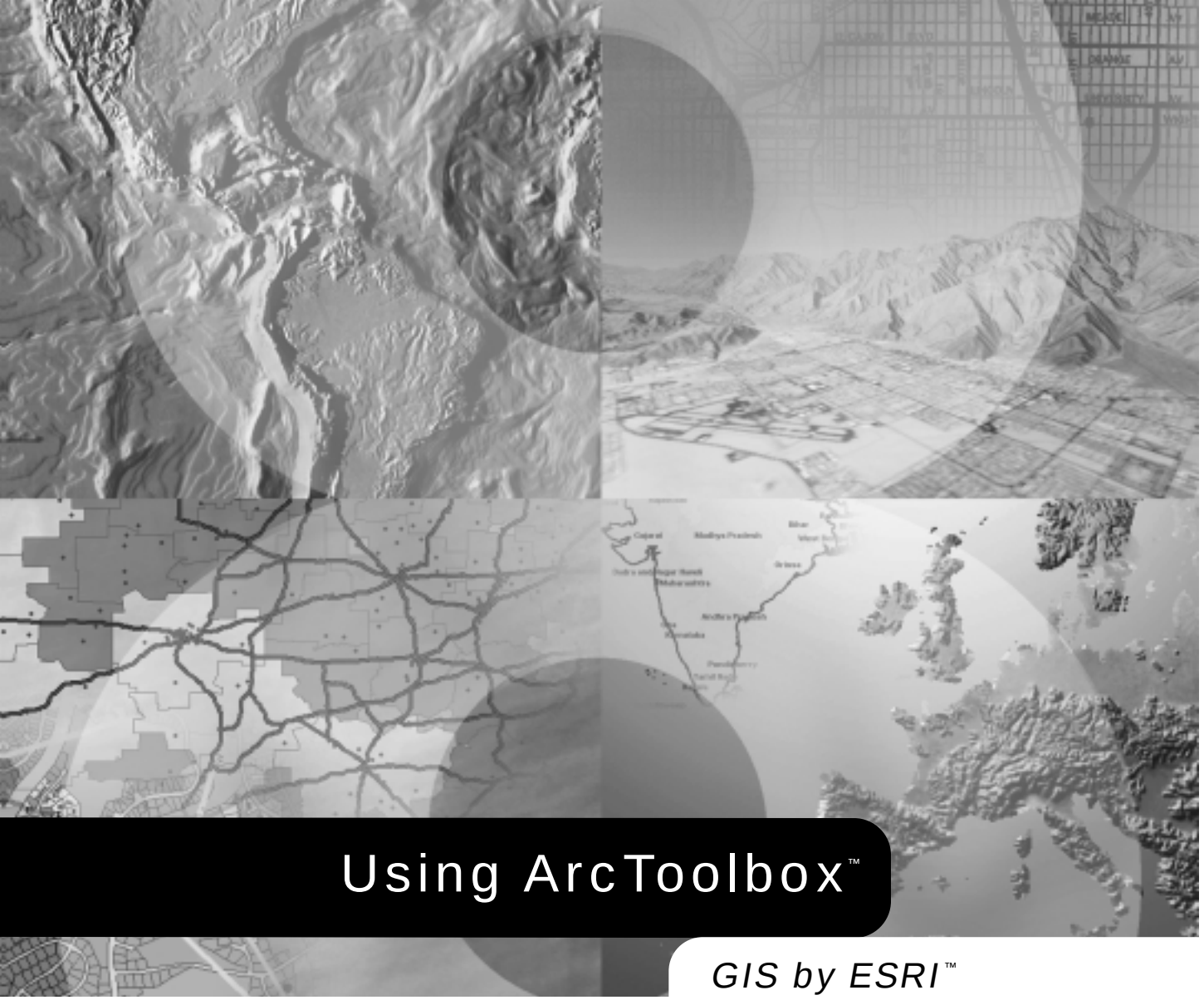


Using ArcToolbox™

GIS by ESRI™

CONTRIBUTING WRITERS

Corey Tucker, Ian DeMerchant, Barbara Bicking, Chris Boyd, Jason Pardy, Mike Conly and Ghislain Prince, Gary Kabot, and Ashley Pengelley

Contents

Introducing ArcToolbox

1

IN THIS CHAPTER

- **The tools**
- **How ArcToolbox helps you work**
- **Tips on learning ArcToolbox**

Welcome to ESRI® ArcGIS™ ArcToolbox™ software, the application that provides access to all of ArcInfo™ software's powerful *coverage* processing and analysis functions. ArcToolbox provides a very rich and powerful set of *geoprocessing* functions—well over 100 tools for easy access and use. It is easy to find the tool you wish to use and start it to perform the desired geoprocessing operation.

ArcInfo is well known for its advanced geoprocessing capabilities. Most of the tools in ArcToolbox are used to manipulate ArcInfo coverages, the popular geographic information system (GIS) data type. The tools in ArcToolbox create and integrate a vast array of data formats into usable GIS databases, perform advanced GIS analysis, and manipulate GIS data. With ArcToolbox, virtually all major spatial data formats can be converted to and from ArcInfo coverages, grids, and triangulated irregular networks (TINs). Topology can be generated and maintained; map sheets can be joined, clipped, and split; and advanced coverage-based modeling tools can be used.

ArcToolbox allows you to perform tasks such as:

- Finding the datasets necessary for an overlay using ArcCatalog™ software and dragging them onto the Overlay Wizard in ArcToolbox
- Converting 20 coverages into Drawing Exchange Format (DXF) files using the Coverage to DXF tool in a single operation
- Creating ARC Macro Language (AML™) scripts and processing them on a remote machine (UNIX® or Windows NT®) at a scheduled time

- Creating custom tools in the Microsoft® Windows® development environment of your choice and installing them as custom tools in ArcToolbox
- Using the Project Wizard to change the coordinate reference system of a coverage and look at its new geographic view in ArcCatalog
- Building coverage topology using the Clean tool and adding feature attributes using the Join Tables tool

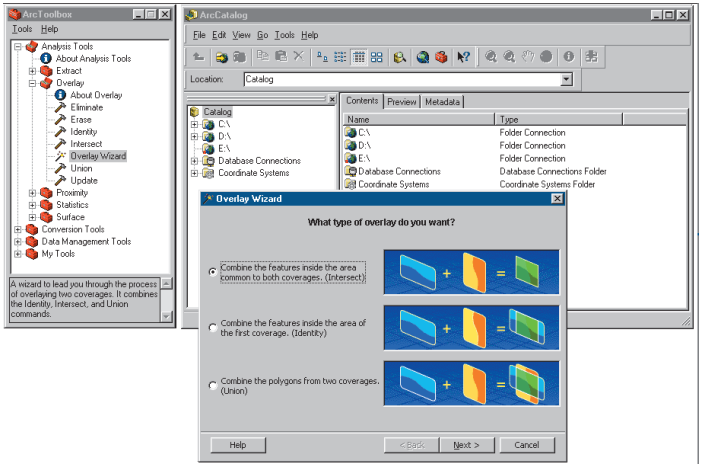
The tools

ArcToolbox is organized into toolsets that provide solutions for different types of tasks. The four main toolsets are:

- Data Management Tools
- Analysis Tools
- Conversion Tools
- My Tools

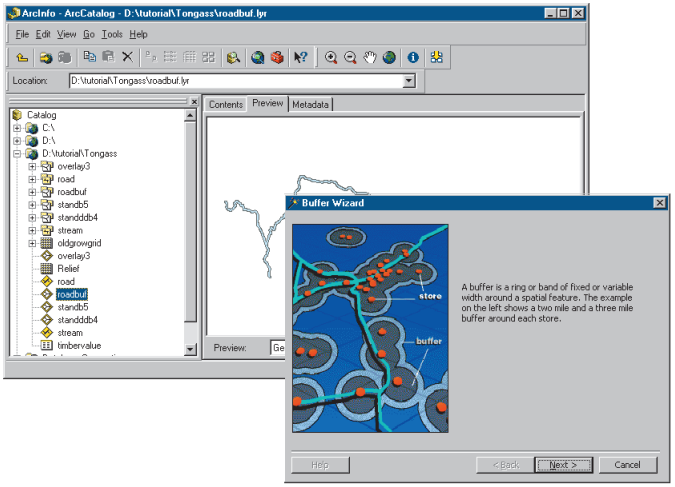
Numerous tools, organized in categories of smaller toolsets, are contained in each main toolset. The tools are organized in a logical manner based on the type of solution they provide.

A fully customizable toolset called My Tools is provided. You can place tools from any toolset within it or add a custom tool you create to build on the power of the existing tools. You will quickly realize how customizable ArcToolbox truly is.



How ArcToolbox helps you work

ArcToolbox offers a simple visual interface that guides you through a task, providing all the information and help you need to efficiently complete the job. If you're a longtime user, you no longer need to type your geoprocessing or conversion commands into a command line or create and debug an AML. ArcCatalog helps you locate and access the datasets you need to complete a variety of tasks easily and efficiently using ArcToolbox.



Processing locally or remotely

You may have very large datasets located on powerful server machines (UNIX or Windows NT) that are connected to your personal computer. ArcToolbox lets you compose a geoprocessing job on your machine and run it on the remote machine using an ArcInfo Geoprocessing Server. For information on how to run jobs remotely using a Geoprocessing Server, see Chapter 3, 'ArcToolbox basics'.

Tips on learning ArcToolbox

If you're new to GIS, remember that you don't have to learn everything about ArcToolbox to get immediate results. Begin learning ArcToolbox by reading Chapter 2, 'Quick-start tutorial'. In Chapter 2, you'll learn how easy it is to use the various tools in ArcToolbox in conjunction with ArcCatalog, and you'll gain insight into the steps you can take to complete certain tasks. ArcToolbox comes with the data used in the tutorial, so you can follow along step by step at your computer. You can also read the tutorial without using your computer.

Finding answers to questions

Like most people, your goal is to complete your tasks while investing a minimum amount of time and effort in learning how to use software. You want intuitive, easy-to-use software that gives you immediate results, without having to read pages of documentation. However, when you do have a question, you want the answer quickly so that you can complete your task. That's what this book is all about—getting you the answers you need when you need them.

This book describes geoprocessing tasks—from basic to advanced—that you'll perform with ArcToolbox. Although you can read this book from start to finish, you'll likely use it more as a reference. When you want to know how to do a particular task, such as operating a tool in batch mode, just look it up in the table of contents or index. What you'll find is a concise, step-by-step description of how to complete the task. Some chapters also include detailed information that you can read if you want to learn more about the concepts behind the tasks. You may also refer to the glossary in this book if you come across any unfamiliar GIS terms or need to refresh your memory.

About this book

This book is designed to introduce ArcToolbox and its capabilities. The topics covered in the various tasks and the tutorial in Chapter 2 assume you are familiar with geoprocessing and the fundamentals of GIS. If you have never used a GIS before or feel you need to refresh your knowledge, please take some time to read *Getting Started with ArcGIS,* which you received in your ArcInfo package. You don't need to read it to continue with this book, but you should use it as a reference if you encounter tasks with which you are unfamiliar.

Chapter 3 covers all of the basic and some of the advanced functions of ArcToolbox. Chapter 4 shows you how to customize the ArcToolbox interface to match your needs.

Chapter 5 discusses custom tools and how to create them using the Visual Basic® (VB) programming language. This task assumes you have some experience with VB and the ArcInfo Open Development Environment (ODE). However, if you have not used either of these environments, you will still be able to complete the tasks as long as you have Visual Basic 5.0 or 6.0 installed.

Getting help on your computer

In addition to this book, use the ArcGIS Desktop Help system to learn how to use ArcToolbox. To learn how to use the ArcGIS Desktop Help system on your computer, see 'Getting help' in Chapter 3 of this book.

Contacting ESRI

If you need to contact ESRI for technical support, see the product registration and support card you received with ArcToolbox or refer to 'Getting Technical Support' in the 'Getting more help' section of the ArcGIS Desktop Help system. You can also visit

ESRI on the Web at *www.esri.com* and *support.esri.com* for more information on ArcToolbox and ArcGIS.

ESRI education solutions

ESRI provides educational opportunities related to geographic information science, GIS applications, and technology. You can choose among instructor-led courses, Web-based courses, and self-study workbooks to find education solutions that fit your learning style. For more information, go to *www.esri.com/education*.

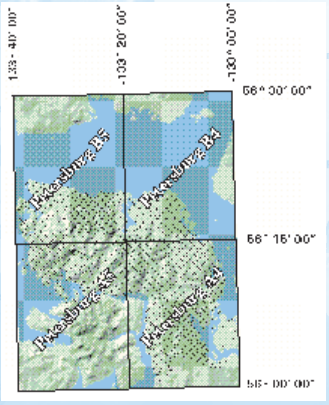
Quick-start tutorial

2

IN THIS CHAPTER

- Exercise 1: Organizing your data in ArcCatalog
- Exercise 2: Processing the forest stands
- Exercise 3: Processing the streams and roads
- Exercise 4: Converting data
- Exercise 5: Creating the analysis coverage
- Exercise 6: Computing the timber value

Conducting a GIS processing project is easier than ever with the powerful tools in ArcToolbox. When used in conjunction with ArcCatalog—the application for browsing, storing, organizing, and distributing data—ArcToolbox lets you meet the geoprocessing needs of your project quickly and efficiently.



In this tutorial, you'll use ArcToolbox and ArcCatalog to conduct a logging study for a portion of the Tongass National Forest in southeastern Alaska. By performing overlays, buffers, and other geoprocessing tasks with ArcToolbox, you'll calculate the dollar value of trees in areas suitable for logging. You'll use ArcCatalog to organize and manage your data, as well as to immediately preview the results for each step.

The study area is shown here. You will conduct the study for the PetersburgB4 and PetersburgB5 15-minute quadrangles. You will use base coverages and grids for forest stands, rivers, roads, and old growth to complete the study. This data has been provided with the software. Additionally, several of the coverages you would normally derive in the course of this project are provided to eliminate repetitive processing steps.

When conducting your analysis, you must keep certain study criteria in mind. First, harvest areas must be 100 meters from all roads and fish spawning streams. Second, areas must not include old growth forest.

You'll use several datasets throughout the course of the project. The following table provides descriptions of these datasets.

Coverage	Description
Oldgrowgrid	Grid of old growth forest stands
Overlay3	Overlay of buffered roads and streams, forest stands, and old growth forest
Road	Major roads
Roadbuf	Major roads buffered 100 meters
Standb5	Forest stands for PetersburgB5 in Universal Transverse Mercator (UTM)
Standddb4	Forest stands for PetersburgB4 in decimal degrees
Stream	Streams

The data you'll need for this tutorial is included on the ArcToolbox installation disk. The datasets were provided courtesy of the USDA Forest Service, Tongass National Forest, Ketchikan Area. They have been simplified by ESRI. The Forest Service cannot assure the reliability or suitability of this information. Original data was compiled from various sources, and spatial information may not meet National Map Accuracy Standards. This information may be updated, corrected, or otherwise modified without notice.

All exercises in the tutorial are processed locally, rather than being sent to a Geoprocessing Server for remote processing. It is recommended that you do not use the Geoprocessing Server for this tutorial if you have ArcInfo Workstation installed locally. If you need to use a Geoprocessing Server, see 'Using a Geoprocessing Server' in Chapter 3 before you start the tutorial.

This tutorial is designed to let you explore the capabilities of ArcToolbox and ArcCatalog at your own pace and without the need for additional assistance. You'll need about one hour of focused time to complete the six exercises in the tutorial. However, you can also perform the exercises one at a time if you wish.

Exercise 1: Organizing your data in ArcCatalog

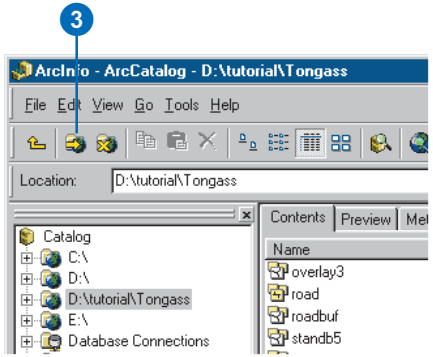
Before you begin your geoprocessing and analysis work, you must first find and organize the data that you'll need. You should organize your data in such a way that you'll be able to find it quickly and efficiently. This will be done using ArcCatalog.

Copying and connecting to data

You'll begin by copying the tutorial data provided with the ArcToolbox software to a folder on a local disk. You will work with a copy of the data locally if the data was installed on a connected disk, in order 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 in ArcCatalog.

1. Start ArcCatalog by either double-clicking a shortcut installed on your desktop or using the Programs list in your Start menu.
2. Copy the ArcToolbox tutorial data from the directory where it is installed to your own tutorial workspace. In the examples shown here, the data has been copied to local drive D:\ and placed in a folder called "tutorial".

   In ArcCatalog, data is accessed through folder connections. When you look in a folder connection, you can quickly see the folders and data sources it contains. You'll now begin organizing your tutorial data by creating a folder connection to it.
3. Click the Connect To Folder button and navigate to the data folder. Click OK to establish a folder connection.



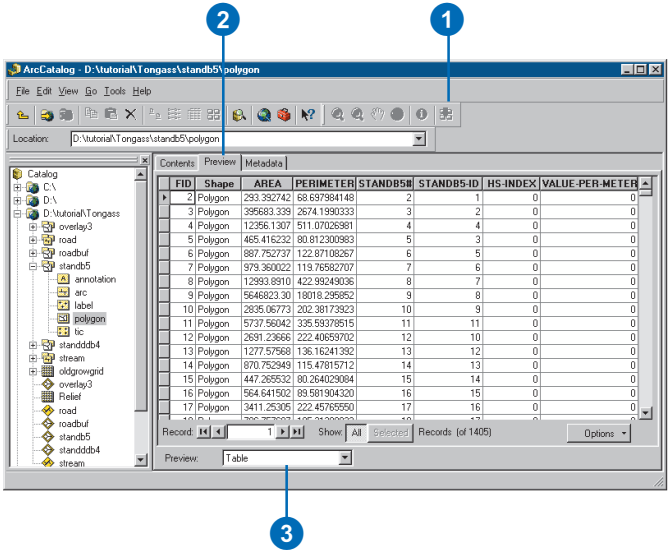
Your new folder connection—D:\tutorial\Tongass—is now listed in the ArcCatalog tree. You will now be able to access all of the data needed for your project through that connection.

Exploring your data

Before you begin your analysis, you should explore the datasets provided for the project. This will help you get a better feel for ArcCatalog and the tutorial data.

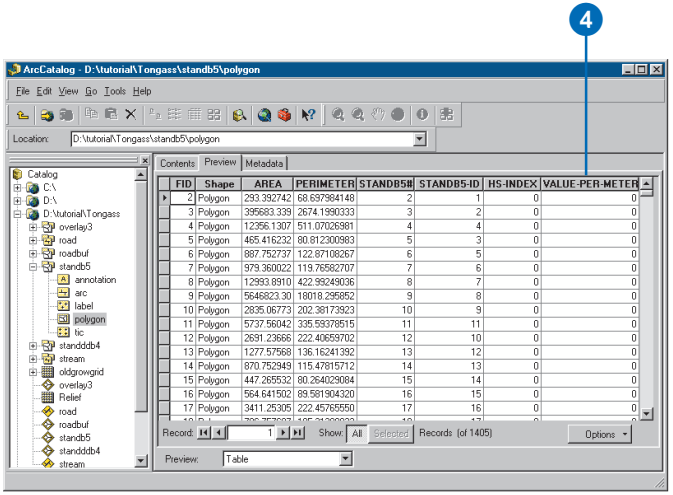
1. Click the Thumbnails button on the Standard toolbar to display previously created thumbnail images of the datasets in your Tongass folder.

   Layers have been created for all the coverages. The Standb5 coverage is in a UTM projection, while Standddb4 is in unprojected decimal degrees. All other coverages and grids have been merged and are in UTM projection.



2. Click the plus sign next to the D:\tutorial\Tongass connection to see the *datasets* contained in the folder. Click the Preview tab and click each dataset in the tree.
3. Double-click the Standb5 coverage to open it. Click the polygon *feature class*. Click the Preview dropdown arrow and click Table to see the feature attribute table contents. The VALUE-PER-METER field stores the value of the timber in each forest stand as a density—dollars per meter squared.

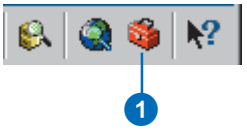
   Polygons with a value-per-meter attribute of zero are nonforested areas such as lakes and grasslands. Because the purpose of this project is to calculate the value of trees in areas suitable for logging, you will exclude the nonforested areas from the timber harvest. You will then compute the value of the timber in the remaining area using this attribute.
4. Right-click the VALUE-PER-METER field to open a context menu. Sort the table into ascending and then descending order. What is the lowest nonzero value per meter? What is the greatest?



Starting ArcToolbox

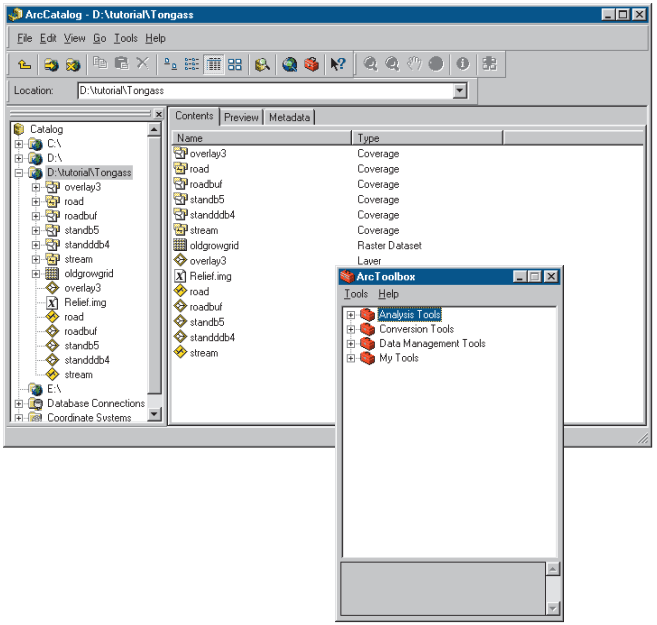
In the remaining sections of the tutorial, you'll conduct your geoprocessing work using ArcToolbox. You'll still need to use ArcCatalog to manage and examine your datasets. Keep both applications open for the remainder of the tutorial.

1. Click the Launch ArcToolbox button on the ArcCatalog toolbar to start ArcToolbox.



You can also start ArcToolbox as you would any other application—from the Start menu or from a shortcut on your desktop.

2. Both the ArcCatalog and ArcToolbox windows should now be open. Size and arrange the two applications on your screen so that both are visible.



You are now ready to start the first part of the tutorial: processing the forest stands. You'll be introduced to several ArcToolbox *tools* and *wizards* and will use these for your analysis.

Exercise 2: Processing the forest stands

In the first exercise, you prepared for the latter exercises by organizing your data. Now you are ready to begin processing your data. Two forest stand coverages—Standddb4 and Standb5—currently cover the entire study area. Before you can begin your study, these coverages must be merged.

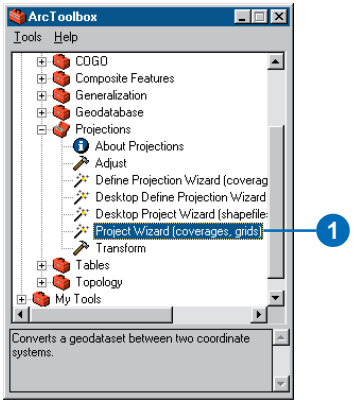
As noted in Exercise 1, the Standb5 coverage is projected in a UTM coordinate system, while the Standddb4 coverage is in unprojected decimal degrees. In order for you to conduct a meaningful analysis of these areas, the coverages must share the same coordinate system. For this reason, you will project Standddb4 to match the coordinate system of Standb5. Once this is done, the topology will have to be rebuilt as it is lost when a coverage is projected.

Projecting a coverage

ArcToolbox software's Project Wizard lets you easily project a coverage to another coordinate system. You can use the Project Wizard to manually define the output projection (you supply all the projection parameters), or you can have the wizard use the projection information stored in an existing coverage. For this study, you'll use the wizard to project the Standddb4 coverage to match the coordinate system of Standb5.

1. Double-click the Project Wizard (coverages, grids) in the Projections toolset of Data Management Tools.

   The first panel of the Project Wizard (coverages, grids) should now be open. You will use the projection information in Standb5 to project Standddb4.



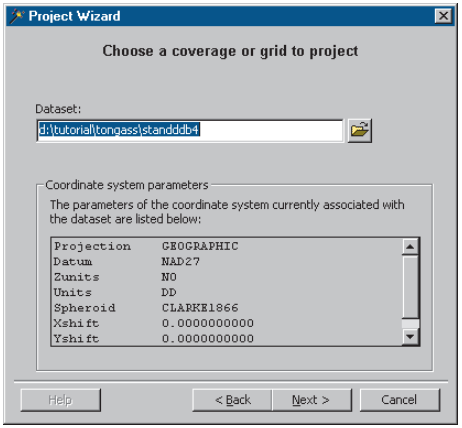
2. Click the option to Project my data to match existing data. Click Next.

   This panel is used to specify your input coverage. ArcToolbox gives you several options for setting input and output dataset names. You can type the full pathname to the dataset into the text box. You can also click and drag a dataset, or datasets, from the ArcCatalog tree or Contents tab and drop it on the text box. Alternatively, you can click the Browse button to open the ArcCatalog browser and navigate to your dataset.

   ArcToolbox has a feature called sticky paths. This means that it remembers the path to the last dataset you specified and will assume that the same path applies when you type only a dataset name into another tool or wizard. It also remembers output dataset paths.

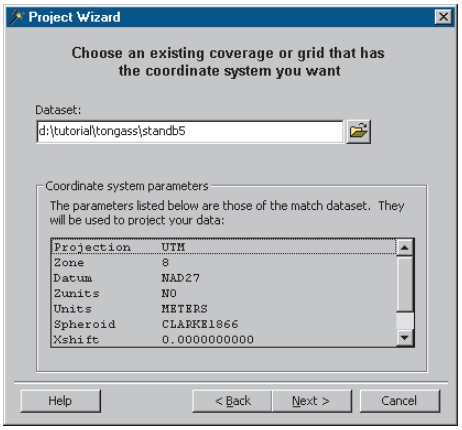
Tutorial instructions will simply ask you to type coverage names and their paths into the appropriate text boxes. However, feel free to use any of the techniques just described to make the entry.

3. Type "D:\tutorial\Tongass\Standddb4" in the Dataset text box. Click Next.



Use the next panel of the wizard to specify the name of the coverage whose projection information will be used to define the output projection. You will use Standb5 to define it. Notice that in both this and the previous panel, the wizard displays the projection information for the coverage you have selected.

4. Type "D:\tutorial\Tongass\Standb5" in the Dataset text box. Click Next.



The next panel appears, where you will specify the output coverage name. You will call the output coverage "Standb4" and store it in your Tongass folder.

5. Type "D:\tutorial\Tongass\Standb4" for the output dataset. Click Next. A summary page appears. Once you have reviewed the summary, click Finish.

   A message appears to tell you that the wizard is processing your request. This message appears in all tools and wizards when a process is active. When the tool or wizard is finished, the message disappears, indicating that the process is complete.

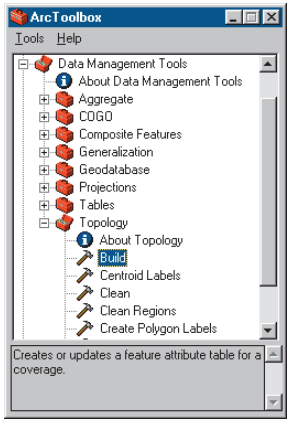
   Your new coverage should now appear in your Tongass folder.

6. Click the Standb4 coverage in the Catalog tree and click the Preview tab. Switch to Geography on the Preview dropdown list to see your new coverage.

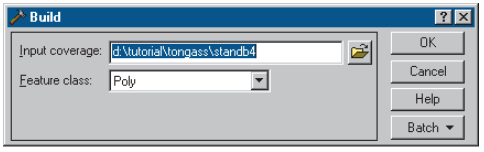
Building topology

If you double-click the Standb4 coverage in the ArcCatalog tree, you'll note that the coverage doesn't contain a polygon feature class. This is because changing a coverage's projection removes its topology. You must now rebuild polygon topology for your new coverage using the Build tool before going any further with your analysis.

1. Double-click the Build tool in the Topology toolset of Data Management Tools.



2. Type "D:\tutorial\Tongass\standb4" in the Input coverage text box.



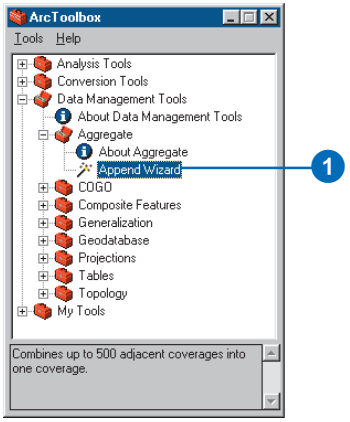
3. Change the Feature class to Poly. Click Yes on the subsequent message box to confirm Poly as the desired feature class. Click OK.

The Standb4 and Standb5 coverages are now in the same projection (UTM), and your new coverage has polygon topology. The next step is to merge the coverages together so that the data can be used as one dataset that matches the extents of the other coverages in the study.

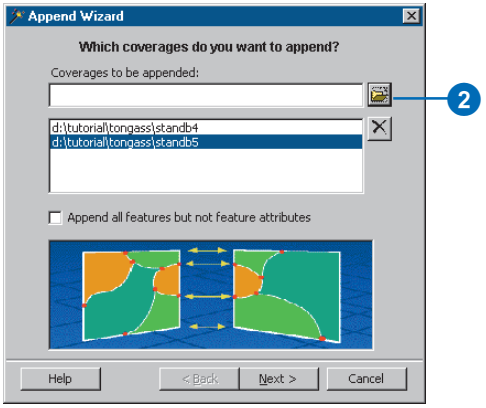
Merging datasets

You can use the Append Wizard to merge the two coverages.

1. Double-click Append Wizard in the Aggregate toolset of Data Management Tools.



The Append Wizard joins multiple coverages together when you type their names in the Coverages to be appended text box. The easiest way to do this is to use the browser and select all of the datasets at once. Simply click the coverage, then hold down the Ctrl or Shift key while clicking the other coverages to select them all.



2. Click the Browse button and navigate to the D:\tutorial\Tongass folder. Select Standb4 and Standb5 and click the Open button.

   Your coverages should now be listed on the wizard. If you made a mistake, you can remove a coverage by selecting it and clicking the Delete button next to the list.

3. Click Next.

   The next panel of the wizard appears. You will use it to specify the feature classes that will be merged.

4. Click Poly in the Feature classes list.

5. Click Next.

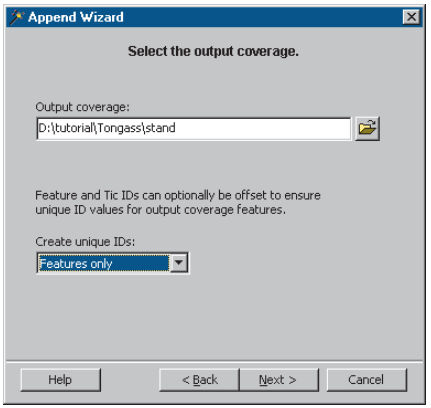
   This panel is used to specify the name of an optional clip coverage. You will not use a clip coverage in this tutorial.

6. Click Next.

   Use the next panel to specify the output coverage name and to offset the feature IDs. Unique feature IDs within the output coverage are necessary in order to maintain a relationship between the new features and the originals.

7. Type "D:\tutorial\Tongass\stand" in the Output coverage text box. Click the Create unique IDs dropdown arrow and click Features only. Click Next.

8. Review the summary panel and click Finish.



Your forest stand data is now ready to be used with the other Tongass datasets in the next exercise. By analyzing the forest stand data along with the coverages you'll create in the upcoming tasks, you will determine which areas are suitable for harvest and how valuable those areas are.

Exercise 3: Processing the streams and roads

In the last exercise, you processed the forest stands; now you will process the streams and roads data to eliminate the areas that do not meet the first of the specified criteria: harvest areas must be at least 100 meters from all fish spawning streams and roads.

To process the streams, you will first select and extract stream segments flagged as fish spawning grounds and place them in a new coverage named Fish. You will then generate a 100-meter buffer around each segment.

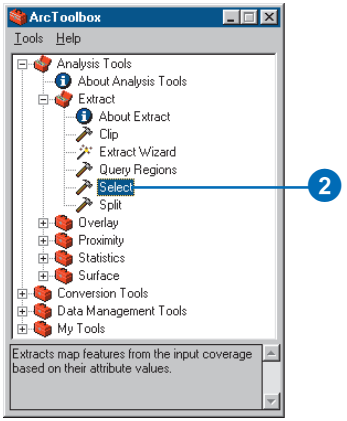
To meet the second part of the criterium—harvest areas must be at least 100 meters from roads—you would have to create a similar buffer. However, this step has already been completed for you to avoid repetition; the extracted roads are represented by the Roadbuf coverage.

Extracting features

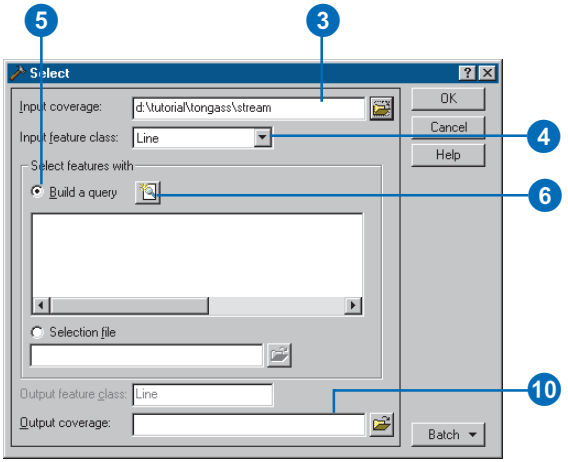
Using the Select tool, you will extract streams that are flagged as fish spawning areas to a new coverage. But first, you should examine the stream coverage in ArcCatalog to see how many streams are in the study area.

1. Click the Stream coverage in the ArcCatalog tree and click the Preview tab to examine the coverage.
2. Double-click the Select tool in the Extract toolset of Analysis Tools.

   You will now specify the coverage and feature class that you are processing, the logical expression that identifies the features (there is a button that opens a query builder), and the output coverage and feature class.



3. Type "D:\tutorial\Tongass\stream" in the Input coverage text box.
4. Click the Input feature class dropdown arrow and click Line.

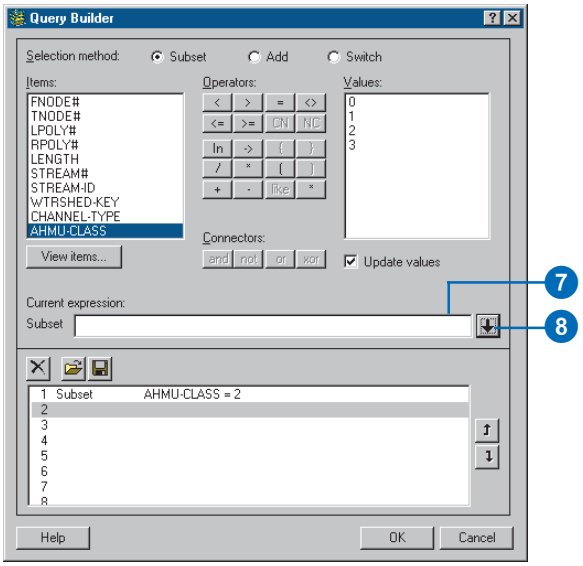


5. Click the first option to Build a query if it is not already selected.
6. Click the Query Builder button to open the Query Builder dialog box.

   You'll use the Query Builder to create the logical expression that identifies the features you want to select. You can type the expression directly into the text box or build the expression by clicking the fields, operators, connectors, and values. You must choose the selection method before creating the expression, as it affects the expression's logic. Once a suitable expression is created, you can add it to the tool's expression list.

   The default selection method is subset. If you are familiar with the ArcInfo TABLES™ module, subset is the same as using the RESELECT keyword when you create a selection expression.
7. Type the expression "AHMU-CLASS = 2" in the Current expression text box. Keep the default selection method.

   The item AHMU-CLASS is used to classify the stream types. All spawning streams have a value of 2. The expression tells the Select tool to extract only those streams with a value of 2—that is, the fish spawning streams.
8. Click the down arrow button next to the Current expression text box to add the expression to the list.



9. Click OK to close the Query Builder.

   Call your new coverage Fish and save it in your Tongass folder.
10. Type "D:\tutorial\Tongass\Fish" in the Output coverage text box on the Select tool. Click OK.

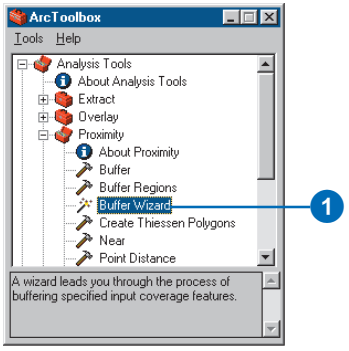
    A message appears when the processing is complete asking whether you want to see the output tool messages. Click Yes and review the number of input and output lines.

    You can now view your new Fish coverage in ArcCatalog.

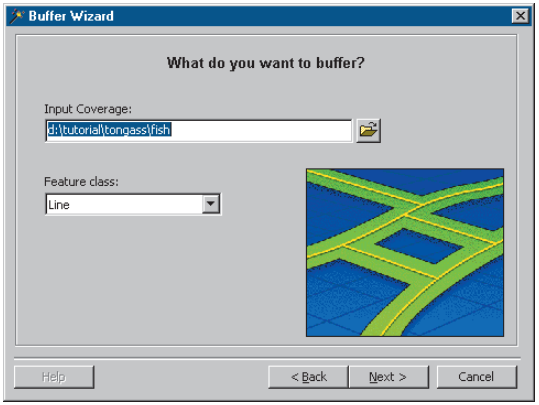
Creating a buffer

Now that you've created a coverage representing the fish spawning streams, you can build a 100-meter buffer around them using the Buffer Wizard. You'll end up with a new coverage called Fishbuf.

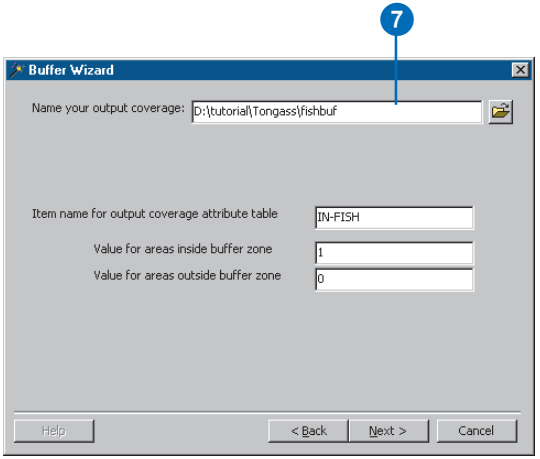
1. Double-click Buffer Wizard in the Proximity toolset of Analysis Tools.



2. Click Next after reading the introductory panel. You want the output coverage to contain polygons, not regions, so accept the default buffer type and click Next.

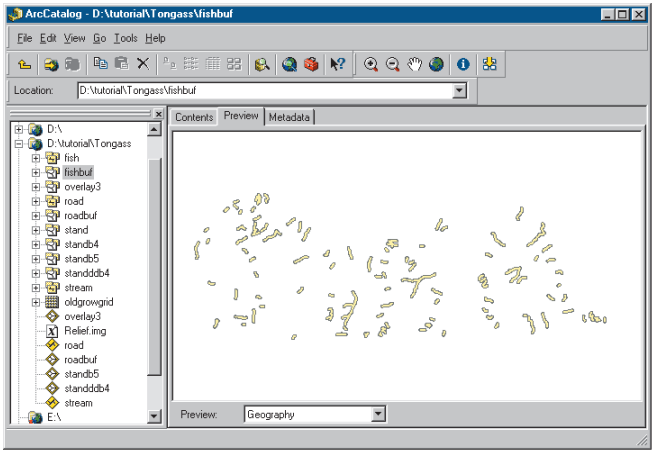


3. Type "D:\tutorial\Tongass\Fish" as the coverage you want to buffer. Click Next.
4. Accept the Single buffer with Specified distance option and click Next.
5. Type "100" as the buffer distance in meters and click Next.
6. Click Both sides with round ends for the buffer style. Click Next.
7. Type "D:\tutorial\Tongass\Fishbuf" as the output coverage.



You must now set the inside and outside values for the output coverage. Inside and outside values are used to determine which areas are inside or outside the buffer area.

8. Type “IN-FISH” as the item name. Type “1” for the inside value and “0” for the outside value. Click Next when finished.
9. After reviewing your choices on the last panel, click Finish to run the wizard.
10. View your Fishbuf coverage in ArcCatalog.
11. Examine the polygon attribute table for Fishbuf. Note the IN-FISH field and its values. Records with an IN-FISH value of 0 indicate a polygon that is not within 100 meters of a stream, while records with a value of 1 are within that distance. This item and its values will be important in a latter step that determines what areas are available for forest harvesting.



As mentioned earlier, one of the timber value criteria requires a 100-meter buffer on major roads. To make this buffer, you would follow the same process you just followed to create the Fishbuf coverage.

However, to eliminate a repetitive step, the Roadbuf coverage was created for you. It is in your Tongass directory; use ArcCatalog to examine it. Pay particular attention to the IN-ROAD polygon attribute.

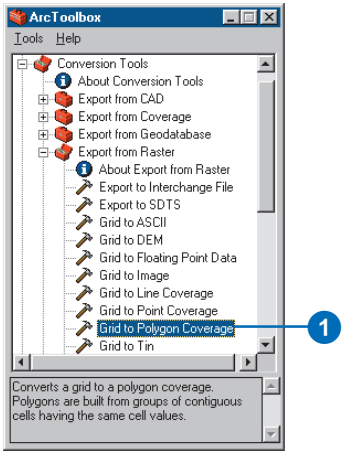
In the next exercise, you’ll create an old growth forest coverage. This coverage, along with the Stand, Fishbuf, and Roadbuf coverages, will be used to generate the final results.

Exercise 4: Converting data

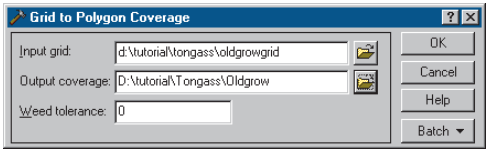
In the previous exercises, you processed forest stands, streams, and roads to help in your study. Now, you'll convert the *grid* of old growth forest areas into a coverage that will be used in an overlay with the forest stands. This grid was created exclusively for this tutorial; it was not provided by the Forest Service.

You'll use the Grid to Polygon Coverage tool to complete the conversion.

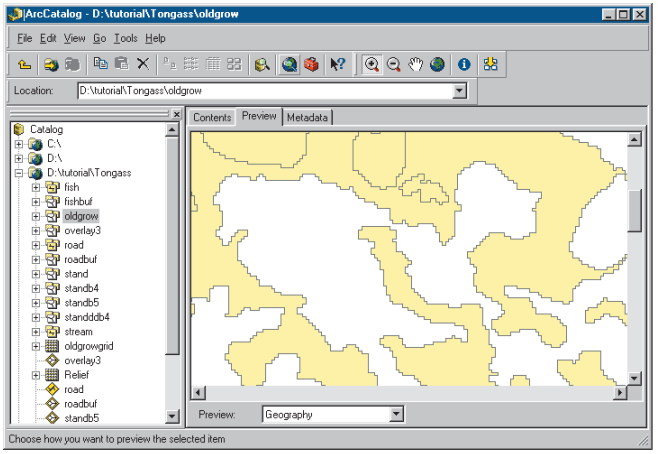
1. Double-click Grid to Polygon Coverage in the Export from Raster toolset in Conversion Tools.



2. Type "D:\tutorial\Tongass\oldgrowgrid" in the Input grid text box. Type "D:\tutorial\Tongass\Oldgrow" in the Output coverage text box. Click OK.

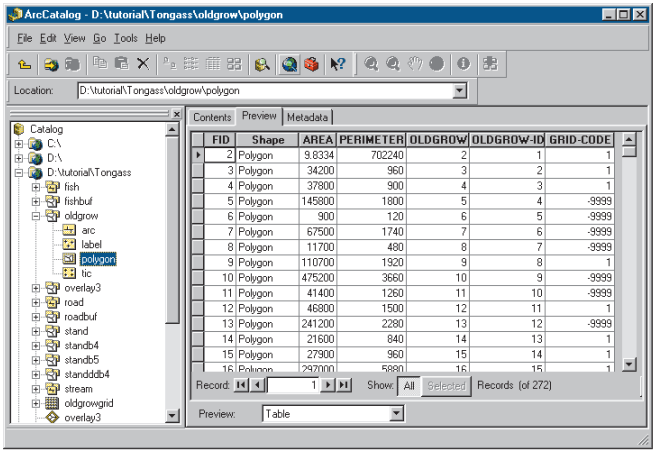


When the tool is finished processing the request, you should see your new coverage, Oldgrow, in the ArcCatalog tree.



3. Click the Preview tab in ArcCatalog to view the Oldgrow coverage in Preview view. Zooming in reveals the common stair-step effect found in vector data that has been converted from raster data.
4. Examine the attributes for the coverage's polygon feature class. Areas of old growth have a GRID-CODE value of 1. All other areas were "Nodata" in the grid and have a value of -9999. When inadequate information is available for a cell location of a grid, the location can

be assigned a value of Nodata. Nodata and "0" are not the same; "0" is a valid value. Because Nodata represents inadequate information, Nodata cells cannot be used in calculating the statistics in a grid's statistics (STA) table.



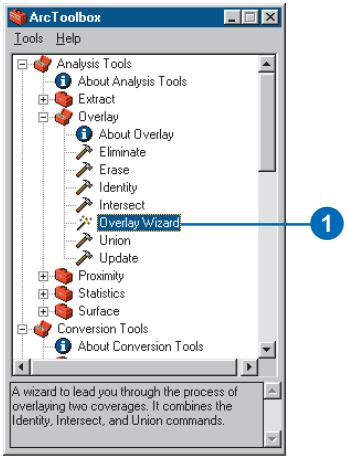
You now have all the coverages you need for your analysis. In the next exercise, you'll overlay them to create a final coverage for the study.

Exercise 5: Creating the analysis coverage

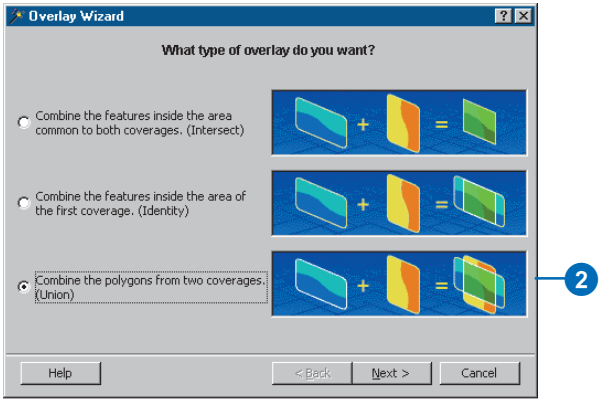
Now that you have organized, processed, and converted your data, you are ready to create the final analysis coverage. To create the final analysis coverage, it is necessary to overlay the Stand, Fishbuf, Roadbuf, and Oldgrow coverages. This would normally require you to perform two union overlays—Roadbuf with Fishbuf, and Stand with Oldgrow—and then intersect the two resulting coverages to create the final analysis coverage.

However, to avoid repetitive tasks, you will only perform one union overlay—Roadbuf with Fishbuf—so that you can experience the Overlay wizard. The final coverage required for the analysis has been created for you and is named Overlay3. You will examine the Overlay3 coverage at the end of this section.

1. Double-click Overlay Wizard in the Overlay toolset in Analysis Tools.

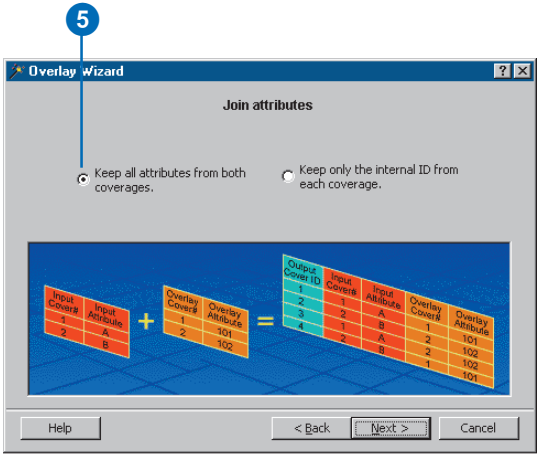


2. Click the third option to Combine the polygons from two coverages, then click Next.



3. Type "D:\tutorial\Tongass\Fishbuf" as the input coverage. Click Next.
4. Type "D:\tutorial\Tongass\Roadbuf" as the overlay coverage. Click Next.

5. Click the option to Keep all attributes from both coverages, as they are needed to determine what areas are outside of the road and stream buffers. Click Next.

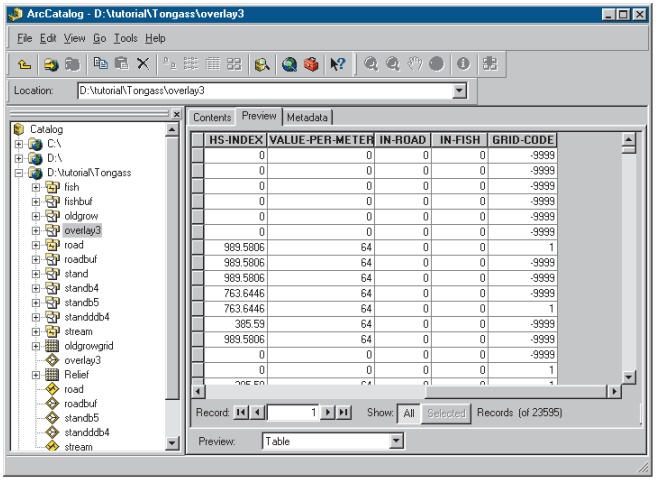


6. Type "D:\tutorial\Tongass\Overlay1" as the name of the output coverage. You want to use the default fuzzy tolerance, so click Next.

7. Review the summary panel to ensure all input is correct. Click Finish when you are done.

   As mentioned, the final overlay coverage, Overlay3, was created for you. Overlay3 was created using the same procedure you just followed. All of the extra fields resulting from the series of overlays (Cover#, Cover-ID) were deleted using ArcCatalog as they are not required for this analysis. The items created in the buffered coverages and the converted grid will be used to determine what areas are harvestable.

8. View the Overlay3 coverage in the Catalog. Examine the attribute values of the polygon attribute table. You should see the items you created when you buffered the roads and streams as well as the item created in the grid conversion.



You should now have a clear understanding of the steps that were needed to arrive at this point. ArcToolbox broke down complex tasks into easy-to-follow tools and wizards, and ArcCatalog allowed you to immediately preview your results.

With all of the geoprocessing work complete, you are ready for Exercise 6: Computing the timber value.

Exercise 6: Computing the timber value

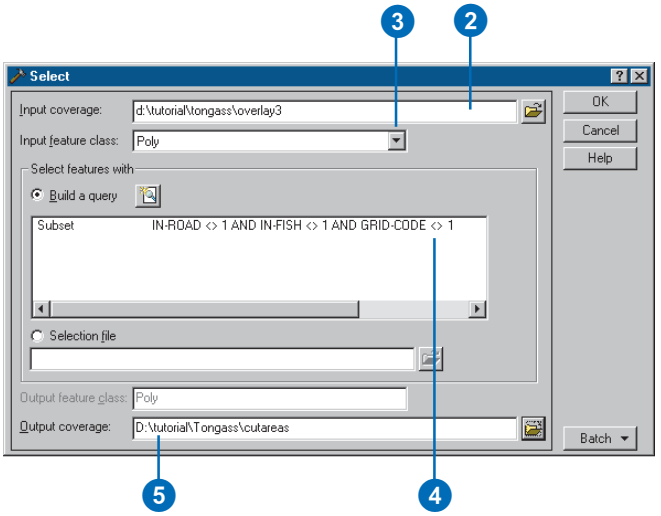
The first five exercises focused on geoprocessing tasks. In this exercise, you will build on these tasks by computing the value of the trees in harvestable areas. These are areas that are not old growth and are 100 meters away from fish spawning streams and major roads. You will use the Select tool to extract the polygons that meet these criteria into a new coverage called Cutareas. The Statistics tool will then weight the VALUE-PER-METER field by the area of each polygon and sum the results.

Remember that value-per-meter is a density expressing the value of the original stands in terms of dollars per square meter. Because this value is a density, it still applies even though the original stand polygons have been divided into many smaller polygons through the sequence of overlays that you have performed.

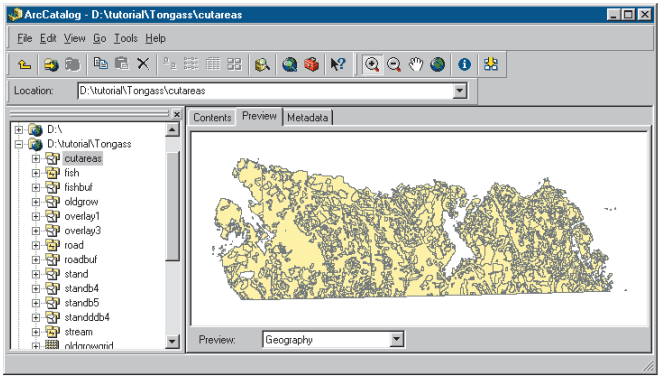
Extracting the polygons

You will begin by extracting the polygons that meet all of the criteria.

1. Double-click the Select tool in the Extract toolset of Analysis Tools.
2. Type "D:\tutorial\Tongass\Overlay3" in the Input coverage text box.
3. Click the Input feature class dropdown arrow and click Poly.
4. Click the Query Builder button to open the Query Builder dialog box. Create a logical expression to select all polygons that have a value not equal to 1 for the IN-ROAD, IN-FISH, and GRID-CODE items. The item that stores the old growth flag was given the name GRID-CODE. Remember that you can type the expression directly into the text box or build it by clicking the fields, operators, connectors, and values. Keep the default selection method of subset.



5. Type "D:\tutorial\Tongass\Cutareas" in the Output coverage text box. Click OK.



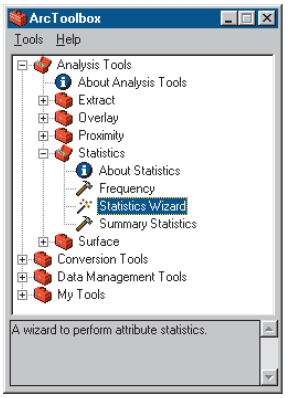
6. Examine the Cutareas coverage in ArcCatalog.

Visually, your new coverage is not much different from your Overlay3 coverage. However, in the new coverage, polygons that can't be harvested now have all their attributes set to 0. This includes the VALUE-PER-METER field. You can easily verify this by opening the Cutareas polygon attribute table and sorting it on cutareas-id. Those polygons with an ID of 0—there are quite a few—didn't meet your criteria.

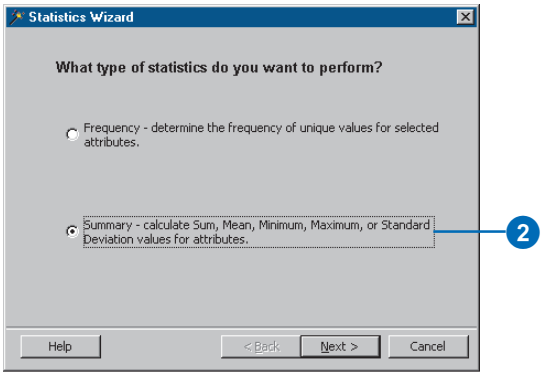
Generating statistics to show timber value

The last part of this tutorial involves using the Statistics Wizard to compute the dollar value of the trees in each polygon and to get a sum of all the values. The wizard will do this by multiplying the polygon areas (which are in square meters) by the VALUE-PER-METER values (which are in dollars per square meter). The result will be written to one record in an INFO™ file.

1. Double-click Statistics Wizard in the Statistics toolset of Analysis Tools.

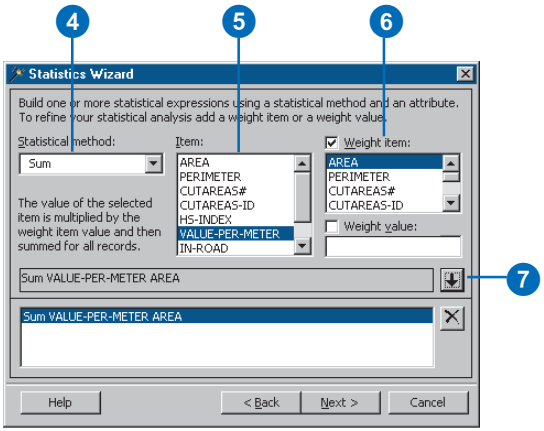


2. Click the second option to sum the contents of the value-per-meter item. Click Next.



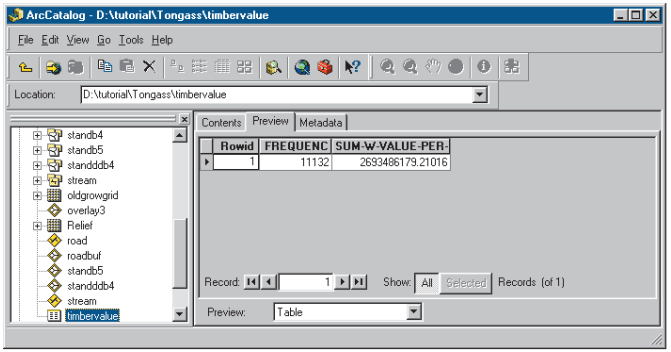
3. Type "D:\tutorial\Tongass\cutareas.pat" as the input table. Click Next.

4. Click the Statistical method dropdown arrow and click Sum.
5. Click VALUE-PER-METER in the Item list.
6. Check Weight item and click AREA in the list.



7. Click the down arrow button to add the expression to the list. Click Next.
8. Click Calculate statistics for all records. Click Next.
9. Type "D:\tutorial\Tongass\timbervalue" as the output table name. Click Next. Click Finish after reviewing the summary.

10. Examine your new timbervalue table in ArcCatalog using Preview view. What is the total value of all harvestable timber in the study area? If your number is about $2.7 billion, then you didn't make a mistake. Trees are worth a lot of money! Of course, this is the value of almost 110 square miles of forest.



This tutorial introduced you to the extensive capabilities of ArcToolbox and ArcCatalog. Using both applications, you quickly and easily performed a number of GIS operations and observed the results. You can now use these applications to perform your own analyses.

You have yet to uncover many features of ArcToolbox. In the next few chapters, you will review all the features that make ArcToolbox a user-friendly and complete GIS application for your daily needs.

ArcToolbox basics

3

IN THIS CHAPTER

- Starting ArcToolbox
- What are toolsets?
- Starting a tool
- Finding the tools you need
- Getting help
- Setting the precision environment
- Using and defining relates
- Using ArcCatalog
- Dragging and dropping data
- Operating tools in batch mode
- Batch processing in wizards
- Using a Geoprocessing Server
- Checking a remote process
- Scheduling a remote process

ArcToolbox lets you work with data from different sources, run a tool or wizard on multiple datasets, and direct where the processing will occur. *Tools* and *wizards* are arranged in a logical manner to help you find them quickly and start the task at hand.

You'll find that using ArcToolbox with ArcCatalog increases your productivity when performing geoprocessing or conversion tasks. It doesn't take long to become familiar with the ArcToolbox interface, as the tools are organized into toolsets that have names that are representative of functions.

Starting ArcToolbox

Starting ArcToolbox is the first step to geoprocessing. However, before you can start, ArcToolbox must be installed on your computer or network. If you don't know whether it's installed yet, check with your system administrator or install it yourself using the installation guide.

Once the software is installed, you can start ArcToolbox from the Start button on the Windows taskbar.

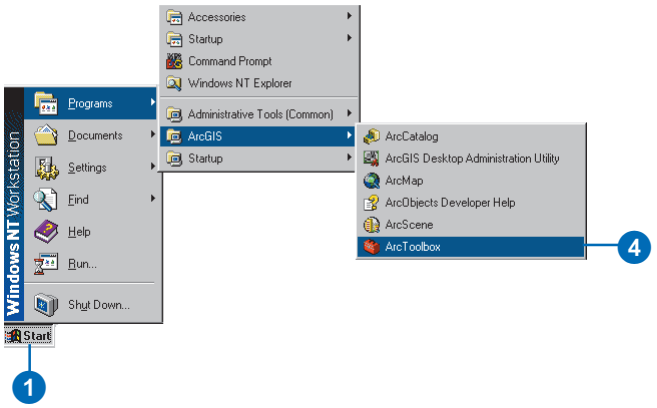
Tip

Starting ArcToolbox from ArcCatalog

You can also start ArcToolbox from ArcCatalog by clicking the Launch ArcToolbox button on the Standard toolbar.

Starting ArcToolbox from the Start menu

1. Click the Start button on the Windows taskbar.
2. Point to Programs.
3. Point to ArcGIS.
4. Click ArcToolbox.



What are toolsets?

The tools in ArcToolbox are organized into four main toolsets. They are:

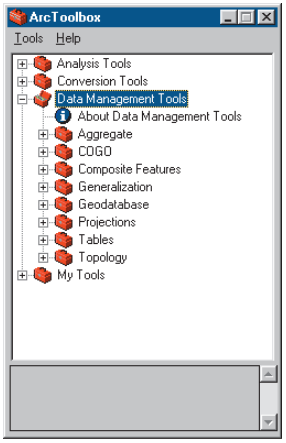
- Data Management Tools
- Analysis Tools
- Conversion Tools
- My Tools

The ArcToolbox tree begins with the four main toolsets. When you click an item in ArcToolbox, a description appears at the bottom of the menu.

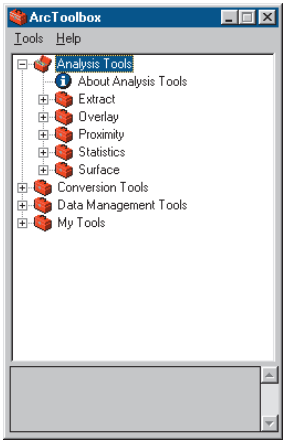
You can change the size of the ArcToolbox window and position it anywhere you like on your desktop. Changes you make to its size and position remain in future sessions.

Each main toolset is comprised of a number of smaller toolsets. These toolsets further organize the tools into groups based on their function. The name of each toolset describes the type of tools it contains.

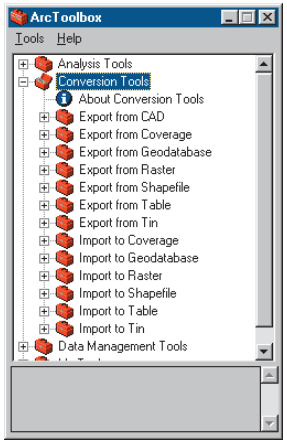
The Data Management Tools toolset contains a number of smaller toolsets that further organize its contents. With Data Management Tools, you can manage a coverage's topology, map projection, and attributes. If you need to modify the characteristics of a coverage or table, you can use this toolset.



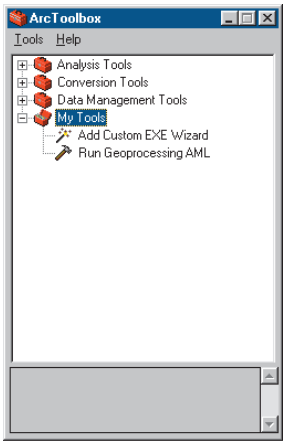
In the Analysis Tools toolset, you find most of the geoprocessing power of ArcToolbox. With the tools in this toolset, you can perform overlays, create buffers, calculate statistics, merge datasets, and much more. When you want to solve a spatial or statistical problem, you should always look in the Analysis Tools toolset.



Within the Conversion Tools toolset are the tools you need to convert coverages, grids, and TINs to other supported data formats. The tools are organized into smaller toolsets that pertain to the type of conversion being performed. Most commonly used data formats are supported.



ArcToolbox provides wizards to assist you in the more complex conversions. Use Conversion Tools whenever you need to change one type of dataset to another.



The last of the main toolsets is My Tools. You can modify this toolset by adding the tools you use most frequently or tools associated with a common task. When you first start ArcToolbox, only two tools are presented. One of these is a wizard that guides you through the process of installing a custom tool; the other tool runs an AML script. My Tools does not initially have toolsets. You can customize it by creating your own toolsets.

Starting a tool

Before using a tool for geoprocessing, you must start it from ArcToolbox. You can start a tool in several ways:

- From the ArcToolbox tree
- From the tool's context menu
- From the Tools menu

The Tools menu displays the last four tools you have used. This lets you start a recently used tool quickly when you need it again.

Tip

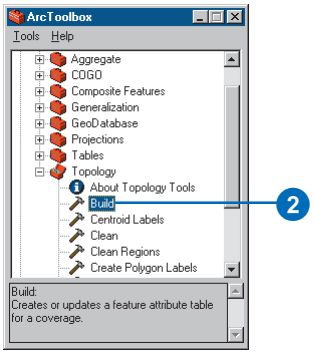
Another way to start a tool from the Tools menu

You can also start a tool by selecting it and clicking the Tools menu, then clicking Open.

Starting a tool from the tree

1. Navigate to the tool you want to start.
2. Double-click the tool.

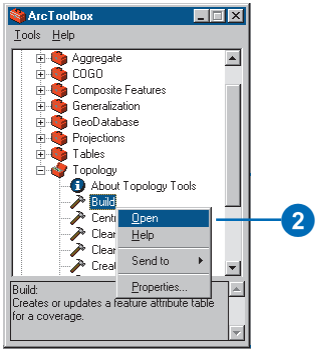
   The dialog box for the tool appears.



Starting a tool from its context menu

1. Navigate to the tool you want to start.
2. Right-click the tool and click Open.

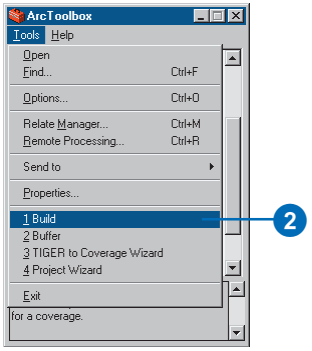
   The dialog box for the tool appears.



Starting a tool from the Tools menu

1. Click Tools.
2. Click the tool you want to start.

   The dialog box for the tool appears.



Finding the tools you need

You can use the ArcToolbox Main menu to help you find the tool you want to use. ArcToolbox provides a Find tool that offers three ways to locate a tool.

You can find the location of a tool by using its name, the equivalent ArcInfo command, or a keyword.

Use an ArcInfo command when you know the command name but not the tool name.

A keyword best describing the task you want to perform is useful when you don't know a tool's name or the equivalent ArcInfo command.

Tip

Are they all here?

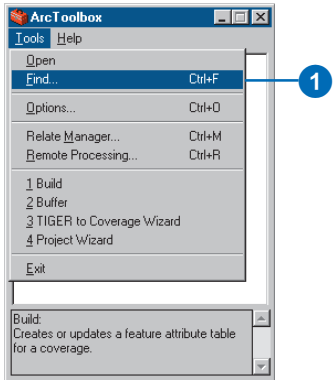
ArcToolbox is a collection of ArcInfo software's most used geoprocessing and conversion commands. It is possible that the tool you are looking for does not exist in ArcToolbox yet. If you are unable to find the tool you need, you can still run the command in ArcInfo.

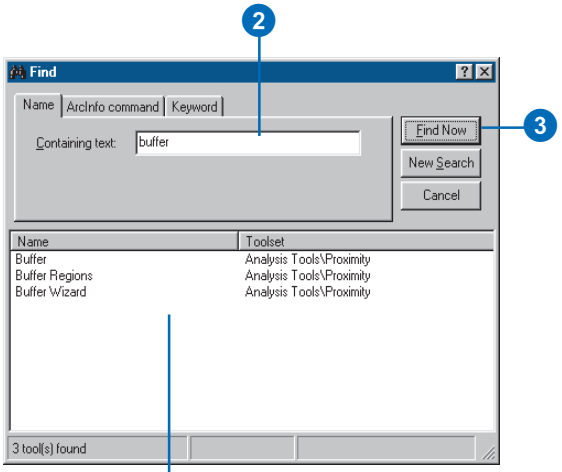
Finding a tool by its name

1. Click Tools and click Find.
2. Type the name of the tool you want to find in the text box.
3. Click Find Now.

   A list of all tools containing the name you typed appears.
4. Double-click a tool to start it.

   The dialog box for the tool appears.





List of tools containing the name you typed in the text box

Tip

Using the Tab key

You can use the Tab key to move the focus on a tool from one ArcToolbox element to another. When a text box has focus, it has an active cursor. A button with focus has a highlighted outline. The element with focus is the one you are currently using.

Tip

Using Help to get more information on a tool

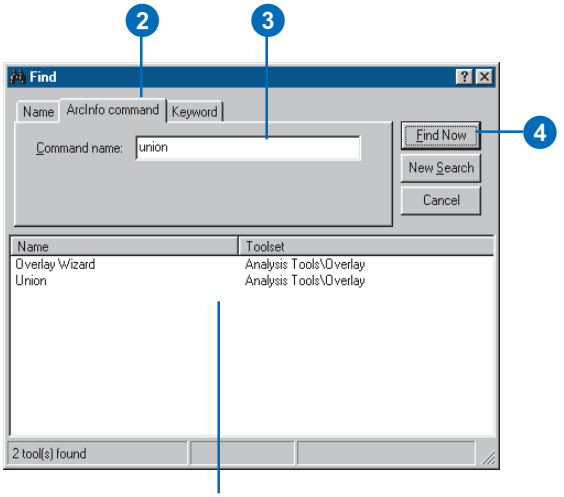
You can use the ArcToolbox online Help system to get information when searching for a tool. When a list of tools appears in the Find dialog box, you can right-click a tool name and click Help. For instructions on how to use Help, see 'Getting help' in this chapter.

Finding a tool using the equivalent ArcInfo command

1. Click the Tools menu and click Find.
2. Click the ArcInfo command tab.
3. Type an ArcInfo command in the Command name text box.
4. Click Find Now.

   A list of tools that run the command you typed appears.
5. Double-click a tool to start it.

   The dialog box for the tool appears.



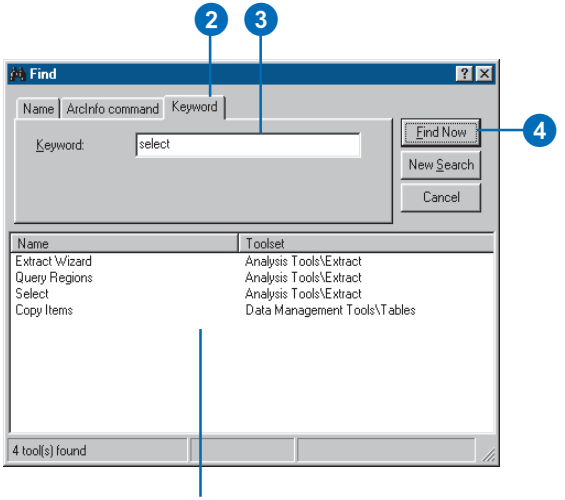
List of tools that run the command you typed in the text box

Finding a tool with a keyword

1. Click the Tools menu and click Find.
2. Click the Keyword tab.
3. Type a keyword in the Keyword text box.
4. Click Find Now.

   A list of tools containing the keyword you typed appears.
5. Double-click a tool to start it.

   The dialog box for the tool appears.



List of tools containing the keyword you typed

Getting help

A quick way to learn about the tools and wizards in ArcToolbox is to use the online Help system. You can get help in ArcToolbox in a variety of ways.

Each toolset in the ArcToolbox tree has a Help node with information about the tools and wizards contained within it. Double-clicking the node displays an overview of what each tool or wizard does.

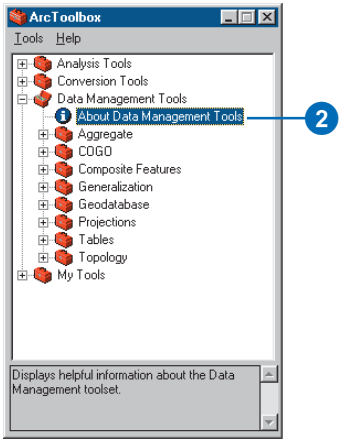
If you want more detailed information about a tool or wizard without actually starting it, you can choose Help from its context menu to open its Help file.

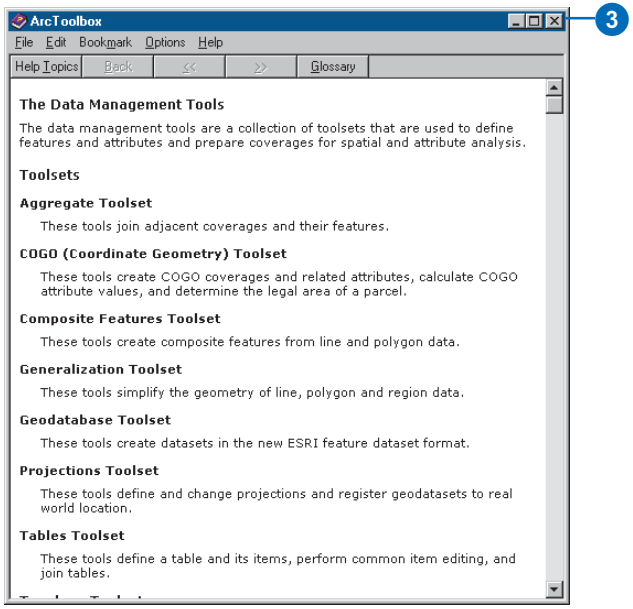
You can also get help in tool dialog boxes. When you click the What's This? button in the upper-right corner and click an item in the dialog box, a description of the item pops up. Tool and wizard dialog boxes also have a second Help button; clicking it opens a Help topic with detailed information about the tool or wizard. ▶

Getting help using Help nodes

1. Navigate to the toolset about which you want information.
2. Double-click the Help node.

   A Help window opens.
3. Click the Close button to close the Help window.





Much of the information in this book is available in the Help system. The Help topics are organized around the main tasks you wish to complete and include some information about the concepts behind the tasks.

You can look up general Help topics on the Contents tab of the Help Topics dialog box. You can search the index for specific tasks and issues. You can also use the Find tab to look up Help topics that have specific words or phrases.

Tip

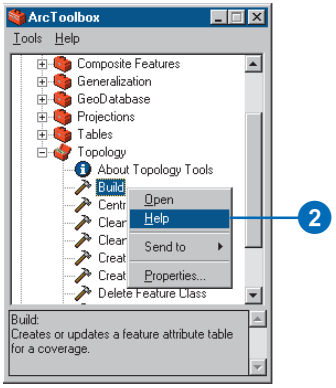
Tips for toolsets, tools, and wizards

When you click a toolset, tool, or wizard, a description of what it does appears in the status bar at the bottom of the ArcToolbox window.

Getting help using a tool or wizard's context menu

1. Navigate to the tool or wizard about which you want detailed information.
2. Right-click the tool or wizard and click Help.

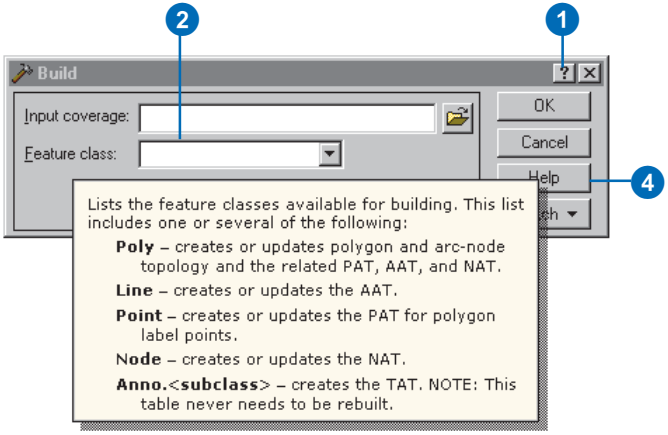
   A Help window with information about the tool or wizard appears.
3. Click the Close button to close the Help window.



Getting help in a dialog box

1. Click the What's This? button in the dialog box.
2. With the Help pointer, click the item in the dialog box about which you want more information.
3. Click anywhere on the screen to close the Help description box.
4. Click the Help button to view more detailed information.

   A Help window with detailed information appears.
5. Click the Close button to close the Help window.



Help window with detailed information about the item

Tip

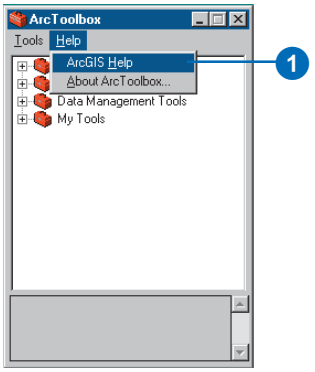
Finding another Help topic

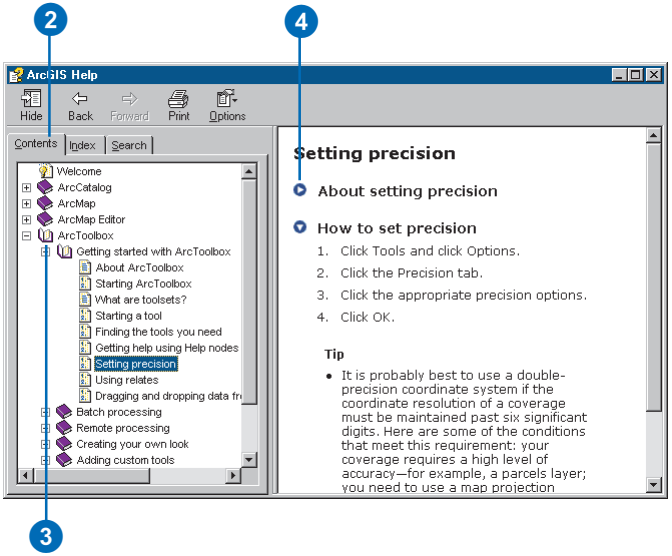
Several topics listed on the Contents tab or in the index may be of interest to you. After reading a topic, click Help Topics at the top of the dialog box to get back to the Help Topics dialog box.

Using the Help Contents to get help

1. Click Help and click ArcGIS Help.
2. Click the Contents tab.
3. Double-click a book to see a list of topics in that category.

   Double-clicking an open book closes its list.
4. Click the topic you want to read.





Tip

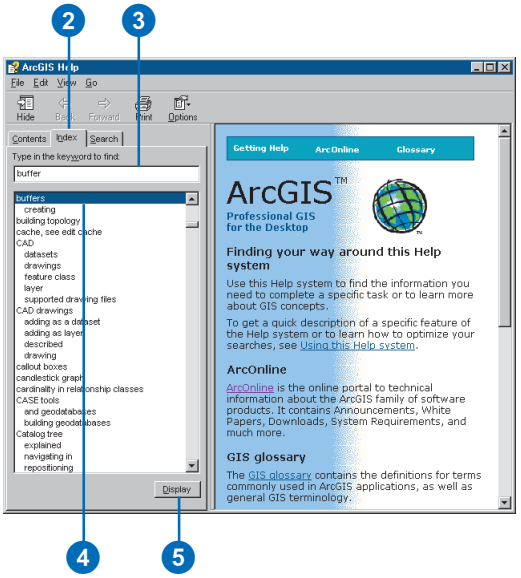
The first time you use Find

The first time you click the Find tab, a setup wizard appears. Choose the default settings and click Finish to close the wizard.

Searching the index for help

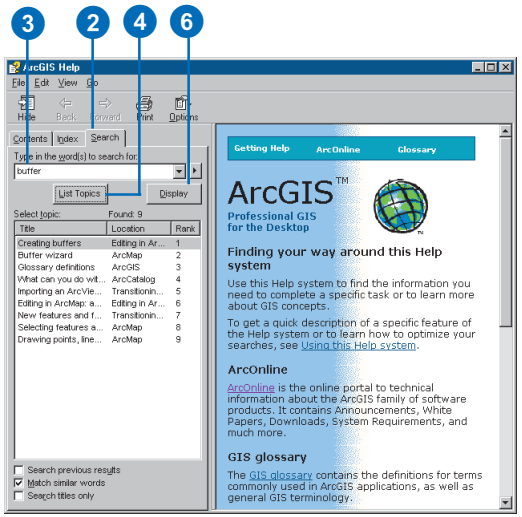
1. Click the Help menu and click ArcGIS Help.
2. Click the Index tab.
3. Type the subject about which you want information.
4. Click the topic you want to read.
5. Click Display.

   If several topics are related to your selection, the Topics Found dialog box appears.
6. Click the topic you want to see.
7. Click Display.



Finding Help topics containing specific words

1. Click the Help menu and click ArcGIS Help.
2. Click the Search tab.
3. Type the word that should be contained in the topics you want to find.
4. Click List Topics.
5. Click the topic you want to see.
6. Click Display.



The precision environment

Coordinate precision refers to the mathematical exactness of a coordinate and is based on the possible number of significant digits that can be stored for each coordinate. ESRI datasets are stored in either single- or double-precision coordinates. Single precision stores up to seven significant digits for each coordinate. This means that values of 4,999,999.6 units to 5,000,000.4 units are rounded off to 5,000,000 units when stored in single precision. However, a value of 5,000,000.6 is rounded off to 5,000,001 units when stored in single precision. This discrepancy can easily be avoided by using a double-precision coordinate system. Double precision stores up to 15 significant digits (typically 13 to 14) and retains a mathematical precision of much less than one meter at a global extent. Note that mathematical precision does not, by itself, define accuracy. It is, however, a major factor in coverage resolution.

The computer can only discern a limited number of decimal places, depending on the precision being used. For single-precision coordinates, the computer assumes that values such as 1.2345678 and 1.23456789 are equal because numbers beyond the seventh digit are ignored. You might imagine a very, very small halo surrounding each coordinate that is equal to the resolution of coordinate storage on the computer. Coordinates with overlapping halos are seen by the computer to represent the same location.

Coverages stored with either single- or double-precision coordinates can be used interchangeably; for example, you can display single- and double-precision coverages over each other, overlay them, merge them, and so on. This capability, however, does not replace the need to be clear and deliberate about how you encode, store, and manage each coverage. Base your decision of which type of precision to use on the desired level of coordinate accuracy to be maintained for each coverage.

The coordinate systems for many map projections use large coordinate values—for example, State Plane and UTM coordinates contain values in the two million to six million range. In these cases, you can use double-precision coordinates to maintain accuracy of less than one unit beyond the decimal point.

As mentioned, double-precision coverages can store up to 15 significant digits. This is sufficient to map any point on the earth to better than a millimeter of accuracy. However, double-precision coverages require additional storage space. Thus, it may be worthwhile to store coverages that require high levels of accuracy (such as parcels) in a double-precision coordinate system and coverages needing less accuracy (such as soils) in a single-precision coordinate system.

The Precision tab in the Options dialog box defines the coordinate precision of new and derived coverages created during an ArcToolbox session. The tab establishes two rules: creation and processing.

The creation rule specifies the precision with which to create all new coverages. Any time a new coverage is created, the coordinate precision of the new coverage is defined by the current creation rule. Single precision is the default. Thus, if you want to create double-precision coverages, you must set the precision of a new dataset to double.

The processing rule specifies the precision with which to create all derived coverages. When a new coverage is derived from one or more existing coverages (for example, the result of the Buffer or Update tools), the coordinate precision of the new coverage reflects the current processing rule. For example, the output coverage precision can be the highest of a set of input coverages. Thus, if all input coverages are in a single-precision coordinate system, the output coverage will be single precision; if at least one of the coverages is double precision, the output coverage is double precision, and so on. To create double-precision coverages regardless of the precision of the input coverages, set the processing rule to double before you begin.

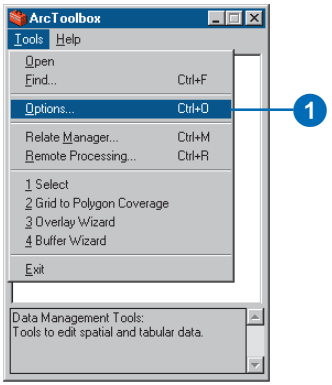
Setting the precision

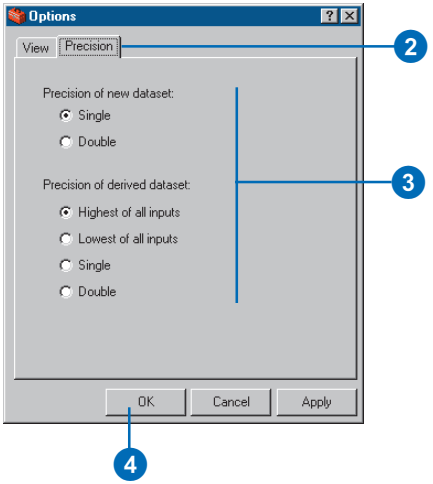
Before you begin using the tools in ArcToolbox, you should determine what the precision of your output coverages should be. Be careful not to use more precision than is necessary, as the higher the precision setting, the more disk space you will need to store it.

It is probably best to use a double-precision coordinate system if the coordinate resolution of a coverage must be maintained past six significant digits. Here are some of the conditions that meet this requirement:

- Your coverage requires a high level of accuracy—for example, a parcels layer.
- You need to use a map projection whose coordinate values exceed the available coordinate precision.
- You are changing datums—for example, from NAD27 to NAD83.

1. Click Tools and click Options.
2. Click the Precision tab.
3. Click the appropriate precision options.
4. Click OK.





Using relates

The ability to store related information in tables is an example of the flexibility of ArcToobox. Attribute data can be stored in feature attribute tables, related files, or external *database management system (DBMS)* tables.

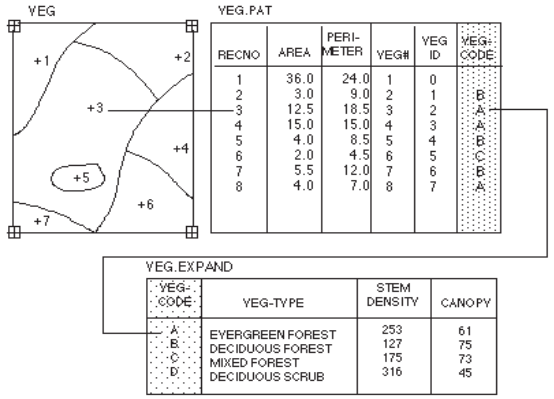
You can also temporarily associate additional attribute tables with a coverage's feature attribute table using an operation known as a relate. A relate makes a connection between a record in the feature attribute table and a corresponding record in the related attribute table. An item in one table is used as a relate key to a corresponding item or column in the related table.

The relate is a defined and named *relational join* between feature attribute tables, INFO files, and external DBMS tables. The relate environment exploits all the advantages of a relational data structure. In ArcToolbox, a relate is a named relationship between an item appearing in a feature attribute table (or many tables) and a related INFO datafile or external attribute table. A relate consists of:

- A relation name—used to identify and access the relate.
- A table identifier—identifies the related table to be accessed. This can be an INFO file, table, or view in an external DBMS.
- A database name—identifies the DBMS in which the table is stored (for example, INFO or a connection to one of the supported external DBMS systems).
- An item—item in the feature attribute table or INFO file that relates to the related table.
- A relate column—the item or column name in the related attribute table that relates back to the feature attribute table or INFO file item.
- A relate type—specifies the type of connection made and depends on the database being interfaced. For INFO access, type can be Linear, Ordered, or Link; for DBMS tables, it must be First.
- A relate access—specifies the mode of access (for example, RW, RO, or AUTO).

No specific source table name is stored as part of the relate; relates are item based, not table based. This way, the same relate can be used to relate more than one feature attribute table to the related table.

To establish the relate environment, use the Relate Manager. Once a relate is active, the related items are displayed in all lists of available items when the relate is applicable to the currently selected table or coverage feature class. For example, if a relate is defined for a standalone INFO table that relates to a coverage's arc attribute table (AAT) and the Query Builder is used to select records from that AAT, the list of available items will display related items contained in that standalone INFO table.



An example of a simple relate

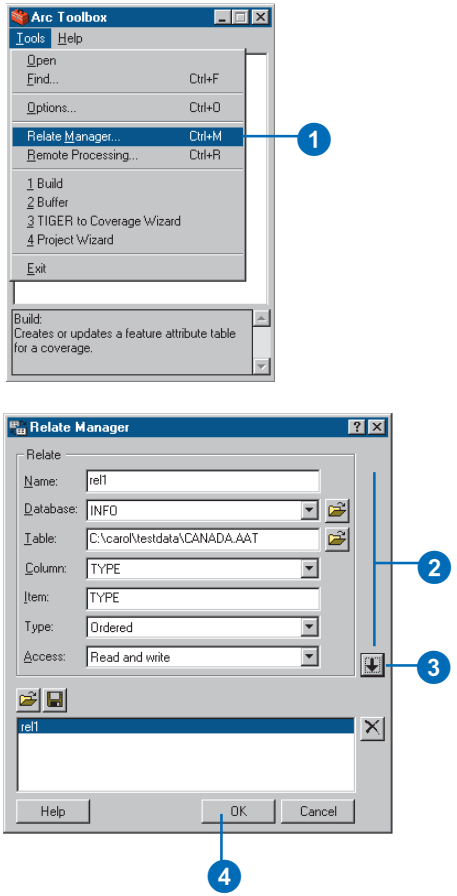
Defining relates

Related tables are accessed by the Query Builder. This is used by a number of tools that offer the option of an attribute selection such as the Select tool or Extract Wizard. Your relate is maintained by ArcToolbox, so you need only define the relate once.

You can define up to 100 different relates as part of the relate environment at any time. However, only 30 can be used simultaneously during an operation. Once the relate environment contains 100 relates, you can't replace a relate with a new relate. However, you can either update the parameters of an existing relate or delete a relate and define another.

Defining a relate

1. Click Tools and click Relate Manager.
2. Fill in the fields with the required information.
3. Click the Add button to finish the relate and make it active.
4. Click OK.



Saving and restoring relates

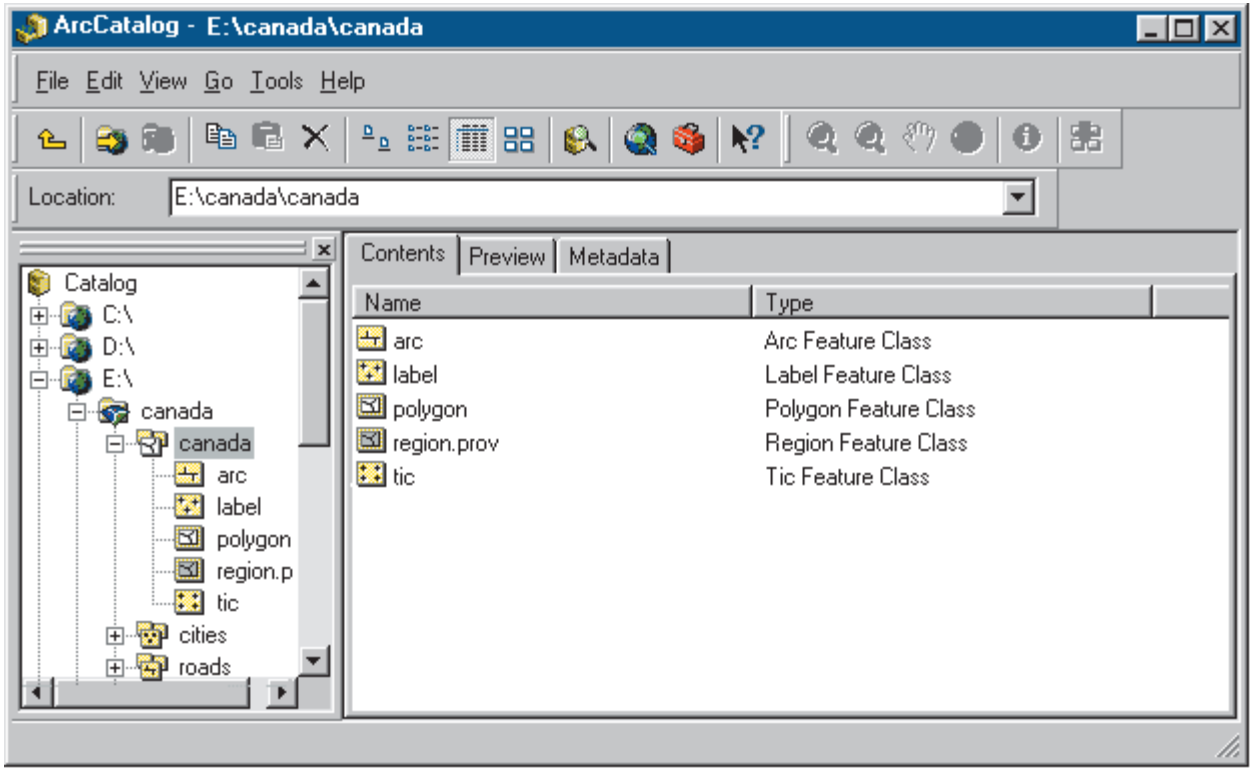
1. Click Tools and click Relate Manager.
2. Fill in the fields with the required information.
3. Click the Add button to finish the relate and make it active.
4. Click the Save button to open a browser.
5. Type a filename and click OK.
6. Click the Open button and click the file just saved.
7. Click OK.

Using ArcCatalog

ArcCatalog helps you browse, find, and manage data. You can search for the data you wish to use and preview it. ArcCatalog is the ideal application for managing your datasets and works in conjunction with ArcToolbox.

ArcToolbox provides browsers for all inputs and outputs of each tool, but you may find it easier to locate data and manage your projects using ArcCatalog. Once you have found your dataset in ArcCatalog, you can drag it to the appropriate text box in an ArcToolbox tool using your mouse.

You can start ArcToolbox from ArcCatalog using the Launch ArcToolbox button on the Standard toolbar. Data resulting from operations performed with ArcToolbox tools is available in ArcCatalog; you can see it once a tool has successfully completed its task and the ArcCatalog tree has been refreshed.



ArcCatalog provides a different approach to managing your data. ArcToolbox works in conjunction with ArcCatalog, providing an easy and efficient way to work.

Dragging and dropping data

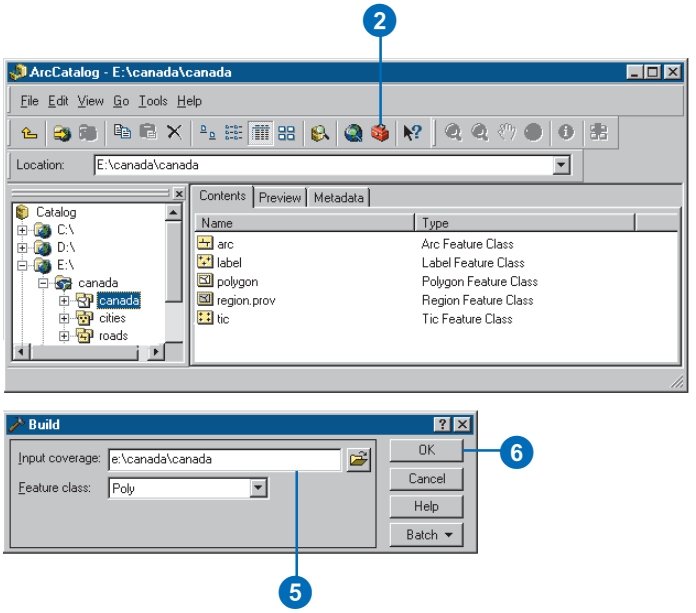
With ArcCatalog, you can easily locate datasets you wish to use for geoprocessing in ArcToolbox. One way of moving the data from ArcCatalog to ArcToolbox is by dragging and dropping it. To drag and drop a dataset, simply click the dataset in ArcCatalog and drag it to the tool while holding down the left mouse button. Release the mouse button and drop the dataset on the correct text box. Dragging and dropping more than one dataset from ArcCatalog to a tool opens the tool's batch table, allowing the tool to run in batch mode and process all datasets at once.

Tip

Using Windows Explorer

Datasets and files can also be dragged onto tools from Windows Explorer when the required access is read-only. For example, you can drag a coverage folder onto a text box on the Union tool; however, you can't drag a coverage from Windows Explorer onto the Build tool, as the tool requires write access to the coverage. Multiple datasets and files can also be dragged and dropped from Windows Explorer.

1. Start ArcCatalog from the Start menu.
2. Click the Launch ArcToolbox button to start ArcToolbox if it is not already started.
3. In the ArcToolbox tree, double-click a tool to start it.
4. In ArcCatalog, navigate to a dataset you wish to move.
5. Click and drag the dataset and drop it on the input text box of the tool.
6. Click OK to run the tool.



Operating tools in batch mode

You can run multiple jobs of a tool at once using a tool's *batch mode* capability. When batch mode is activated, the tool expands to show a table displaying the parameters of each job.

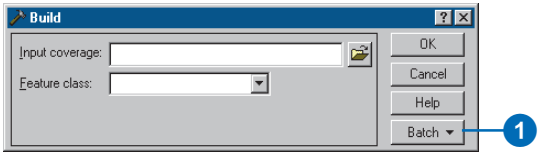
The *batch table* can be activated in several ways. The first is to activate it on the tool itself. You can also activate a batch table automatically when you drag and drop multiple records onto a tool from ArcCatalog or select multiple input datasets from a browser.

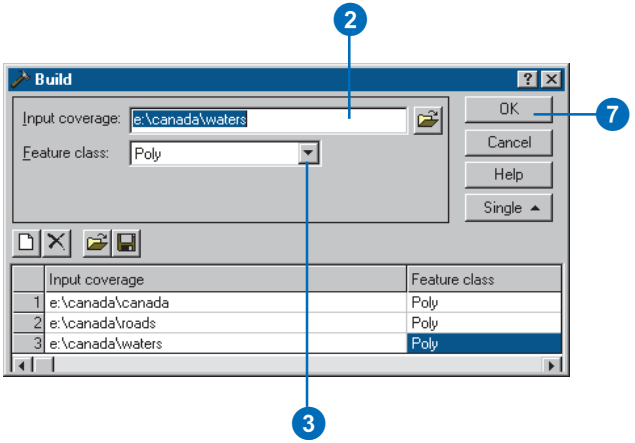
The Build tool is used in this task as an example of the batch mode capability applicable to every tool in ArcToolbox. Keep in mind that although the tools look different from one another, batch mode works the same for all of them.

You may not always want to run the tool after you have populated the batch table. You may prefer to run the batch job at another time such as at the end of the day when your system is not as busy. You may want to stop working on a tool and come back later to finish populating the table. The batch table makes this possible by ▸

Starting the batch table from the tool

1. Start the tool from ArcToolbox. Click Batch to open the batch table.
2. Type a dataset name to populate the first row in the table.
3. If necessary, change the default parameters by clicking the Feature class dropdown arrow and clicking a new feature class.
4. Click the Add Row button to create a new row in the batch table.
5. Repeat steps 2 and 3 to populate the second row of the batch table.
6. Repeat the steps above until all the datasets you want to use have been added to the table.
7. Click OK to run the tool.





letting you save its contents to an AML script.

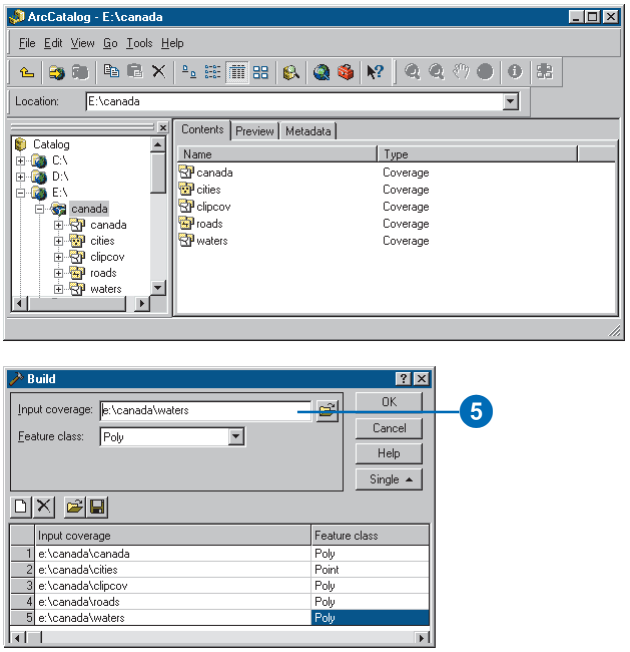
You can edit the AML scripts created by ArcToolbox, but this is not recommended as it may corrupt them. Rather, you should append a script by saving to one that already exists. In so doing, you can compose an AML that performs many tasks.

You can load an AML that has parameters from numerous tools, but only the valid parameters for the current tool are loaded. When you save to an AML, the tool's parameters are always appended to the contents of the script if it already exists. For example, suppose an AML has entries from the Build and Clean tools. When it is loaded into the Build tool, only the entries pertaining to the Build tool are used, and the rest are left out.

Within the My Tools toolset is a tool that can run any AML that starts at the Arc: prompt. You can run any existing AML in your organization or one you have created using a tool.

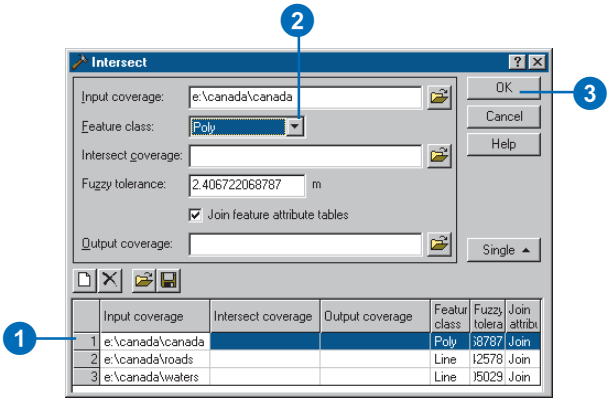
Starting the batch table from ArcCatalog

1. In ArcCatalog, click the first dataset you want to process.
2. Hold the Shift key down and click the other datasets you want to process.
3. Double-click the tool in the ArcToolbox tree to start it.
4. Arrange the ArcCatalog window and the tool dialog box so that both are showing.
5. Drag and drop the selected items from ArcCatalog onto the input text box of the tool.
6. Click OK to run the tool.



Editing the batch table

1. Click the row in the batch table whose parameters you want to edit.
2. If necessary, change the default parameters by clicking the Feature class dropdown arrow and clicking a new feature class.
3. Click OK when the batch table is complete to run the tool.



Tip

Closing the batch table

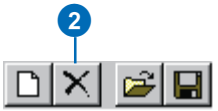
You can close the batch table in a tool by clicking the Single button in the tool's dialog box.

Tip

Saving your work

By creating a script each time you run a tool in batch mode, you create a log of what you have done. This log, in the form of an AML script, will help you if you need to run the tool again or remind yourself of the tools and parameter values used to create a dataset. This may not be necessary, but it is an effective way of logging your work.

Deleting a row from the batch table



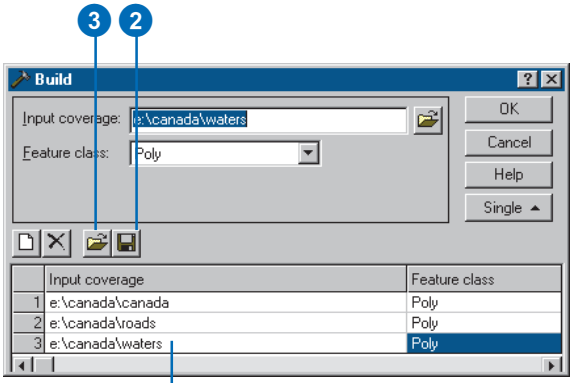
1. Click the row you want to delete from the table.
2. Click the Delete Row button.

   The row is deleted from the batch table.

Saving batch table contents to a script

1. Follow steps 1 through 7 for starting a batch table from a tool.
2. Click the Save To Script button to save the contents of your batch table to a script.
3. Click the Open button and click a script that already exists for that tool. Click Save.

   The batch table is populated with the script's parameters.



The batch table is populated.

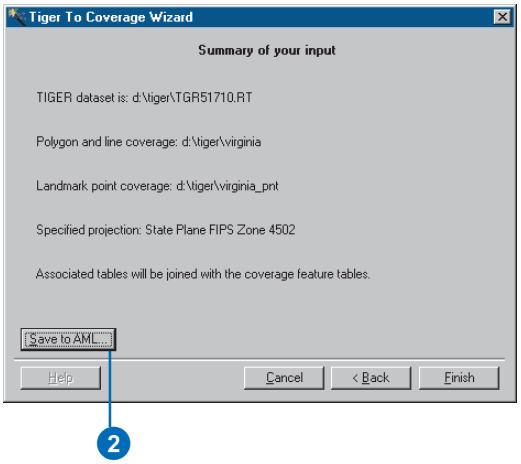
Saving a wizard's contents

The summary panel at the end of each wizard has a button that lets you save the contents of the wizard to an AML script. After you have gone through all the panels of the wizard, you can click Save to AML to save the wizard's contents for later use.

This button also lets you create a batch script that will run more than one job with the wizard. After you've saved the contents of the wizard to an AML script, you can save subsequent inputs to the same script.

You can't import or edit a script while using a wizard. Be sure about the parameters you enter before you save. You can always go back to previous steps in a wizard to check your input and fix any mistakes.

1. Follow the steps on each panel of the wizard.
2. Once you reach the summary panel, click the Save to AML button.
3. Type the name of an AML to create or click an existing AML in the browser.
4. Click OK.



The Geoprocessing Server

ArcToolbox can perform remote processing on another machine that has been set up as a *Geoprocessing Server*; this machine can be either a Windows NT or UNIX system. The Geoprocessing Server is installed separately from ArcToolbox and runs as a separate application. For complete information about the installation of a Geoprocessing Server and how to enable Geoprocessing Server client desktops, see Appendix B, the 'Geoprocessing Server Administration Guide'.

In ArcToolbox, you can connect to any machine that has been activated as a server. After connecting to a server, you can specify whether or not you want tasks to run remotely. This front- and back-office approach lets you compose tasks on your desktop but process the data where it resides, creating a more efficient work environment.

Before you can process *jobs* remotely, you must add a server name to a list of available servers. You must provide and verify connection information before the server can become active on a client desktop. Once a server is activated and chosen, all of the ArcToolbox tasks run on the remote server. All subsequent tasks run by ArcToolbox will use the remote server. You can't run some tasks locally and others remotely without changing the processing state.

When you connect to a Geoprocessing Server, you must specify the Geoprocessing Server instance running on the server you want to use. An instance is one installation of the Geoprocessing Server running on a host machine. Multiple Geoprocessing Server instances may exist on one server, so an instance name identifies the installation you want to use. Your desktop computer can communicate with an instance once the instance name and its communication information is specified in your Windows services file.

The Geoprocessing Server manages the jobs you post to the server and sends you the results of the job when it is complete. Typically, the job order is decided by the first in, first out rule, but you can also specify when a job should run using the scheduling option of the server. You may schedule your jobs to maximize efficiency on your server and complete tasks when their results are needed.

Only AML scripts that execute nongraphic commands available in the ARC module are supported by the Geoprocessing Server. This means the AML you use can't switch from one module to another. For example, you can't run an AML that starts in ARC and then switches to ARCPLOT™ software to start a display canvas. Using the Run AML Script tool, you can run existing AMLs from ArcToolbox and have them processed anywhere you like.

The following pages show you how to connect to a Geoprocessing Server, send and schedule jobs, and check for results.

Using a Geoprocessing Server

Before you can submit a job to a Geoprocessing Server, you must set your processing environment and create a connection to a Geoprocessing Server instance. Once this is done, all jobs will be submitted to the server.

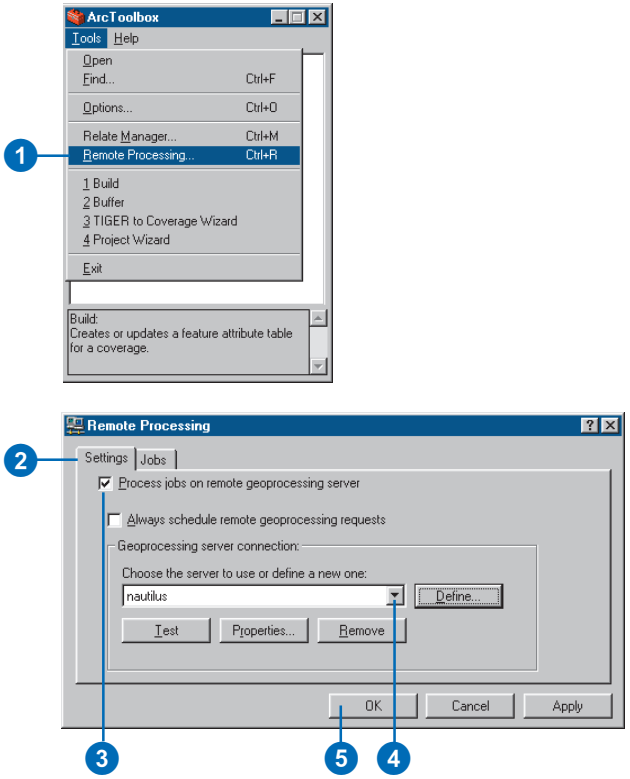
Tip

Where is your data?

If your data is located on a remote system that has superior resources, such as a faster CPU, make it a Geoprocessing Server and process your data there instead of on a desktop client. This is a great way to take advantage of your powerful UNIX system.

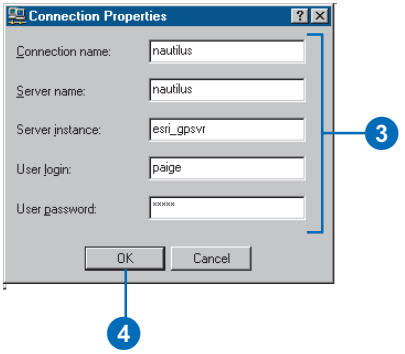
Setting your processing environment

1. Click Tools and click Remote Processing.
2. Click the Settings tab.
3. Check the first check box to process jobs on a remote Geoprocessing Server.
4. Click the dropdown arrow and click a Geoprocessing Server. If the list is empty, follow the steps below to connect to a server.
5. Click OK.



Connecting to a Geoprocessing Server

1. Click the Tools menu and click Remote Processing.
2. Click Define.
3. Type the connection information in the text boxes.
4. Click OK.



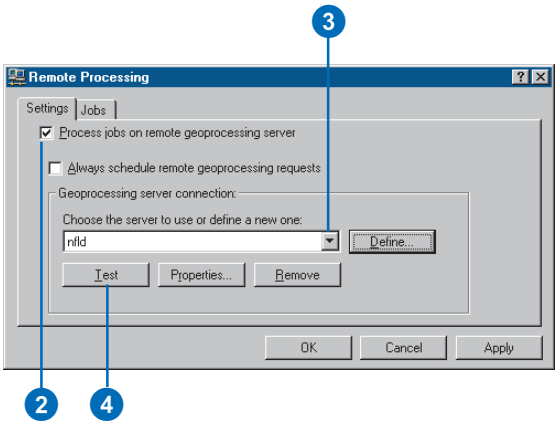
Connection properties

Once you have specified the properties of a Geoprocessing Server connection, you can test the connection to ensure the connection is valid. If one or more of the connection properties are incorrect, you can change them using the Connection Properties dialog box.

Testing your connection

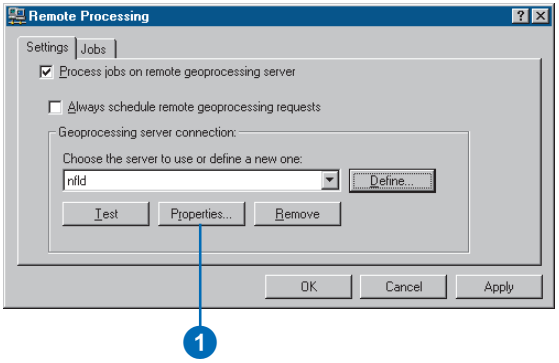
1. Click the Tools menu and click Remote Processing.
2. Check the first check box to Process jobs on a remote Geoprocessing Server.
3. Click the dropdown arrow and click a Geoprocessing Server. If the list is empty, follow the steps for connecting to a Geoprocessing Server to define a server.
4. Click Test.

   The cursor will change to an hourglass while the test is being executed. The connection is valid if the cursor changes back to the default and the test button becomes disabled. If the connection is invalid an error message describing the problem will be displayed.
5. Click OK.



Changing properties

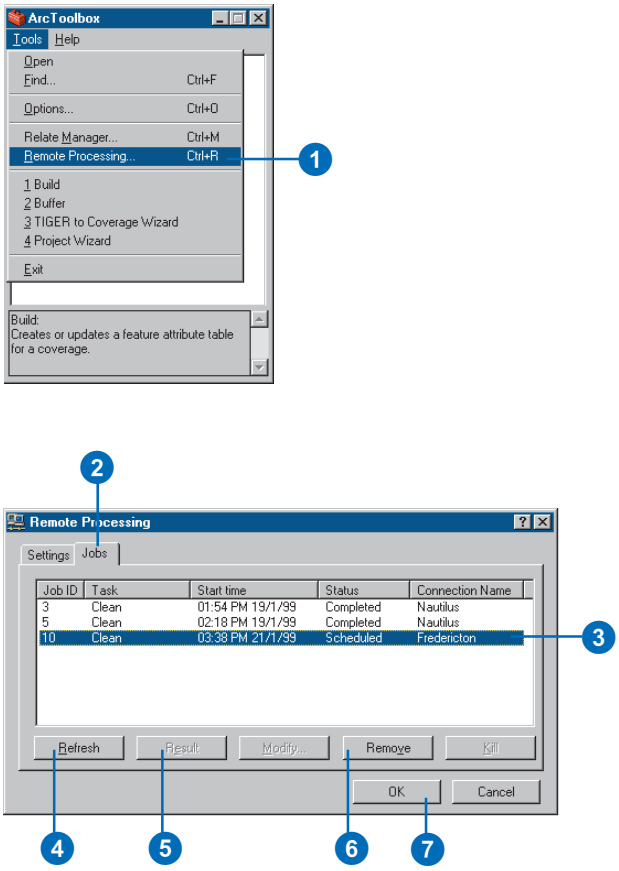
1. Click Properties.
2. Edit the properties of the currently selected connection. A password is always required when defining connection properties.
3. Click OK.
4. Click Test to validate the connection.



Checking a remote process

Once a job has been sent to a Geoprocessing Server from a tool or wizard, the client can request the status of the job. When you send a job to the server, the server gives you a job ID to identify the job. When you check the status of a job, the Geoprocessing Server tells you whether the job is scheduled, is processing, was completed, or has failed. The status of each job submitted to the server is displayed in a dynamic list started from ArcToolbox. Once a job has been completed, you can view the messages that occurred during the job's processing. Completed job records can then be deleted from the list in order to keep the list current.

1. Click Tools and click Remote Processing.
2. Click the Jobs tab.
3. Click any scheduled job in the list.
4. Click Refresh to update the list. Continue to periodically update the list until the job is completed.
5. Click Result to see the messages from the server regarding the job.
6. Click Remove to remove the entry from the list.
7. Click OK.



Scheduling a remote process

Once you choose to schedule all of your remote processes, you are asked to submit a time when the job will be sent to the Geoprocessing Server. Once a remotely scheduled process is submitted, the server manages its schedule and reports the status of the process back to the ArcToolbox client.

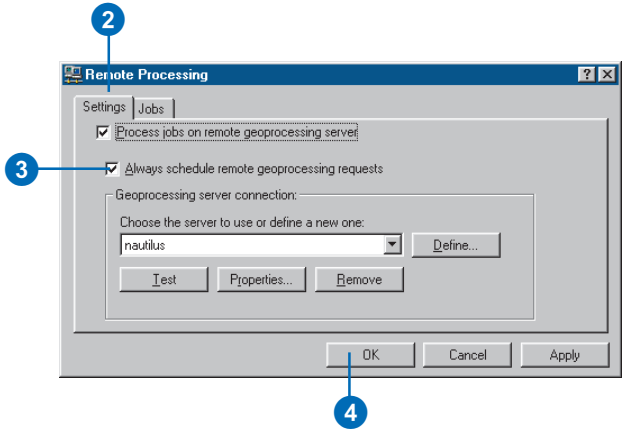
Tip

Synchronize your clocks

It is important to synchronize the clocks on the server and the client machines in order to accurately schedule remote processes. Differences in time may cause confusion, especially when multiple people are trying to schedule jobs during assigned times.

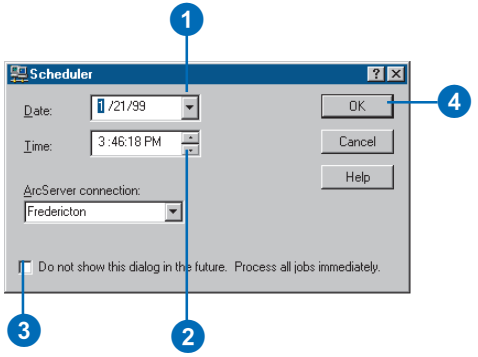
Activating scheduling

1. Click Tools and click Remote Processing.
2. Click the Settings tab.
3. Check the second check box to Always schedule remote Geoprocessing requests.
4. Click OK.



Scheduling a job

1. Click the Date dropdown arrow and click a date. The current date is the default.
2. Click the up and down arrows on the Time text box or type the desired start time. (You must select the hour, minute, or second sections first when using the arrows.)
3. If you don't want to schedule future jobs, check the check box so that the dialog box is not shown in the future and all jobs are processed immediately.
4. Click OK.



Creating your own look

4

IN THIS CHAPTER

- Turning off toolsets and tools
- Turning off Help nodes and tool descriptions
- Adding to the My Tools toolset
- Placing tools on top

As you use ArcToolbox, you may notice that you are only using tools from a few specific toolsets or that you are using tools from numerous toolsets to complete a routine task. You may not want to navigate through all of these toolsets to find the tools you commonly use. To make your work easier, ArcToolbox offers a way to personalize the organization of your tools. You can customize the ArcToolbox tree so that only the toolsets and tools you need are visible, thereby creating your own look and feel.

In Chapter 2, you were introduced to the My Tools toolset and learned how it can serve as your custom work area. In this chapter, you will create custom toolsets and add preexisting tools. You will also learn how to change the choices available in the Data Management Tools, Analysis Tools, and Conversion Tools toolsets, eliminating those tools you do not use. This will help you find the tools you want more efficiently.

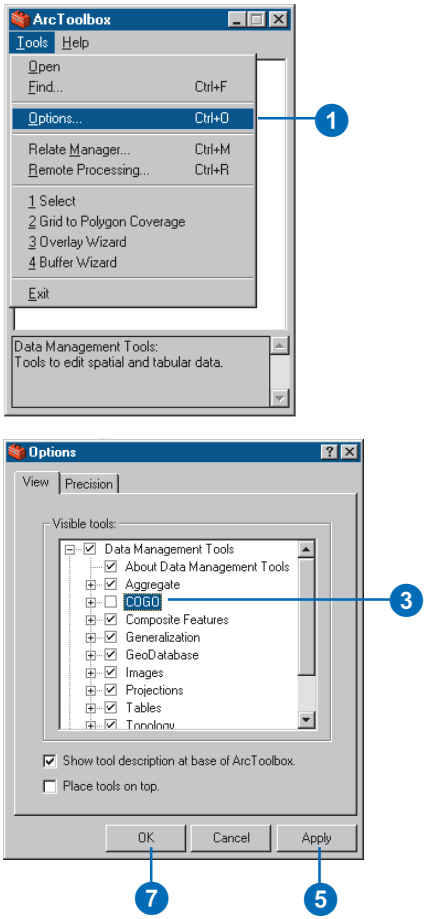
Turning off toolsets and tools

To simplify your work with ArcToolbox, you can turn off toolsets and tools in the first three toolsets in the ArcToolbox tree. (The My Tools toolset is already completely customizable.) You may want to turn off unwanted or infrequently used tools so that they don't clutter the tree.

This feature is also very helpful when you want to create a custom interface for performing specific tasks. You can control what users can do with ArcToolbox by customizing the interface based on the tasks each individual is assigned to do.

If you are an administrator, you now have more control over how the software is used. You can turn off a tool when you don't want it available to users. For example, if you don't want your data cleaned, you can turn off the Clean tool.

1. Click Tools and click Options.
2. Navigate to the toolset or tool you want to turn off.
3. Uncheck the toolset or tool to turn it off.
4. Repeat steps 2 and 3 until you are finished turning off all of the toolsets and tools you don't want.
5. Click Apply to apply the changes without closing the Options dialog box.
6. Verify the changes you made in the ArcToolbox tree.
7. Click OK to close the Options dialog box.



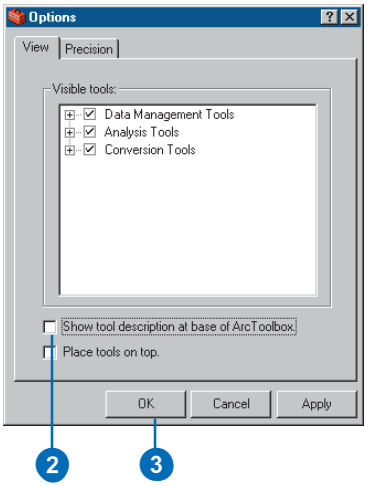
Turning off tool descriptions and Help nodes

After using ArcToolbox for a while, you'll learn what the tools do, the purpose of each toolset, and what tools they contain. You may want to reduce the space needed to display this information, giving you more space on your screen.

Help nodes and tool descriptions are turned on by default to help you become familiar with ArcToolbox. By turning off the tool descriptions, you eliminate the text at the bottom of the ArcToolbox window that describes the item currently selected in the ArcToolbox tree. When Help nodes are turned off, only the tools or toolsets contained in each main toolset are displayed.

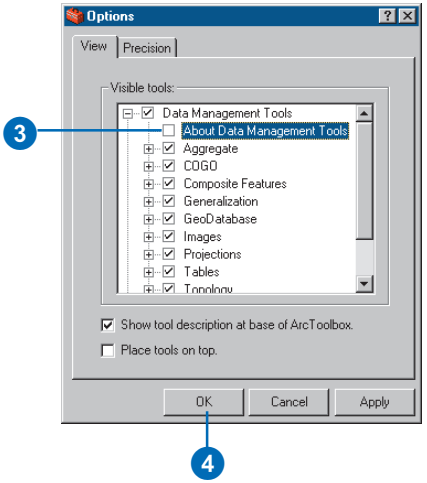
Turning off tool descriptions

1. Click the Tools menu and click Options.
2. Uncheck the the first option to turn off the tool description at the base of ArcToolbox.
3. Click OK to apply the property and close the dialog box.



Turning off Help nodes

1. Click the Tools menu and click Options.
2. Navigate to the Help node you want to turn off.
3. Uncheck the Help node to turn it off.
4. Click OK to apply the property and close the dialog box.



Adding to the My Tools toolset

As discussed in Chapter 3, the My Tools toolset is designed to be customized to simplify your work. This involves adding frequently used tools or tools associated with a common task. An easy way to add tools to the My Tools toolset—or to a custom toolset you create within it—is to send them from their original location in the ArcToolbox tree. This is done using the Send to command in the tool's context menu.

The context menu initially gives you the option of sending tools only to the My Tools toolset. However, you can also create custom toolsets within My Tools to organize the tools you use most frequently. You can then send the tools to the appropriate toolset. Once added to My Tools, the custom toolsets' names appear automatically in the Send to list.

Tip

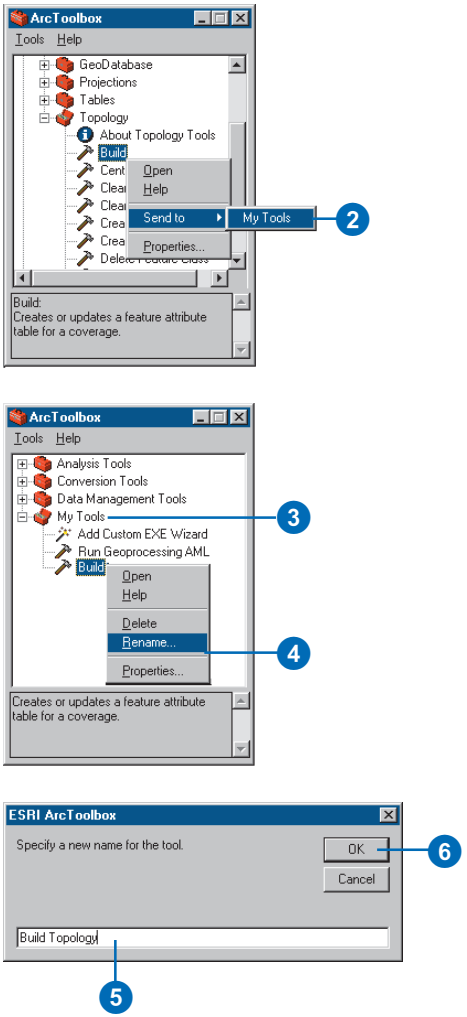
Deleting a tool or custom toolset from My Tools

To delete a tool or custom toolset, right-click the tool or toolset and click Delete.

Sending tools to the My Tools toolset

1. Navigate to the tool you want to send.
2. Right-click the tool, point to Send to, and click My Tools.
3. Double-click My Tools.
4. Right-click the tool you sent and click Rename.
5. Type a name describing the tool.
6. Click OK.

   The tool appears under My Tools.



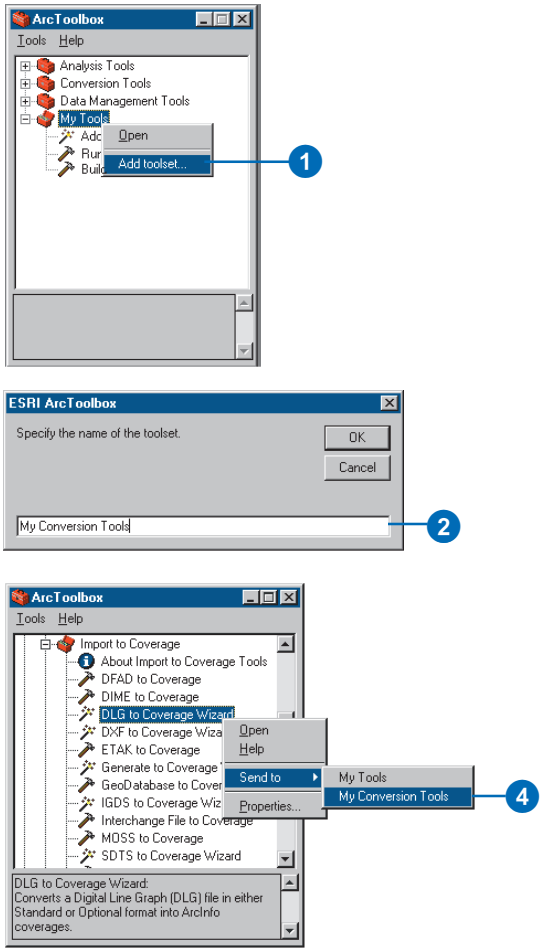
Tip

What's in a name?

Give your custom toolsets names that clearly specify the types of tools found within them. You can use up to 66 characters in the name, so use as many words as it takes to clearly describe the toolset's purpose.

Adding a custom toolset

1. Right-click My Tools and click Add toolset.
2. Type a name for the custom toolset and click OK.
3. Navigate to the tool you want to add to the custom toolset.
4. Right-click the tool, point to Send to, and click the custom toolset to which you want to add the tool.
5. Double-click the My Tools toolset.
6. Double-click the custom toolset to see the tool you added.

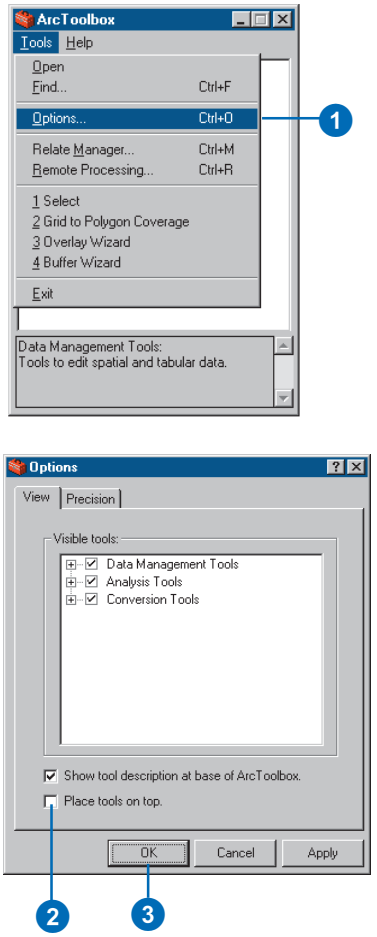


Placing tools on top

Sometimes you need to use several dialog boxes and windows at the same time—for example, when you are dragging and dropping datasets from ArcCatalog onto a tool from ArcToolbox. Often when you are doing this, tool dialog boxes get lost behind windows of other applications, especially when you have a small screen.

ArcToolbox has an option to place tools on top to make these tasks easier. Once this property is set and applied, all tool dialog boxes remain "on top" of windows from other applications. However, it's important to set this property before you start using a tool as it won't affect tools that have already been started.

1. Click Tools and click Options.
2. Check Place tools on top.
3. Click OK to apply the property and close the dialog box.



Adding a custom tool

5

IN THIS CHAPTER

- Requirements for a custom tool
- Creating a custom tool in Visual Basic 6.0
- Enabling drag and drop
- Adding a custom tool

Although ArcToolbox provides a long list of tools, you may want to add to its capabilities so that you can perform additional tasks. You can do so by developing and installing new tools to meet your requirements.

A custom tool can be a system *executable (EXE)* or a class contained in a *dynamic link library (DLL)*. The executable tool can run any application that runs on the platform you're using. This could be a word processor, spreadsheet, or *computer-aided design (CAD)* application that supplements your use of ArcToolbox. If you have a DLL, it must meet some simple requirements in order for it to work.

In this chapter, you will learn how to create a very simple custom tool and add it to ArcToolbox. The example in this chapter uses Visual Basic 6.0 as the development environment, but you could use any developer environment, such as Visual C++®, Delphi®, or Sybase® PowerBuilder®, to construct a custom tool EXE or DLL.

The custom tool must be fully self-contained. ArcToolbox provides only the Arc automation server, which allows access to the full range of functions in the ARC module. The tool must handle all validation, execution, and error checking.

Requirements for a custom tool

When creating and adding custom tools, certain requirements must be met. These requirements depend on the type of custom tool you're working with: DLL or EXE.

DLL

Every DLL that is installed as a custom tool must have a class that implements the IArcToolBoxTool Component Object Model (COM) interface. This interface provides ArcToolbox with the information it needs to store and execute the tool from the ArcToolbox tree. Custom tools must also support the IArcToolboxFind interface if the tool is to be supported by the ArcToolbox Find tool. The IArcToolboxFind interface provides information required by the Find tool such as keywords and what ARC commands are used. If the custom tool references the ODE Arc automation server, then it should implement the IArcToolBoxODE interface so it can use the Arc automation server defined in the ArcToolbox application—this saves resources and improves performance. You can learn more about these interfaces by using the ESRI Object browser or the VB Object Browser.

The class can be called whatever you want. For an example of a class definition that meets the above requirements, refer to the example in Appendix A. In this example, a VB form called frmDescribe is opened from inside the class.

Executables

There are no requirements for executables used as ArcToolbox custom tools, except that the executable must operate in the current operating environment and handle its own errors and validation. For example, you could add ArcCatalog as a custom tool simply by using the executable found in its installation directory.

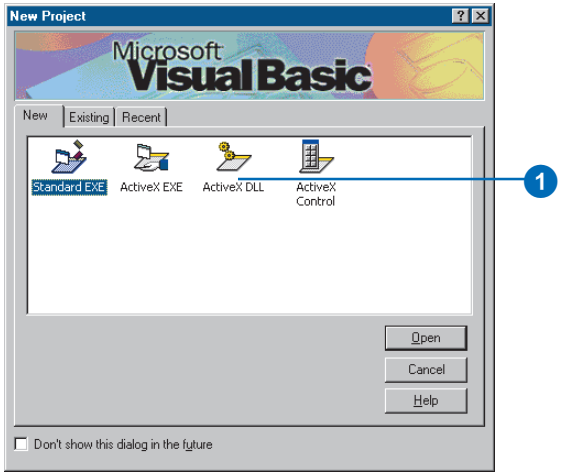
Creating a custom tool in Visual Basic 6.0

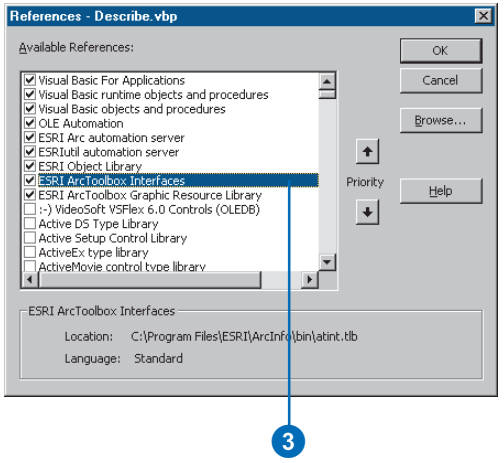
You can create a custom tool for ArcToolbox using Visual Basic 6.0. This involves calling a VB form from a VB class. In this example, the form has a text box for the name of a dataset, which is then described using the ArcInfo Describe command. The results are then displayed in a list box on the form.

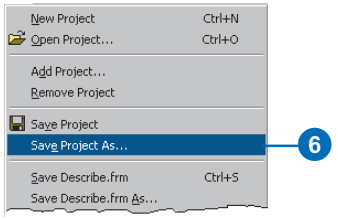
The sample tool in this example uses ArcInfo ODE, but you are not limited to this type of tool. You can create any type of tool you want using third-party *ActiveX* technology. You can build a report writer, CAD viewer, or anything else that may help you use ArcToolbox.

Creating a VB class

1. Open Visual Basic 6.0 and double-click ActiveX DLL.
2. Click the Project menu and click References.
3. In the References dialog box, check ESRI Arc automation server, ESRIUtil automation server, ESRI Object Library, ESRI ArcToolbox Graphic Resource Library, and ESRI ArcToolbox Interfaces. Click OK.
4. Click the Name property for the class module and type “clsDescribe” as its new name.
5. See Appendix A for Visual Basic code for the class. Type the code or copy and paste it from the online Help system. Add the code to the code window.
6. Click the File menu and click Save Project As. Type “describe.cls” to name the class. Click Save.







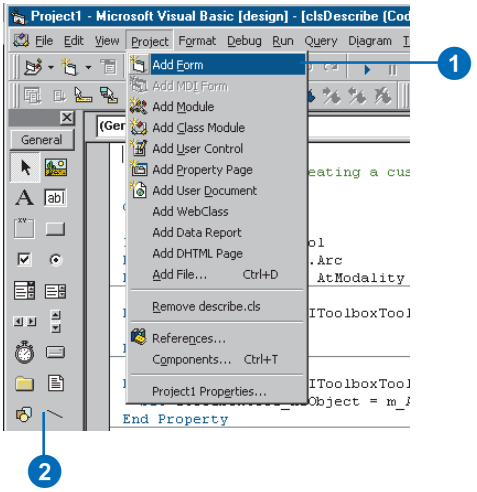
See Also

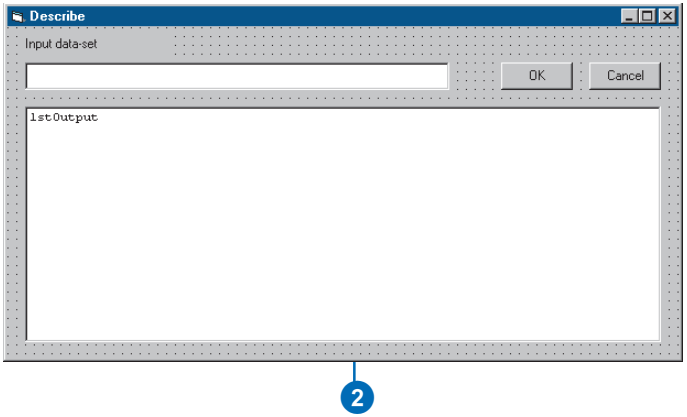
For more information on developing custom tools with ArcInfo ODE, see Working with the ArcInfo Open Development Environment: ActiveX Controls and Automation Servers for Windows NT Developers.

Creating a tool form

1. In Visual Basic, click Project and click Add Form. Add a regular form and set its name property to “frmDescribe”. Set the caption property to “Describe”.
2. Add a text box, list box, label, and two button controls by clicking the appropriate buttons on the General toolbar and placing them on the tool form.
3. Name the controls as follows:

   list1 = lstOutput
   command1 = cmdOK
   command2 = cmdCancel
   label1 = lblInput
   text1 = txtInput
4. Type “OK” into the caption property of cmdOK and type “Cancel” in the caption property of cmdCancel.
5. Make the text property of the txtInput text box blank and type “Input dataset:” in the caption property of lblInput. ►



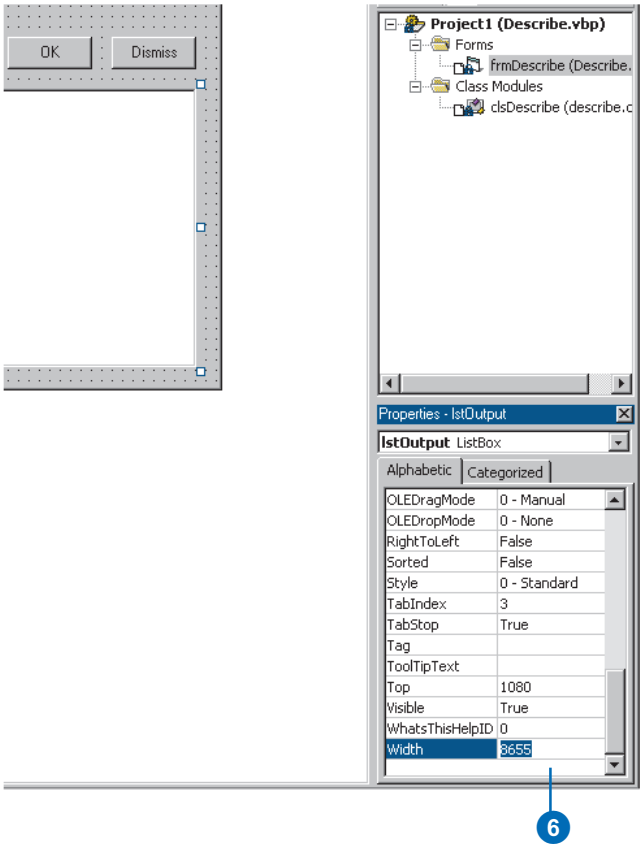


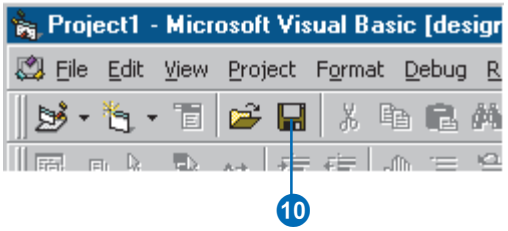
Tip

Naming controls

When naming your VB controls, use prefixes to help distinguish the type of control being used. For example, all text boxes on a form would start with "txt". You should choose appropriate prefixes and use them consistently.

6. Click the Alphabetic tab in the Properties - lstOutput dialog box. Type "8655" for the Width field. Change the font to 8 pt. Courier New. Widen the form to accommodate the size of lstOutput by simply dragging the sides of the form or by setting the form's width property.
7. Click the File menu and click Save frmDescribe As. Type "describe.frm" as the form file name.
8. Click the File menu and click Save Project As. Type "Describe" in the Project File box.
9. Refer to Appendix A for Visual Basic code for the form. You can also copy and paste the code from the online Help system.
10. Click the Save button and save the form and the project. ►

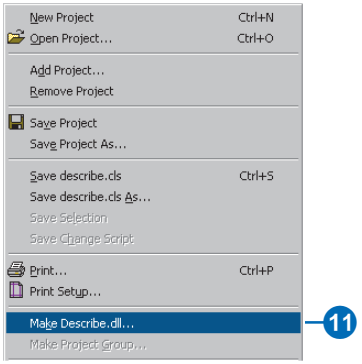




11. Click the File menu and click Make Describe.dll.

    If errors occur, review the code to make sure it exactly matches what is in Appendix A. Create the DLL in the same location as your project.

12. Exit or minimize Visual Basic.



Enabling drag and drop

All ArcToolbox tools accept data that is dragged and dropped from ArcCatalog or Windows Explorer. Your custom tool should also offer this feature, as it's an easy and efficient method of adding input to tools. You can add the drag-and-drop capability to your custom tool using a VB form.

Unlike Windows Explorer, ArcCatalog doesn't simply provide a string containing a dataset's path. Instead, it provides an object that contains the path and its type.

Due to this difference, enabling drag and drop from ArcCatalog requires more from you such as handling the type of object being passed to a tool. ►

Tip

Validating your input and output

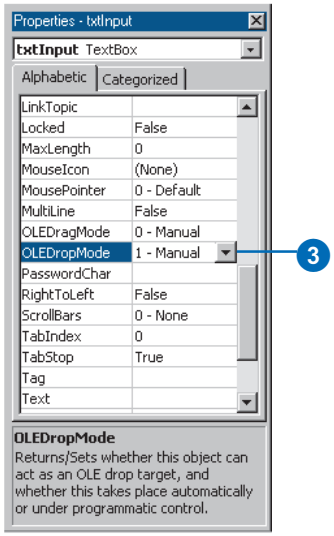
To make a robust tool, you should include routines to validate the input and output of your tool. Make sure the input is the type you need and that, if the output already exists, the tool will address the issue by displaying an appropriate message.

Enabling drag and drop from Windows Explorer

1. Open your project in Visual Basic 5.0 or 6.0.
2. Select the form and input text box you want to enable.

   In the sample code, the text box is named txtInput. Change the code to match your text box name or rename your text box to match the code.
3. Click the OLEDropMode dropdown arrow and click 1 - Manual.
4. Refer to Appendix A for the VB code that enables drag and drop from Windows Explorer. You can also copy and paste the code from the online Help system.

   If you already have a routine for OLE drag and drop, simply update it with the example routine; otherwise, place it at the end of your VB form's code window. Save the form.
5. Create a new DLL or EXE.



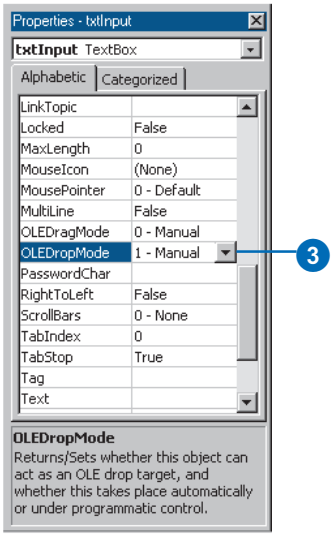
ArcCatalog also provides much more information than Windows Explorer, allowing you to filter which datasets can and can't be dropped onto a control. Name objects are supplied when you either drag and drop a dataset from ArcCatalog or use the ArcCatalog browser.

Enabling drag and drop from ArcCatalog

1. Open your project in Visual Basic 5.0 or 6.0.
2. Select the form and input text box you want to enable.

   In the sample code, the text box is named txtInput. Change the code to match your text box name or rename your text box to match the code.
3. Click the OLEDropMode dropdown arrow and click 1 - Manual.
4. Refer to Appendix A for the VB code that enables drag and drop from ArcCatalog. You can also copy and paste the code from the online Help system.

   Place the code at the end of your VB form's code window. Save the form.
5. Create a new DLL or EXE.



Adding a custom tool

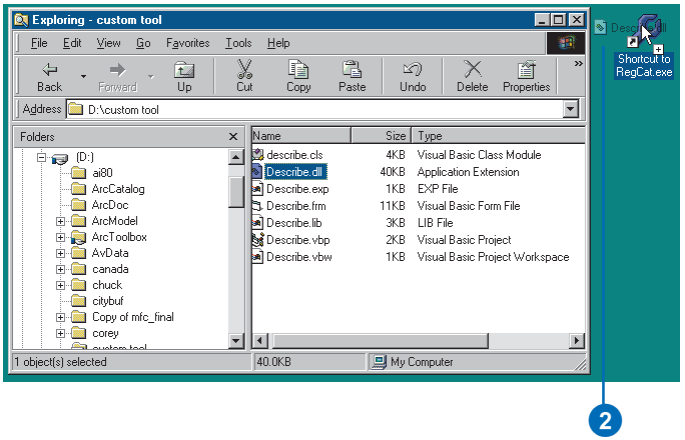
Custom tools from DLLs are added to ArcToolbox using the RegCat utility found in your $ARCHOME\bin folder. This executable registers all custom tools in the DLL with your ArcToolbox application. On startup, the ArcToolbox scans the registry for all the registered tools and displays them in the tree. RegCat may also be used to unregister (remove) tools with ArcToolbox.

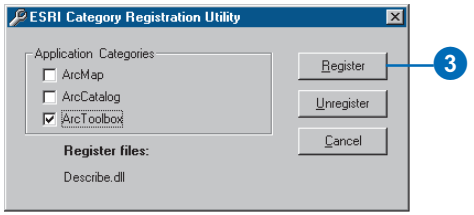
Custom tools in the form of system executables can be added to the tree by running the Add Custom EXE Wizard found in the My Tools toolset. These EXE-based tools must be added to the My Tools toolset or to a custom toolset found within My Tools. There is no such restriction with tools compiled into DLLs.

RegCat may be used either as a standalone executable from a command prompt or as a desktop tool. To see the standalone usage, type “regcat -h” at a command prompt. To use RegCat as a desktop tool, create a windows shortcut and add it to your desktop. Simply drag and drop DLLs from Windows Explorer onto the Regcat icon, then click the ▶

Adding a DLL

1. Open Windows Explorer and click the DLL you want to register with ArcToolbox.
2. Drag the DLL from Windows Explorer and drop it onto the RegCat icon on your desktop.
3. Check ArcToolbox and click Register.





appropriate application. This example uses RegCat as a desktop tool.

Tip

Managing custom tools

The location of a custom tool's DLL or EXE file should be write-protected to ensure that it isn't mistakenly deleted. If a custom tool's file is not found, a message notifies you that the tool can't be started as it can't be found. Create a directory specifically for custom tools and store the files there.

Tip

Deleting a custom tool

You can delete a custom tool by right-clicking the tool in the ArcToolbox tree and clicking Delete if it is a custom EXE or by unregistering the DLL that contains the tool using RegCat.

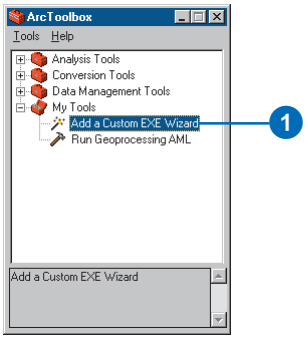
See Also

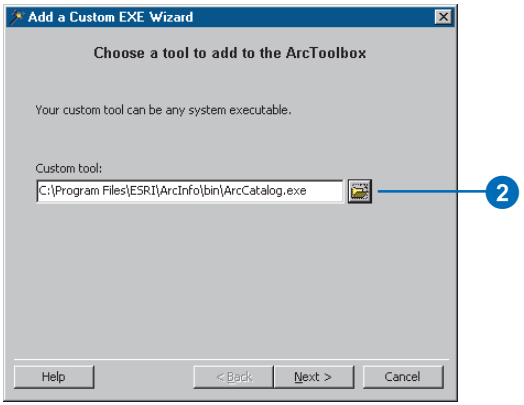
For more information on the My Tools toolset and creating toolsets for My Tools, see Chapter 4, 'Creating your own look'.

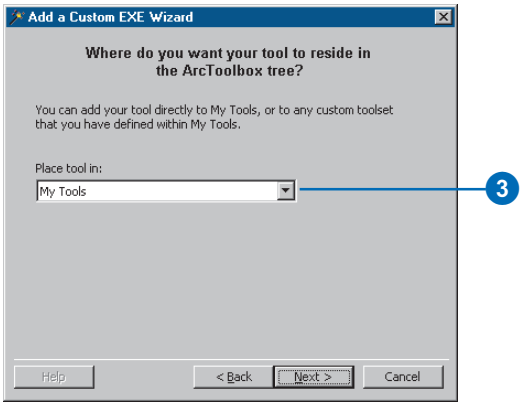
Adding an executable

1. Double-click My Tools and double-click Add a Custom EXE Wizard.
2. Click the Browse button and navigate to the executable you want to install. Click Open, then click Next on the wizard.
3. Click the dropdown arrow and click the toolset in which you want the new tool to appear. Click Next.

   Custom EXE tools may be added to the My Tools toolset or any other custom toolset you have already created in My Tools.
4. If the EXE requires one or more arguments, specify them and click Next. If it does not, click Next. ▸





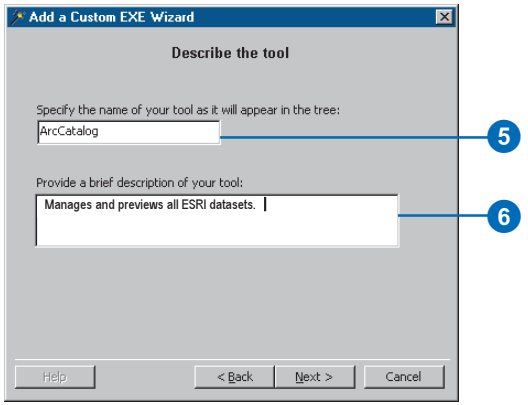


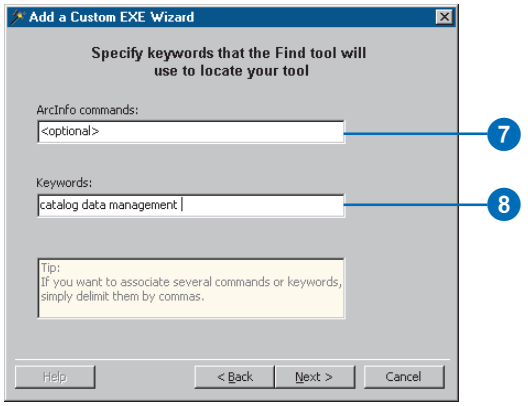
Tip

Names, descriptions, and keywords

Always give your new tools names that indicate their function. Include detailed yet concise descriptions and give a minimum of four keywords that you feel best describe your tool's function. This is very important if several people will be using the tool.

5. Type the name of your tool in the appropriate text boxes. Click Next.
6. Give a description of the tool. Click Next.
7. If the tool uses the ARC module of ArcInfo Workstation, type the ArcInfo command or commands that are utilized.
8. Type several keywords that may be used to find the tool using the ArcToolbox Find tool. Use a comma to separate each keyword. Click Next.
9. If you want to provide a help document from the tool's context menu, type a helpfile and a help context ID in the text box and click Next; otherwise, simply click Next.
10. Review the summary panel to ensure the information is correct. Click Finish.





Appendix

A

IN THIS APPENDIX

- **VB code for a custom tool class**
- **VB code for a custom tool form**
- **Enabling drag and drop in a text box for ArcCatalog**
- **Enabling drag and drop in a text box for Windows Explorer**

The Visual Basic samples found in this appendix are also available in the 'Adding custom tools' section of the ArcToolbox online Help system. Use those samples for copying and pasting into Visual Basic.

VB code for a custom tool class

```
' Describe.cls
' ESRI example for creating a custom tool for the ArcToolbox
Option Explicit
' Implement the necessary interfaces for a tool. IArcToolboxTool is required
' for all tools.
Implements IArcToolboxTool
' IArcToolboxFind is required if you want to use the Find tool to locate
' your tool.
Implements IArcToolboxFind
' IArcToolboxODE is required if your tool uses the Arc Automation server to
' execute an Arc command.
Implements IArcToolboxODE
' Define a member variable to reference the Arc automation server
' passed to the class from the IArcToolboxODE interface.
Private m_Arc As ESRI.Arc
' Define member variable for the modality setting.
Private m_Modality As AtModality
' Define a bitmap object for the tree bitmap.
Private m_Bitmap As IPictureDisp

Private Sub Class_Initialize()
  ' Default the modality to MODAL.
  m_Modality = atModal
  ' Refer to the ArcToolbox Graphic resource library for a regular tool bitmap.
  ' For a wizard use a resource value of 15123. The help node bitmap value is 15119.
  Dim pGraRes As New ATGraRes.GraphicRes
  Set m_Bitmap = pGraRes.GetBMP(15122)
End Sub
```

```
Private Property Get IArcToolboxFind_ArcCommands() As String
  ' The Arc command used by the tool, if applicable.
  IArcToolboxFind_ArcCommands = "Describe"
End Property

Private Property Get IArcToolboxFind_Keywords() As String
  ' Keywords that may be used to identify the tool.
  IArcToolboxFind_Keywords = "describe,dataset,properties,features,list"
End Property

Private Property Get IArcToolboxTool_Bitmap() As Stdole.IPictureDisp
  ' Identifies the bitmap to display in the Toolbox tree.
  Set IArcToolboxTool_Bitmap = m_Bitmap
End Property

Private Sub IArcToolboxTool_Execute(Optional InputData As Variant)
  ' Pass Arc automation server to the tool form, then show the form
  ' with the modality setting.
  frmDescribe.ArcInit m_Arc
  frmDescribe.Show m_Modality
End Sub

Private Property Get IArcToolboxTool_HelpContextID() As Long
  ' This indicates the help ID in the help file for the tool.
End Property

Private Property Get IArcToolboxTool_HelpFile() As String
  ' The help file that contains the help context ID.
End Property
```

```
Private Property Get IArcToolboxTool_Message() As String
  ' This is used at the base of the Toolbox to describe the tool when selected.
  IArcToolboxTool_Message = "Describes a dataset, resulting in a list of properties."
End Property

Private Property Get IArcToolboxTool_Modality() As esriCore.esriATModality
  ' Return if the tool is modal or modeless.
  IArcToolboxTool_Modality = IArcToolboxTool_Modality
End Property

Private Property Let IArcToolboxTool_Modality(ByVal RHS As esriCore.esriATModality)
  ' Set if the tool is modal or modeless.
  IArcToolboxTool_Modality = RHS
End Property

Private Property Get IArcToolboxTool_Name() As String
  ' The name of the tool or wizard in the ArcToolbox Tree.
  IArcToolboxTool_Name = "Describe"
End Property

Private Sub IArcToolboxTool_OnCreate(ByVal hook As Object)
  ' Use if you want access to the application itself.
End Sub

Private Property Get IArcToolboxTool_TreeviewLocation() As String
  ' This specifies where the tool lives in the ArcToolbox Tree.
  IArcToolboxTool_TreeviewLocation = "My Tools\Custom Tools"
End Property
```

```
Private Property Set IArcToolBoxODE_ArcObject(ArcAutomationServer As ESRI.Arc)
  ' Identifies the Arc automation server used by the tool.
  Set m_Arc = ArcAutomationServer
End Property

Private Property Get IArcToolBoxODE_ArcObject() As ESRI.Arc
  ' Returns the Arc automation server used by the tool.
  Set IArcToolBoxODE_ArcObject = m_Arc
End Property
```

VB code for a custom tool form

```
' Describe.frm
' Example custom tool for ArcToolbox

' Define all variables and objects that are needed.
Option Explicit
Dim lngI As Long
Dim MyArc As ESRI.Arc
Dim MyEsriStrings As New ESRIutil.Strings
Dim lngSev As Long

Private Sub cmdCancel_Click()
  ' Unload the form when you press the Cancel button.
  Unload Me
End Sub

Public Function ArcInit(Optional AIControl As ESRI.Arc)
  ' Purpose: Passes in the automation server from ArcToolbox and shows tool GUI.
  If AIControl Is Nothing Then
    Set MyArc = New ESRI.Arc
  Else
    Set MyArc = AIControl
  End If
End Function

Private Sub cmdOK_Click()
  ' If the input text box does not contain any characters, display an error
  ' message and exit the subroutine.
  If Trim(txtInput) = "" Then
    MsgBox "You must provide a dataset name.", vbExclamation, Me.Caption
```

```
    Exit Sub
  End If
  ' If the text box does contain characters, disable the OK button to prevent
  ' message and exit the subroutine.
  If Trim(txtInput) = "" Then
    MsgBox "You must provide a dataset name.", vbExclamation, Me.Caption
    Exit Sub
  End If
  ' If the textbox does contain characters, disable the OK button to prevent
  ' reentrance. Push the string 'YES' into the Arc automation server's buffer
  ' so that it is used when the question 'Do you want to continue?' is asked
  ' by the DESCRIBE command.
  cmdOK.Enabled = False
  MyArc.PushString "YES"
  ' Run the command, setting a variable to contain the ARC Macro Language (AML) severity returned
  ' by the Arc automation server.
  lngSev = MyArc.Command("describe " & txtInput, MyEsriStrings)
  If lngSev = 2 Then
    ' If the severity is 2, the command failed. Display the error in a message
    ' box. Clear the strings collection and set the focus back to the input text
    ' field with all of the characters selected. Enable the OK button.
    MsgBox MyEsriStrings.Item(0), vbExclamation, "Error Occurred"
    MyEsriStrings.Clear
    txtInput.SetFocus
    txtInput.SelStart = 0
    txtInput.SelLength = Len(txtInput)
  End If
End Sub
```

```
Private Sub Form_Unload(Cancel As Integer)
  ' Set the Arc automation server and the ESRI strings collection to nothing
  ' so that the objects are dropped from memory when the form closes. Set the
  ' entry in the registry so that ArcToolbox knows no other tool is running.
  Set MyArc = Nothing
  Set MyEsriStrings = Nothing
End Sub

Private Sub txtInput_KeyDown(KeyCode As Integer, Shift As Integer)
  ' If the user presses the Enter key, run the command by invoking the
  ' cmdOK click event.
  If KeyCode = 13 Then
    cmdOK_Click
  End If
End Sub
```

Enabling drag and drop in a text box for ArcCatalog

```
Private Declare Function RegisterClipboardFormat Lib “user32” Alias _
“RegisterClipboardFormatA” (ByVal lpString As String) As Long
' Declare function that registers a new clipboard format. This
' format can then be used as a valid clipboard format.

Private Sub txtInput_OLEDragDrop(Data As DataObject, Effect As Long, _
Button As Integer, Shift As Integer, X As Single, Y As Single)
  ' With name objects, you must first determine if the data being
  ' dropped is a name object or a regular drop from Windows Explorer.
  ' If you want to support drag and drop from ArcCatalog and  Windows Explorer, add
  ' the Windows example to this subroutine. In other words,
  ' If Data.GetFormat(lngFormat) Then
  '    accept name object
  ' Else
  '    accept a string

  ' Declare variables.
  Dim lngFormat As Long
  Dim pNameFactory As INameFactory
  Dim varBytes As Variant
  Dim pEnumNames As IEnumName
  Dim pName As IName

  Effect = vbDropEffectCopy

  lngFormat = RegisterClipboardFormat(“ESRI Names”)
  'Convert to signed integer.
  If lngFormat > 32767 Then
    lngFormat = CInt(lngFormat - 65536)
```

```
  End If
If Data.GetFormat(lngFormat) Then 'Name Object format from ArcCatalog
    'Unpackages dropped name objects.
    varBytes = Data.GetData(lngFormat)
    Set pNameFactory = New NameFactory
    Set pEnumNames = pNameFactory.UnpackageNames(varBytes)
    ' Populate the m_ValidData collection.
    pEnumNames.Reset
    Set pName = pEnumNames.Next
    If Not pName Is Nothing Then
      If InStr(1, pName.NameString, "Coverage", vbTextCompare) > 0 And _
        InStr(1, pName.NameString, "FeatureClass", vbTextCompare) > 0 Then
        txtInput = GetINFOFullPath(pName)
      ElseIf InStr(1, pName.NameString, "Coverage", vbTextCompare) > 0 Then
        txtInput = GetDatasetFullPath(pName)
      ElseIf InStr(1, pName.NameString, "Info", vbTextCompare) > 0 Then
        txtInput = GetINFOFullPath(pName)
       Else
        MsgBox "Input must be a coverage, coverage feature class, or INFO table.", _
          vbExclamation + vbApplicationModal
       End If
    End If
  End If

  ' Clean up.
  Set pNameFactory = Nothing
  Set pEnumNames = Nothing
  Set pName = Nothing
End Sub
```

```
Private Function GetDatasetFullPath(ByRef Name As IName) As String
  ' Returns the full path of a dataset name object.
 Dim pDatasetName As IDatasetName

  If TypeOf Name Is IDatasetName Then
    Set pDatasetName = Name
    If pDatasetName Is Nothing Then
      GetDatasetFullPath = “”
      Exit Function
    End If
   GetDatasetFullPath = pDatasetName.WorkspaceName.PathName & “\” & pDatasetName.Name
    Set pDatasetName = Nothing
  Else
    GetDatasetFullPath = “”
  End If
End Function

Private Function GetINFOFullPath(ByRef Name As IName) As String
  ' Returns the full path of a dataset name object.

  Dim strNameString As String
  Dim Pos As Integer
  Dim strPath As String
  Dim strTable As String
  Dim strCover As String

  strNameString = Name.NameString
 If InStr(1, strNameString, “FeatureClass =”, vbTextCompare) > 0 Then
```

```
' Feature class
' In other words, ARCINFO: Workspace = d:\data\redlands\cov\; Coverage = uuu; FeatureClass = arc;
Pos = InStr(1, strNameString, ";", vbTextCompare)
strPath = Mid(Left(strNameString, Pos - 1), 22)
strNameString = Mid(strNameString, Pos + 13)
Pos = InStr(1, strNameString, ";", vbTextCompare)

strCover = Mid(strNameString, 1, Pos - 1)

strTable = Mid(strNameString, Pos + 17)
strTable = Mid(strTable, 1, Len(strTable) - 1)
Select Case UCase(strTable)
  Case "POINT"
    GetINFOFullPath = strPath & strCover & ".pat"
  Case "ARC"
    GetINFOFullPath = strPath & strCover & ".aat"
  Case "POLYGON"
    GetINFOFullPath = strPath & strCover & ".pat"
  Case "NODE"
    GetINFOFullPath = strPath & strCover & ".nat"
  Case "TIC"
    GetINFOFullPath = strPath & strCover & ".tic"
  Case Else
    Pos = InStr(1, strTable, ".", vbTextCompare) + 1
    If InStr(1, UCase(strTable), "REGION", vbTextCompare) > 0 Then
      GetINFOFullPath = strPath & strCover & ".pat" & Mid(strTable, Pos)
    ElseIf InStr(1, UCase(strTable), "ROUTE", vbTextCompare) > 0 Then
      GetINFOFullPath = strPath & strCover & ".rat" & Mid(strTable, Pos)
```

```
      ElseIf InStr(1, UCase(strTable), "SECTION", vbTextCompare) > 0 Then
        GetINFOFullPath = strPath & strCover & ".sec" & Mid(strTable, Pos)
      ElseIf InStr(1, UCase(strTable), "ANNOTATION", vbTextCompare) > 0 Then
        GetINFOFullPath = strPath & strCover & ".tat" & Mid(strTable, Pos)
       Else
        GetINFOFullPath = ""
       End If
   End Select

 ElseIf InStr(1, strNameString, "InfoTable =", vbTextCompare) > 0 Then
   ' Standalone INFO table.
   ' In other words, ARCINFO: Workspace = d:\data\redlands\cov; InfoTable = monk1.add;
   Pos = InStr(1, strNameString, ";", vbTextCompare)
   strPath = Mid(Left(strNameString, Pos - 1), 22)

   Pos = InStr(1, strNameString, ";", vbTextCompare)
   strTable = Mid(strNameString, Pos + 14)
   strTable = Mid(strTable, 1, Len(strTable) - 1)
   GetINFOFullPath = strPath & "\" & strTable
 Else
   GetINFOFullPath = ""
 End If
End Function
```

Enabling drag and drop in a text box for Windows Explorer

```
Private Sub txtInput_OLEDragDrop(Data As DataObject, Effect As Long, _
Button As Integer, Shift As Integer, X As Single, Y As Single)
  ' Set the contents of the text box to be blank and then set the
  ' text box to the value of the dataset being dropped.
  Dim strFile As String
  txtInput.Text = ""
  strFile = CStr(Data.Files.Item(1))
  txtInput.Text = strFile
End Sub
```

Appendix B

IN THIS APPENDIX

- **Starting the Geoprocessing Server**
- **Accessing data on UNIX or Windows NT**
- **Setting server defaults**
- **Pausing, resuming, and shutting down a service**
- **Adding another instance**
- **Troubleshooting server problems**

This appendix is the 'Geoprocessing Server Administration Guide'. This guide is intended to help administrators add, start, and stop a Geoprocessing Server instance. Setting up a client requires you to have either administrator or owner privileges.

Starting the Geoprocessing Server

The following tasks show you how to start the Geoprocessing Server on a UNIX or Windows NT system and how to configure a client so that it can submit jobs to a server.

You will need administrator privileges to install the server. Once the server has been installed on a UNIX system, the server must be started from the same login account that installed it. This account would have ownership of the ARCHOME directory.

The password specified when creating the service on Windows NT or starting the service on UNIX will be the administrator password for the service. This password can be anything when it is first used and can be changed by stopping and starting the service again with a new password.

When the Geoprocessing Server service is started, it searches the system services file for a service name that matches the service name in the services.rqm file. If a match is not found, an error is returned to the UNIX prompt or the event log in Windows NT. ►

Installing the Geoprocessing Server as a Windows NT service

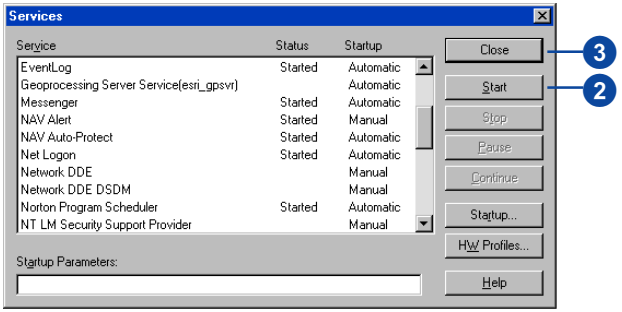
1. Run the command line utility %ARCHOME%\bin\gpservice.exe at the DOS prompt. Click the create option and a new password. The default name given for the instance is esri_gpsvr.

1. %ARCHOME%\bin\gpservice -o create -p gps -H D:\arcexe80 -i esri_gpsvr

Add the instance name to your server services file before you try to start the service. The password "gps" was determined by an administrator for this service. You can use any password you want.

Starting the Geoprocessing Server service on Windows NT

1. Open the Services dialog box from the Windows NT Control Panel.
2. Click Geoprocessing Server Service(esri_gpsvr) and click Start.
3. Click Close when the status of the Geoprocessing Server is "Started".



When a match is found, a background process called giomgr starts. It listens to the Transmission Control Protocol/ Internet Protocol (TCP/IP) port number assigned to the service name for user connection requests.

The services.rqm file should contain only one service name. If more than one exists, then the Geoprocessing Server will use the first one it encounters.

If you are on UNIX, you must change ownership of the arcrqmgr executable located in $ARCHOME/bin so that it is owned by root. The permissions for this executable must be set to 6755.

Some UNIX systems direct applications to search the NIS services file rather than the hosts. You can force a search of the local services file on these systems so you can properly administer the Geoprocessing Server.

Tip

Using gpmon on Windows NT

The gpmon utility is also available on Windows NT in the %ARCHOME%\bin directory. The Geoprocessing Server instance can be managed the same way on Windows NT as on UNIX if you prefer to use the command line.

Starting the Geoprocessing Server on UNIX

1. Run the command line utility $ARCHOME/bin/gpmon with the start option and administrator password.

   Once the server has been installed on a UNIX system, the server must be started using the same login account that installed it. This account should have ownership of the ARCHOME directory.

1 $ARCHOME/bin/gpmon -o start -p gps

Specifying the server instance is not necessary on UNIX unless multiple geoprocessing servers have been installed.

Updating service access on HP-UX®

1. Copy the nsswitch.conf file from the /usr/newconfig/etc directory to the etc directory.
2. Edit the file and change the line ‘services: nis files’ to ‘services: files nis’.

Updating service access on IBM® AIX®

1. Create the file netsvc.conf in the etc directory.
2. Open the file and add the line ‘services=local,nis’.
3. Save the file.

Accessing data on UNIX or Windows NT

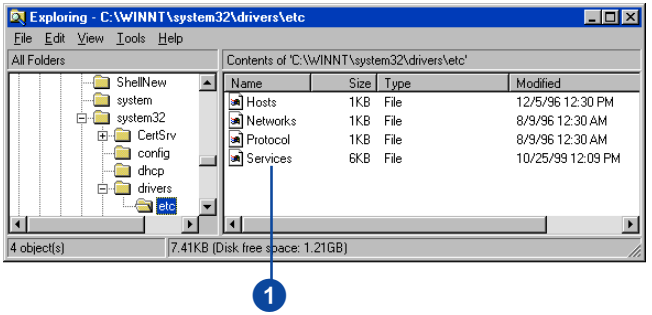
If your data resides on a Windows NT client machine, it must be stored in a shared drive if it is to be submitted to a Geoprocessing Server. A user-defined share name must be used instead of the default share name. For example, the default share name C$ will not work, but C_drive will.

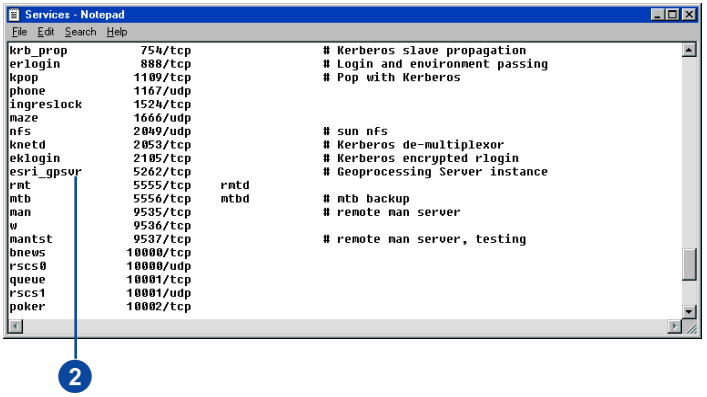
Permissions must be set so that the login used to create the Geoprocessing Server connection has the proper access to the data being used in the Geoprocessing Server job.

A UNIX Geoprocessing Server requires that all data input and output be from a UNIX directory accessible from the UNIX server. Data stored on Windows NT should not be passed to a UNIX Geoprocessing Server, as ESRI does not support UNIX connections to Windows NT disks.

Enabling a client

1. Open the system services file on the client machine. On Windows NT, it is found in C:\winnt\system32\drivers\etc\, while on UNIX, it is in /etc.
2. Add the name of the server instance and its port number to the file. To see the existing server instance name and port information, read the %ARCHOME%\gpserver\services.rqm file.
3. Save the file.





Setting server defaults

As a Geoprocessing Server administrator, you can set the maximum number of requests that can be managed by the server at one time. You can also limit the number of server processes that can be activated. By setting the maximum number of processes, you can control how many jobs are executed concurrently by the server.

A Geoprocessing Server only uses one license at a time, regardless of the number of jobs being processed. If a license is not available, requests are placed in a queue until a license becomes available. The queue operates on a first in, first out rule. Administrators can list or update the contents of the queue using the gpmon utility.

The allowable time difference in seconds, which is the maximum difference a client's clock can deviate from the server's clock, can also be set.

A path may be set to a location where temporary disk space is also available for the server.

Setting the maximum number of requests

1. Open the rqmgr.defs file located in $ARCHOME/gpserver.
2. Change the maximum number of requests as desired.
3. Save the file and quit.

Setting the maximum number of processes

1. Open the rqmgr.defs file located in $ARCHOME/gpserver.
2. Change the maximum number of processes as desired.
3. Save the file and quit.

Setting the maximum time difference allowed

1. Open the rqmgr.defs file located in $ARCHOME/gpserver.
2. Set the maximum client/server time difference allowed as desired.
3. Save the file and quit.

Tip

Temporary disk space

The result messages of each server request are stored in the temporary disk space location defined in the rqmgr.defs file. In order to save disk space and decrease the number of active processes being managed by the server, remove request result messages from the server when a job has completed and when the result messages for the job are no longer needed. Remove these messages with the gpmon utility or the Remote Processing tab in ArcToolbox.

Setting a path to temporary disk space

1. Open the rqmgr.defs file located in ARCHOME/gpserver.
2. Set the path to temporary disk space.

   All relative pathnames must exist in the directory specified by the environment variable $ARCHOME; otherwise, an absolute pathname must be used.
3. Save the file and quit.

Listing requests on a server

1. Run the gpmon utility with the list option to display all requests on a server.

1 $ARCHOME/bin/gpmon -o list -p gps -t all

Removing a submitted request

1. Run the gpmon utility with the remove option, specifying the request ID you want removed.

1 $ARCHOME/bin/gpmon -o remove -p gps -t 13

Pausing, resuming, and shutting down a service

The Geoprocessing Server manager has three modes: running, paused, and shutdown. When a server is running, clients may submit a job. When paused, current running jobs will continue, but additional job requests may not be submitted. This is convenient when you want to shut down the server but want existing jobs to finish first. Shutting down a server will terminate all running jobs, and new jobs cannot be submitted. Shutting down also relinquishes all server processes and operating system resources.

Specify the server administrator password that was created when the service was started.

Tip

Using a remote computer

To pause, resume, or stop the service from a remote machine, on UNIX or Windows NT, specify the server and instance names when running gpmon.

Pausing the server

1. Run the command line utility $ARCHOME/bin/gpmon using the pause option.

1. $ARCHOME/bin/gpmon -o pause -p gps

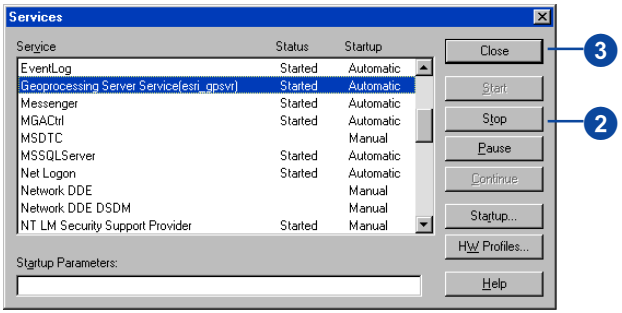
Resuming operation

1. Run the command line utility $ARCHOME/bin/gpmon using the resume option.

1. $ARCHOME/bin/gpmon -o resume -p gps

Shutting down a Windows NT server

1. Open the Services dialog box from the Windows NT Control Panel.
2. Click Geoprocessing Server service and click Stop.
3. Confirm that you want to stop the service and click Close when the status of the server is blank.



Shutting down a UNIX server

1. Run the command line utility $ARCHOME/bin/gpmon using the shutdown option.

1. $ARCHOME/bin/gpmon -o shutdown -p gps

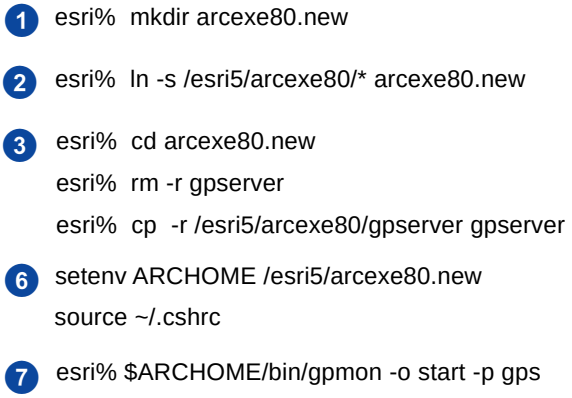
Adding another instance

You may want more than one server instance running on one machine. This is not recommended on Windows NT, as you must have two installations of ArcInfo on one machine, making ArcInfo administration more difficult. On UNIX, you may create another ArcInfo installation directory and create links for all of the directories in the existing ARCHOME directory with the exception of the gpserver directory. That directory must be different and contain a services.rqm file with another instance name. It is helpful to have another administrator account to manage this new ARCHOME directory.

To add a new instance, you must update your system services file and services.rqm file with the new instance name and port number. While the default port number is 5262, you must ensure that it is not already in use, as port numbers must be unique. This port number is not registered with the Internet Services Institute/Internet Assigned Numbers Authority, so it may already be in use by another application. If so, check your system services file and add the next unique ID.

Adding a Geoprocessing Server instance on UNIX

1. Create a new directory that will be your new ARCHOME directory.
2. Add symbolic links to all directories in your old ARCHOME directory.
3. Delete the gpserver link and copy the gpserver directory from your old ARCHOME directory to your new one.
4. Edit the services.rqm file in the new ARCHOME/gpserver directory to add the new instance name and port number.
5. Add the same instance and port information to the system services file on the server (/etc/services).
6. Reset the ARCHOME variable to point to the new ArcInfo installation directory. Source your .cshrc file to reset any variables that use the ARCHOME variable.
7. Run the command line utility $ARCHOME/bin/gpmon using the Start option, then specify the new instance name.



Troubleshooting server problems

The Geoprocessing Server has a request server error log that captures all of its errors. When a command fails, consult this log for helpful information.

Occasionally, you may not be able to shut down an instance due to hung processes or because the instance itself has been corrupted. These problems may occur for various reasons such as a system malfunction. In this case, you can use the Killp utility to stop the instance so you can start a new one.

You may want to reset the server so you can start it from its original state. Doing this will delete all submitted jobs and all of the results currently residing on the server. Only do this if you are sure this information is not required by anyone who previously submitted a job to the server.

Reading the server request error log

1. Open and read the file rqmgr.err found in %ARCHOME%\gpserver.

Killing the server process

1. Find the process ID of the Geoprocessing Server. Use the task manager on Windows NT or use 'ps -ef | grep gp' on UNIX.
2. Run the command line utility %ARCHOME%\gpserver\killp with the process ID to stop the server instance.

2 $ARCHOME/gpserver/killp 4270

Resetting the server

1. Stop the server if it is still running (refer to 'Shutting down a Windows NT server' or 'Shutting down a UNIX server', earlier in this appendix).
2. Delete the directory with the same name as the instance in the gpserver directory located in ARCHOME.
3. Delete all of the files in the tmp_results and aml directories also located in the gpserver directory.
4. Start the server.

This section outlines the causes and solutions to some errors you may experience when using the Geoprocessing Server.

Unable to find results of a job submitted to a UNIX server

I submit a job request to my remote Geoprocessing Server, and the Remote Processing dialog box states that it is completed. When I review the results of the job, I see something similar to this:

AML ERROR—unable to run file

line 3 of file /test/temp/arcexe80/gpserver/amls/request9.aml

AML MESSAGE—stopping execution of AML file due to ERROR condition

The request9.aml in $ARCHOME/gpserver/amls on the UNIX server looks like this:

&codepage ANSI

Precision Single Highest

&RUN \\tashi\hippo1\james\sample.aml

Cause:

1. The Geoprocessing Server was unable to find the AML as the drive /hippo1 was not mounted on the server.

2. The share name set in the NFS software is HIPPODRIVE, but the mount point of the drive on the UNIXserver is /hippo.

For example, Windows NT sees \\HIPPO\HIPPODRIVE rather than \\hippo\hippo1.

Solution:

1. Mount the disk with the AML on it to the UNIX server.

2. Ensure the Windows NT share name and the UNIX disk name are the same.

When running a job on a UNIX server, the job completes but the results of the job state an AML error because the input data does not exist.

Here are the job results:

SET SCHEMAWRITE lock on \\tashi\ashley\data\streams

Coverage \\tashi\ashley\data\streams does not exist

AML MESSAGE—stopping execution of AML due to ERROR condition

The AML request ($ARCHOME/gpserver/amls/request2.aml) appears as follows:

&codepage ANSI

LOCKMANAGER \\tashi\ashley\data\streams SET SCHEMAWRITE

Precision single highest

Build \\tashi\ashley\data\streams lines

LOCKMANAGER \\tashi\ashley\data\streams

Cause:

UNIX cannot resolve the pathname. UNIX is looking for the data at the location of /ashley/data/streams. In reality, the data is located at /regina/ashley/data/streams. The problem is that when the UNIX box was mapped from Windows NT, it was mapped to \\tashi\ashley. The share name ashley was set by the UNIX system administrator through an NFS package—in this case it was SAMBA.

Solution:

Make sure that the share name is the same as the disk on UNIX. The UNIX administrator needs to make sure that the share name is the same as the UNIX disk name. In this example, the problem is resolved by mapping \\tashi\regina on Windows NT.

Unable to start the Geoprocessing Server, error 1067

The following error is given when gpmon is used to start the server:

> Error 1067: The process terminated unexpectedly
>
> C:\arcexe80\gpserver> gpmon -o start -p gps -i esri_gpsvr
>
> esri_gpsvr service failed during initialization.
>
> Please check event log or error log files.
>
> Error starting esri_gpsvr service(997)
>
> Could not start Geoprocessing Server - Check Network and $ARCHOME disk.

Cause:

The appropriate files or directories were not created in the %ARCHOME%\gpserver directory.

The gpserver directory should contain the following files:

> rqmgr.defs, services.rqm

Solution:

Reinstall ArcInfo Workstation to ensure the required files for the Geoprocessing Server exist.

Unable to start the Geoprocessing Server, error 2140. An internal Windows NT error occurred.

Solution:

1. Make sure the following directories exist:

%ARCHOME%\gpserver\tmp_results

%ARCHOME%\gpserver\amls exist.

2. Make sure all users sending jobs to the server have full access to the tmp_results and amls directories.

3. If you still have this problem, make a note of the contents of the %ARCHOME%\gpserver\rqmgr.err file and contact ESRI Technical Support.

Glossary

ArcToolbox tree

Displays ArcToolbox toolsets and tools grouped by functionality.

batch mode operation

Executes a given tool two or more times. Includes a utility to save the batch entries as a geoprocessing ARC Macro Language (AML) script and reload the AML for later execution.

batch table

Displays the input name, user-selected parameters, and output name, where applicable, for all batch entries.

CAD

See computer-aided design.

computer-aided design

An automated system for the design, drafting, and display of graphically oriented information.

coverage

1. A digital version of a map forming the basic unit of vector data storage in ArcInfo. A coverage stores geographic features as primary features (such as arcs, nodes, polygons, and label points) and secondary features (such as tics, map extent, links, and annotation). Associated feature attribute tables describe and store attributes of the geographic features.

2. A set of thematically associated data considered as a unit. A coverage usually represents a single theme such as soils, streams, roads, or land use.

custom tool

A tool created by a user and added to the ArcToolbox My Tools toolset.

custom toolset

A folder created by a user within the ArcToolbox My Tools toolset to hold custom tools or group frequently used ArcToolbox tools.

database management system

A set of computer programs for organizing the information in a database. A DBMS supports the structuring of the database in a standard format and provides tools for data input, verification, storage, retrieval, query, and manipulation.

dataset

A named collection of logically related data items arranged in a prescribed manner.

DBMS

See database management system.

DLL

See dynamic link library.

dynamic link library

Modules that contain functions and data. A DLL is loaded at runtime by its calling modules (EXE or DLL).

EXE

See executable.

executable

A program file created from one or more source code files translated into machine code and linked together.

exit state

The condition of a tool upon closure. If a tool fails due to a programming bug or command failure, the exit state will be "failed".

feature class

1. A classification describing the format of geographic features and supporting data in a coverage. Coverage feature classes for representing geographic features include point, arc, node, route-system, route, section, polygon, and region. One or more coverage features are used to model geographic features; for example, arcs and nodes can be used to model linear features such as street centerlines. The tic, annotation, link, and boundary feature classes provide supporting data for coverage data management and viewing.

2. The conceptual representation of a geographic feature. When referring to geographic features, feature classes include point, line, area, and surface.

geoprocessing

GIS operations such as geographic feature overlay, coverage selection and analysis, topology processing, and data conversion.

Geoprocessing Server

A UNIX- or Windows NT-operating computer that processes geoprocessing tasks remotely. It includes a utility to schedule remote processing.

grid

A geographic data model representing information as an array of equally sized square cells arranged in rows and columns. Each grid cell is referenced by its geographic x,y location.

Help node

A branch in the ArcToolbox tree, providing overview Help for ArcToolbox toolsets.

job

A task scheduled on a Geoprocessing Server for immediate or later remote processing. A job can be a single tool or batch mode execution.

layer

A collection of similar geographic features—such as streams, lakes, highways, political boundaries, or wildlife habitats—in a particular geographic area for display on a map. A layer doesn't store geographic data itself, but instead references the data stored in a data source such as a coverage or shapefile. You can create and manage layers as you would any other type of data in your database.

relate manager

A utility used to build, modify, save, and delete connections (relates) between tables. Saved connections can be reactivated in future ArcToolbox sessions.

relational join

The operation of relating and physically merging two attribute tables using their common item.

tool

An entity in ArcToolbox that performs a specific geoprocessing task such as generalizing lines. A tool can belong to one or more toolsets.

toolset

A grouping of tools that perform a similar geoprocessing task. Custom toolsets can be created within the My Tools toolset to hold custom tools as well as to group frequently used ArcToolbox tools.

wizard

A tool that leads a user step by step through an unusually long, difficult, or complex task.

Index

E

F

G

H

I

J

K

L

M