STK Level 1 and Level 2 Training Manual

STK VERSION 13.1.0, MAY, 2026

Table of Contents

STK Level 1 - Beginner Tutorials

The Level 1 - Beginner tutorials are designed to teach you the fundamentals of the Ansys Systems Tool Kit® (STK®) digital mission engineering software and get you started using the STK application.These tutorials cover the basics of the STK software that can be completed with an Evaluation license.

Once you have completed this Fundamentals training, you will be ready to take the free Level 1 - STK Certification test.

Tutorial	Capabilities	Required License	Required Install
Lesson One: Build Scenarios (Design Reference Missions [DRM]) Learn how to create a scenario in the STK application.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Lesson Two: Objects and Properties Learn how to add STK objects to a scenario and modify them.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Lesson Three: Access Report and Graphs Learn how to compute access between objects and generate reports on scenario data.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Lesson Four: Movies and Visual Data Files Learn how to make a movie in the STK application.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Lesson Five: Introduction to Connect Learn how to send Connect commands in the STK application.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro

Part 1: Build Scenarios

> **Note:** Visit **help.agi.com/stk/#training/Day1Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators need an environment to quickly model and simulate their missions both analytically and visually.

Solution

The Ansys Systems Tool Kit® (STK®) digital mission engineering software provides a physics-based simulation environment for digital mission engineering. Your system and component models can interact in the STK application, enabling you to measure their performance in the context of the complete mission.

Creating a new STK scenario

Create a new STK scenario using the STK: New Scenario Wizard. A Scenario ( ) object defines the context in which the properties and behavior of other objects are defined.

1. Click Create a Scenario in the Welcome to STK dialog box.

   The Welcome to STK wizard is an easy way to jump into the STK application by creating a new scenario, opening an existing one, or learning about the STK software.

2. You can use the Central Body selection, which defaults to Earth, to change the scenario's central body.
3. Enter the following in the STK: New Scenario Wizard:

Option	Description

Name	STK_NewScenario
Description	This is my first STK scenario.
Location	STK Default Directory (for example, C:\Users\<username>\Documents\STK_ODTK 13).
Start	Default Start Time
Stop	+ 24 hrs

4. Click **OK** when you finish.

Saving the scenario

When the scenario first opens or when you complete a set of steps, you should Save (💾) the scenario. By using the following steps, the STK application will create a unique folder with your scenario files in it.

1. Click Save (💾) when the scenario loads. The STK application creates a folder with the same name as your scenario for you.
2. Verify the scenario name and location in the Save As dialog box.
3. Click **Save** .

> ⚠ **Important:** It's important that each scenario is saved in a unique folder; you don't want to mix scenario files from different scenarios in the same location.

Understanding the STK workspace

The STK workspace offers many ways for you to organize and interact with its many varied windows. At the completion of this tutorial, you can practice workspace customization.

1. Click **Close** in the Insert STK Objects () tool.
2. Select the View menu.
3. Hover over the Toolbars menu option to extend the Toolbars menu. This is one way to choose the toolbars you require.
4. Right-click in the empty space to the right of the currently available toolbars to show the shortcut menu. This is another way to quickly enable and disable Menu selections and toolbars.

Using the 2D Window Defaults toolbar

The 2D Graphics window graphically displays information about your scenario. The 2D Graphics window gives you one giant advantage: you can see the whole central body at one glance.

Updating the 2D Graphics window's properties

You can customize the 2D Graphics window by modifying its properties ( ), such as colors, geographic features, display formats, and latitude / longitude line spacing.

Using Grab Globe

When selected, Grab Globe enables you to move or pan the central body in grab mode. Grab Globe is enabled by default.

1. Click the Grab Globe ( ) button in the 2D Graphics window's 2D Window Defaults toolbar if it's not already enabled.
2. To pan, place your cursor on the 2D Graphics window map.
3. Hold the left mouse button down and drag your cursor around to move around in the 2D Graphics window.

Using Zoom In

Zoom In brings the view closer to the selected area and changes the map center. Each time you zoom in, it adds one zoom level to the zoom stack.

1. Click Zoom In ( ) in the 2D Window Defaults toolbar.
2. Place your cursor on the 2D Window Defaults window map.
3. Hold down the left mouse button.
4. Drag your cursor across the area of the 2D map window to which you want to zoom. This draws a box around the area of interest.
5. Release the left mouse button. This zooms in to the area you selected.

Using Zoom Out

Zoom Out widens the view of the 2D Graphics window one zoom level per click.

1. Click Zoom Out ( ) in the 2D Window Defaults toolbar to zoom out one zoom level.
2. You can continue to click Zoom Out ( ) until the entire map is shown in the 2D Graphics window.

Using the mouse scroll wheel

You can zoom in and zoom out using the mouse scroll wheel. Using the mouse scroll wheel does not add any zoom levels to the zoom stack, so after zooming in using the mouse scroll wheel, clicking Zoom Out zooms all the way out to a global view.

1. Place your cursor on the 2D Graphics window map.
2. Move the mouse scroll wheel up to zoom in to the map in the 2D Graphics window.
3. Move the mouse scroll wheel down to zoom out of the map in the 2D Graphics window.

Using the Microsoft Bing™ Maps toolbar in the 2D Graphics window

The Microsoft Bing™ Maps toolbar allows you to choose a background image (Microsoft Bings™ Maps or Basic.bmp) for the 2D Graphics window.

1. Click the Microsoft Bing™ Maps ( ) button in the Microsoft Bing™ Maps toolbar in the 2D Graphics window.
2. Try selecting different map styles from the drop-down menu.
3. When finished, reset the map style to Aerial.

Understanding the 3D Graphics window

The STK application enables you to view complex information in a 3D Graphics window. You can observe relationships among space, air, land, and sea objects as they exist in the present, as well as in the past and future. When you create a new scenario, the default primary central body (Earth) appears in the 3D Graphics window unless you selected a different primary central body. Like you did with the 2D Graphics window, you'll practice using selected tools.

Using the 3D Window Defaults toolbar

Most of the tools have functions similar to the 2D Window Defaults toolbar. You will focus on the Grab Globe and Zoom In tools, but this toolbar also gives access to Properties, Snap Frame, and Snap Properties.

Updating the 3D Graphics window's properties

You can customize the 2D Graphics Window by modifying its properties ().

Using Zoom In

Zoom In enables you to zoom into a portion of the 3D Graphics window. The camera is then fixed on the center point of the zoomed in area.

1. Click Home View () in the 3D Graphics window's 3D Graphics toolbar to reset your view. You will become familiar with Home View () in the 3D Graphics toolbar section of this lesson.
2. Rotate the globe by holding down the left mouse button and moving the mouse around to obtain a view of a landmass in the 3D Graphics window, such as North America.
3. Click Zoom In () in the 3D Window Defaults toolbar.
4. Hold down the left mouse button and drag your cursor across the area of the 3D globe you want to zoom in to, such as the west coast of the United States.
5. Continue to zoom in using the mouse scroll wheel until you're fairly close to the ground.

Using Grab Globe

Although it is similar to the Grab Globe tool in the 2D Graphics window, there's one important difference: you need to turn Grab Globe on and use the Shift key along with the left mouse button.

1. Click Grab Globe () in the 3D Window Defaults toolbar.
2. Hold down the Shift key and the left mouse button and drag the mouse around the 3D Graphics window to grab the globe and pan.
3. Release the Shift key and the left mouse button.
4. Use the mouse scroll wheel to zoom in or out depending on the detail you are trying to obtain.

5. Hold down the left mouse button and move the mouse around to rotate and tilt the globe. The globe pivots on the center point of the zoomed in area.

Using the mouse with the 3D Graphics window

You can use the mouse to interact with the 3D Graphics window. In addition to tools available in toolbars, the 3D Graphics window has additional mouse controls that manipulate the 3D Graphics window. Get familiar with the mouse controls.

1. Double-click on any spot on the globe to display the latitude and longitude of the selected spot.
2. Place your cursor on the 3D Graphics window globe.
3. Hold down the left mouse button and move the mouse around to rotate the globe.
4. In one motion, hold down the right mouse button and push the mouse away from you to Zoom out.
5. While holding down the right mouse button, pull the mouse closer to you to Zoom in.
6. Hold down the Shift key and left mouse button while moving the mouse around. This will pan and tilt the virtual camera.
7. After you pan and tilt in the previous step, release the Shift key and left mouse button.
8. Hold the left mouse button down and move the mouse around to keep the camera in a fixed point in space and change its orientation.

Using the Home View

The Home View displays the default view. If you are lost in your view, this is one of your best friends. It will bring you back to your home view with one click.

1. Use the scroll wheel to zoom in close to the surface of the central body (Earth).
2. Click Home View ( ) in the 3D Graphics toolbar to reorient the 3D Graphics camera back on the default Earth-centered view.

Using the Animation toolbar

The Animation toolbar helps you control the progression of your scenario. With a few clicks, you can control the start time, stop time, and how quickly the scenario progresses from start to stop. Use the animation controls to play through the scenario.

1. Select the Window menu.
2. Select Tile Vertically.
3. Click Home View ( ) in the 3D Graphics toolbar.
4. In the Animation toolbar, click Start ( ) to have the scenario animate forward in time from the current point.
5. Click Pause ( ) to stop the scenario animation.
6. Click Step Forward ( ) or Step Backward ( ) to progress one step at a time through the scenario.
7. Click Reverse ( ) to have the scenario animation rewind from the current point.
8. Experiment with increasing ( ) and decreasing ( ) the speed of the animation, which by default is 10 seconds. This will also affect the time step of Step Forward and Step Backward as well.
9. At any point, click Reset ( ) to have the scenario return to the start time. This does not reset the speed of the animation.

Using the Timeline View

Use the Timeline View to visualize a variety of time intervals within your scenario. The user interface of the Timeline View is comprised of a toolbar, three timelines, and rows of time components.

Understanding the Time Display

The Time Display is the largest and most granular of the timelines. Use the gray pointer to adjust the animation time to any point within the Time Display's current boundary to visualize time events.

1. Click Home View ( ) in the 3D Graphics toolbar to reset the view.
2. Click Zoom Out ( ) in the 2D Window Defaults toolbar to zoom out of the 2D window. Click Zoom Out ( ) as many times as needed until the 2D map fills the display window.
3. While looking at the 2D Graphics or 3D Graphics window, put your cursor on the Gray Pointer ( ) in the Timeline View.
4. Hold down the left mouse button and slide the Gray Pointer to the right.
5. Notice the globe rotating and the day-night shading on the 2D Graphics window moving as you scroll through the scenario time period.

Moving external files to your scenario folder

There are times that you might want to add external files to your scenario folder (for example, image, .txt, and .csv files). This is common when sending your scenario to AGI Support or Certification.

The common location if you use the default path during the STK application setup for STK_ODTK 13 is:

- C:\Users\<username>\Documents\STK_ODTK 13\<Scenario name>

All you have to do is copy and paste or click and drag the required files into the scenario folder.

Saving a scenario to a new folder

There are instances when you may want to save the current scenario in a new location, and possibly with a new scenario name. This step often causes people to save their scenario outside of a unique folder. If you end up saving multiple scenarios in the same folder, or worse, on your desktop, you will run into problems when analyzing data. Remember: you always want your scenario in a unique folder. It's a good idea to give it a try.

1. Open the File menu.
2. Select Save As... in the File menu.
3. Select STK User ( ) in navigation pane on the left side of the Save As dialog box.
4. Click New folder in the ribbon.
5. Rename the folder anything you desire.
6. Select the new folder you created.
7. Click **Open** .

   This is where a lot of people make a mistake using the Save As procedure. They forget to select the new folder and end up placing the scenario and all of its files out in the open in the STK_ODTK 13 folder. Do this more than once with different scenarios and you will run into problems.

8. When the folder opens, enter a name in the File name field.

   This will be the name of the Scenario object of your new scenario, so it follows the naming rules of all objects in the STK application.

9. Check to make sure Scenario Files (*.sc) is selected for the Save as type.
10. Click **Save** .

You now have a complete copy of your original scenario, but now it's located in a different folder and has a new scenario name.

> ⚠ **Important:** If you have non-STK files, such as text or CSV files, in the original scenario folder, they won't transfer to the new folder. You must manually move them or copy them to your new scenario folder.

Creating a Visual Data File (VDF)

You can convert your STK scenarios to Visual Data (.vdf) files for sharing your scenario with managers, coworkers, customers and prospects for collaboration, without the need for a full STK installation. Although it's not a true compressed file, creating a VDF places all your scenario files into one file. There may also be instances when you need to send your scenario to AGI Support or Certification where you will be instructed to create a VDF and send them the file.

1. Select the File menu at the top of STK.
2. Select VDF Setup... in the File menu.
3. Select the Exclude Install Files check box when the VDF Setup dialog box opens.
4. Look at the Custom/User Data panel.

   You can select the Copy scenario folder contents check box to include both scenario files and non-STK files, like Word document and Excel workbooks, with the VDF file.
5. Look at the Quick Reports For STK Viewer list.

   Quick Reports will be covered in the Access Reports and Graphs tutorial. If you have Quick Reports, you can include them in your VDF.
6. Click **Create VDF...** .
7. Ensure the Save location is in your scenario folder.
8. Click **Save** .

Closing the scenario

This seems simple enough, but you have a couple of choices to make. Do you want to continue working with a different scenario, or do you want to completely shut down the STK application?

Closing the scenario but not the STK application

If you're moving on to a new or different scenario, you'll want to keep the STK application open.

1. Select the File menu.
2. Select Close in the File menu.

   You can now create a new scenario or open an existing scenario. The STK application remains open.

3. Click **Open a Scenario** in the Welcome to STK dialog box.
4. Select one of your scenario folders in the Open window.
5. Click **Open** .
6. Select the STK scenario file.
7. Click **Open** .

Summary

You got a thorough introduction of the STK: New Scenario Wizard and a brief outline of STK tools and windows such as the Object Browser and the Timeline View. The bulk of the information in this tutorial helped you to become familiar with the functions and tools of the 2D and 3D Graphics windows. These windows are important for your situational awareness while using the STK application. You ended by understanding how to save and close a scenario.

Part 2: Objects and Properties

> **Note:** Visit **help.agi.com/stk/#training/Day1Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators need to quickly add realistic analytical and visual properties to objects in the STK application. They may need a realistic satellite attitude, analysis of an enclosed area in a deep canyon, a mission plan for an aircraft flight route, a sensor footprint, or a briefing with detailed visuals and analysis.

Solution

Use the STK software to insert STK objects into your scenario. Then update the objects' properties to define their characteristics that are relevant to your scenario.

Modifying the Scenario object's properties

The Scenario ( ) object defines the context that influences the properties and behavior of other objects.

1. Right-click on the Scenario ( ) object in the Object Browser.
2. Select Properties ( ).

Insert STK Objects tool

The Insert STK Objects tool provides an easy, convenient way to populate a scenario. The Insert STK Objects tool, by default, displays the most commonly-used scenario objects. You can customize the tool to display all the scenario objects or a user-defined subset of objects.

There are two ways to reopen the Insert STK Objects tool:

- Click Insert Object ( ) located in the Default toolbar.
- Expand the Insert menu and select New.

Objects and Methods

In the Insert STK Objects tool, you will see Scenario Objects, Attached Objects, and Methods. Scenario objects are children of the larger Scenario ( ). Attached objects are the children of scenario objects. You must add scenario objects before you can add attached objects.

To insert an object, you must choose a particular method. These methods determine the starting properties of each object. You can change these properties later. For instance, when this tutorial says to "Insert a Default Object," that means insert an object using the Insert Default method. You should pick the method for the demands of your scenario, as you will do later in this tutorial.

For instance, you could insert an Aircraft ( ) object (a Scenario object) using the Insert Default method. Then, you could insert a Sensor ( ) object (an Attached object) onto that aircraft.

Satellite object

The Satellite object models the properties and behavior of a vehicle in orbit around a central body.

Inserting using From Standard Object Database method

1. Select Satellite ( ) in the Insert STK Objects tool.
2. Select the From Standard Object Database ( ) method.
3. Click Insert... .
4. In the Name or ID field, enter 25544 (the SSC number or Satellite Catalog Number).
5. Click Search .
6. In the Results field, select ISS (Zarya).
7. Click Insert .

The ISS (Zarya) is downloaded from AGI's Standard Object Data Service. If you were to choose ISS or Zarya from the Local Database, you would need to ensure you've updated your satellite database.

8. Click **Close** to close the Search Standard Object Data window.

Viewing in 2D

1. Bring the 2D Graphics window to the front.
2. Zoom Out ( ) if needed to see the entire earth and the current orbit track of ISS_ZARYA_25544 ( ).
3. Using the Animation Toolbar, make any adjustments to the time step.
4. Click Start ( ) to animate the scenario.
5. When finished, click Reset ( ).

Viewing in 3D

1. Bring the 3D Graphics window to the front.
2. Click Home View ( ).
3. Right-click on ISS_ZARYA_25544 ( ) in the Object Browser.
4. Select Zoom To.
5. Using the Animation Toolbar, you can make adjustments to the time step.
6. Click Start ( ) to animate the scenario.
7. When finished, click Reset ( ).

Viewing ISS_ZARYA_25544's properties

1. Right-click on ISS_ZARYA_25544's ( ) in the Object Browser.
2. Select properties ( ).
3. The Propagator is set to the SGP4 Propagator. The STK software uses the Simplified General Perturbations (SGP4) Propagator with two-line mean element (TLE) sets.

4. In the TLE Source panel, click **Preview...** . Here, you can preview or modify the two-line element (TLE) information that the STK software uses to propagate the SGP4 satellite.
5. **Close** the TLE Preview window.
6. Click **OK** to close ISS_ZARYA_25544's ( ) Properties ( ).

Using Orbit Wizard

The Orbit Wizard is a satellite-level tool designed to assist you in either creating any one of several standard orbits or designing your own satellite orbit. The configurable options available depends on the orbit type selected.

1. Select Satellite ( ) in the Insert STK Objects tool.
2. Select the Orbit Wizard ( ) method.
3. Click **Insert...** .
4. When the Orbit Wizard opens, set the following:

Option	Value
Type	Repeating Ground Trace
Satellite Name	RGT_Sat
Approximate Altitude	600 km

5. Click **OK** .
6. Just like you did with ISS_ZARYA_25544 ( ), view RGT_Sat ( ) in the 2D Graphics and 3D Graphics windows.
7. Click Start ( ) to animate the scenario.
8. When finished, click Reset ( ).

Viewing RGT_Sat's properties

1. Right-click on RGT_Sat ( ) in the Object Browser.
2. Select properties ( ).

3. You can see that the propagator is set to J2 Perturbation.

   Unlike a Satellite ( ) object propagated using TLEs, you can edit a Satellite ( ) object propagated using the Orbit Wizard using your own data.

Showing satellite vectors and attitude

Sometimes you need to understand your attitude.

1. Return to RGT_Sat's ( ) Properties ( ).
2. Select the 3D Graphics - Vector page.
3. Select the Axes tab.
4. Select the Show check box for Body Axes.
5. Change the Body Axes color.
6. Click **Apply** .

Viewing in 3D

1. Right-click on RGT_Sat ( ) in the Object Browser.
2. Select Zoom To.
3. Bring the 3D Graphics window to the front.

Inserting an Aircraft object

The Aircraft ( ) object models the properties and behavior of a vehicle that travels in a great arc route, generally above the surface of the earth.

1. Bring the STK Insert Objects tool to the front.
2. Insert an Aircraft ( ) object using the Insert Default ( ) method.

Renaming objects

It's usual to rename objects in the STK application. If you choose not to rename your objects, the STK software will name them, giving each object a number.

1. Right-click on Aircraft1 ( ) in the Object Browser.
2. Select Rename in the shortcut menu.
3. Rename Aircraft1 ( ) to My_Plane.

Changing the Aircraft object's properties

1. Open My_Plane's ( ) Properties ( ).
2. Select the Basic - Route page.

Clicking on a map

1. Leave My_Plane's ( ) Properties ( ) open.
2. Bring the 2D Graphics window to the front by clicking on the 2D Graphic tab below the Properties Browser.
3. Click somewhere in the Atlantic Ocean near the African coast and then click somewhere in the Indian Ocean near the Indian coast.
4. Bring My_Plane's ( ) Properties ( ) back to the front.

Modifying a waypoint location

1. Select the first waypoint.
2. Select the **Clicking on map changes current point** check box.
3. Bring the 2D Graphics window back to the front.
4. Click somewhere in the Pacific Ocean near the United States coast.
5. Bring My_Plane's ( ) Properties ( ) back to the front.

6. Click Delete Point until all waypoints are deleted.
7. Clear the **Clicking on map changes current point** check box.
8. Click Apply .

Inserting a point

You can insert waypoints manually. This is more precise than clicking on the map.

1. Click Insert Point .
2. Enter the following by clicking in the associated cell. Press the Enter key on the keyboard after each entry:

Option	Value
Latitude	38.00 deg
Longitude	-120.00 deg

3. Click Insert Point .
4. Enter the following for the second waypoint:

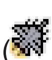
Option	Value
Latitude	30.00 deg
Longitude	-99.00 deg

5. Click Insert Point .
6. Enter the following for the third waypoint:

Option	Value
Latitude	40.00 deg
Longitude	-77.00 deg

7. Click Apply .

Viewing the waypoints in the 2D and 3D Graphics windows

My_Plane's () flight route is located in the Continental United States.

1. Bring the 2D Graphics window to the front to view the flight route.
2. Bring the 3D Graphics window to the front to view the flight route.
3. Return to My_Plane's ( ) Properties ( ).

Changing a waypoint's altitude

When modifying one waypoint, you need to know proper units. For instance, look at the first waypoint:

- The Altitude unit uses km (kilometers).
- The Speed unit uses km/sec (kilometers per second).
- The Turn Radius unit uses km.

Change the first waypoint's altitude to 20000 ft (feet).

1. Select the first waypoint.
2. Click inside the Altitude cell.
3. Enter 20000 ft.
4. Press Enter on the keyboard.

Changing a waypoint's speed

Change the first waypoint's speed to 450 mi/hr (miles per hour).

1. Click inside the Speed cell.
2. Enter 450 mi/hr.
3. Press Enter on the keyboard.

Changing a waypoint's turn radius

Modify the first waypoint's turn radius to two (2) km.

1. Click inside the Turn Radius cell.
2. Enter 2 km.
3. Press Enter on the keyboard.
4. Click **Apply** .

Using the Set All Grid Values tool

1. Click **Set All...** .
2. Select Altitude, Speed, and Turn Radius in the Set All Grid Values dialog box.

   To the right of each value is a shortcut menu. These appear throughout the STK application's User Interface (UI). By clicking on them here, you can set units for Altitude, Speed, Acceleration, or Turn Radius. This comes in handy when you can't find or don't know the acronym for the unit you are looking for.
3. Extend the shortcut menu for Speed.
4. Select nm (nautical mile).
5. In the second shortcut menu, select hr (hour).
6. Set the value to 500. The aircraft flies from the first to the last waypoints at a speed of 500 nautical miles per hour.
7. Set the following for Altitude and Turn Radius:

Option	Value
Altitude	45000 ft
Turn Radius	3 km

8. Click **OK** to close the Set All Grid Values dialog box.
9. Click **Apply** .

Setting the animation time

You can jump to a waypoint in the 2D and 3D Graphics windows.

1. Select the second waypoint.
2. Click inside the Time cell.
3. Copy (Ctrl + C) the Time of the second waypoint.
4. Right-click in the Current Scenario Time field in the Animation Toolbar.
5. Select Paste.
6. Click the Enter key on your keyboard.

Viewing in 3D

1. Bring the 3D Graphics window to the front.
2. Right-click on My_Plane ( ) in the Object Browser.
3. Select Zoom To.
4. Using the left mouse button, create a head-on view of My_Plane ( ).

   All three waypoints are using the same values. The units converted back to the default units. The aircraft is flying the entire route at an altitude of 45000 feet MSL, at a speed of 500 nautical miles per hour and will require a turn radius of 3 kilometers at waypoint two. My_Plane ( ) is in the middle of the turn. Notice that there is no roll (banking) associated with the attitude.

Attitude Profiles

The STK application is set by default to the Standard attitude option, which enables you define a vehicle's attitude profile.

You can apply attitude changes to all moving objects in STK: Aircraft, Ground Vehicles, Missiles, Satellites, and Ships.

1. Leave the view in the 3D Graphics window.
2. Return to My_Plane's ( ) Properties ( ).
3. Select the Basic - Attitude page.

Choosing the Coordinated Turn attitude

This profile computes the roll (banking) of an aircraft based on a balancing of the forces acting on the aircraft, assuming a zero angle of attack and no slip condition.

1. Open the Type drop-down list in the Basic panel.
2. Select Coordinated Turn.
3. Leave the Time Offset at 10 sec.
4. Click **Apply** .

Viewing in 3D

1. Bring the 3D Graphics window back to the front.
2. Click Reset ( ) in the Animation Toolbar.

Setting 2D Graphics properties

You can make changes to the way the STK software displays your aircraft's path.

1. Return to My_Plane's ( ) Properties ( ).
2. Select the 2D Graphics - Attributes page.
3. Change Color, Line Style, and Line Width to whatever selections you want.
4. Click **Apply** .

Setting 3D Graphics properties

Use the 3D Graphics Properties - Vector to do the following:

- Control the display of vectors and other geometric elements, such as axes and angles, related to the Earth or other central body in the selected 3D Graphics window.
- Control the display of vectors and other geometric elements related to the selected object.

Follow these steps:

1. Return to My_Plane's ( ) Properties ( ).
2. Select the 3D Graphics - Vector page.
3. Ensure the Vectors tab is selected.
4. Select the Show check box for Sun Vector.
5. Select the Axes tab.
6. Select the Show check box for Body Axes.
7. Click **Apply** .

Viewing in 3D

1. Bring the 3D Graphics window to the front.
2. If needed, Zoom To My_Plane ( ).

Animating the scenario

The body axes remain fixed throughout the flight. Even when the Sun drops below the horizon, the sun vector continuously stays locked on the Sun.

1. Decrease Time Step ( ) to five (5.00) seconds in the Animation Toolbar.
2. Click Start ( ) to animate the scenario.
3. Click Reset ( ) when finished.

Using a 3D Graphics model

You can specify a model to represent a given vehicle, facility, place, or target in the 3D Graphics window.

Select a model for your aircraft.

1. Return to My_Plane's ( ) Properties ( ).
2. Select the 3D Graphics - Model page. The current model you see in the 3D Graphics window is the aircraft.glb.
3. Click the Model File ellipsis (...) in the Model panel. All the models shown in the File dialog box come with the STK software install.
4. Select any model you'd like to use in the scenario.
5. Click **Open** to change to your selected model.
6. Click **OK** to apply your changes and close the Properties Browser.
7. Bring the 3D Graphics window to the front. You have a new model type representing the Aircraft ( ) object.

Area Target object

Using the Insert Default method

Insert Default adds the selected object and applies the default settings to the newly added object.

1. Insert an Area Target ( ) object using the Insert Default ( ) method.
2. Rename AreaTarget3 ( ) to Local_Airfield.

Defining the Area Target

Specify the location of the Area Target.

1. Open Local_Airfield's ( ) Properties ( ).

   The Basic - Boundary page is similar to the Area Target Wizard except you don't get a small 2D Graphic view. Now you'll outline a local airfield. Maybe you're trying to determine when a Satellite ( ) object or an Aircraft ( ) object sees the airfield.
2. Select the Basic - Boundary page.
3. Click **Add** four (4) times.
4. Set the following Latitude and Longitude values for the Points in the order shown (simply copy and paste from the tutorial):

Latitude	Longitude
38.4345 deg	-105.116 deg
38.4267 deg	-105.0999 deg
38.4256 deg	-105.1008 deg
38.4333 deg	-105.117 deg

5. Click **OK** .

Viewing in 3D

1. Bring the 3D Graphics window to the front.
2. Zoom To Local_Airfield ().
3. Use your mouse to get a good view of the area target outlining the airfield.

Inserting a Facility object

The Facility object () models a ground station or other facility on the surface of the central body. Similar to Place ( ) and Target () objects property wise, what makes a Facility () object stand out is when you insert one using the standard object database.

1. Insert a Facility () object using the From Standard Object Database () method.

Entering White Sands - SULF

1. Change Network back to no selection (-).
2. Enter White Sands in the Name field.
3. Click **Search** .
4. In the Results list, select White Sands - SULF.
5. Click **Insert** .
6. Click **Close** to close the Search Standard Object Data dialog box.

Inserting a Missile object

The Missile () object models the properties and behavior of a vehicle following an elliptical path that begins and ends at the surface of the central body.

1. Insert a Missile ( ) object using the Insert Default ( ) method.
2. Rename Missile1 ( ) to Sounding_Rocket.
3. Open Sounding_Rocket's ( ) Properties ( ).
4. Select the Basic - Trajectory page.
5. Notice the Propagator is Ballistic.

Using the Ballistic Propagator

The Ballistic Propagator defines an elliptical path that begins and ends at the Earth's surface. Specifying a fixed flight time, initial velocity or altitude can further refine the shape of the trajectory.

1. Return to Sounding_Rocket's ( ) Properties ( ) Basic - Trajectory page.
2. Set the following:

Option	Value
Launch Latitude - Geodetic	33.7212 deg
Launch Longitude	-106.7364 deg
Launch Altitude	4750 ft (compensate for terrain)
Change Fixed Delta-V to Fixed Apogee Alt	100 km
Impact Latitude - Geodetic	32.9 deg
Impact Longitude	-106.3 deg
Impact Altitude	3910 ft (compensate for terrain)

3. Click **OK** .

Viewing in 3D

1. Bring the 3D Graphics window to the front.
2. Zoom To Sounding_Rocket ( ).
3. Click Increase ( ) Time Step in the Animation Toolbar to set the X Real Time Multiplier to 8.00.

4. Click Start ( ) to animate the scenario.
5. Click Reset ( ) when finished.

Inserting a Place object

1. Insert a Place ( ) object using the Insert Default ( ) method.
2. Zoom To Place1 ( ).
3. Bring the 3D Graphics to the front.

   A Place ( ) object or a Facility ( ) object will default to AGI Headquarters.

Moving Place1 by clicking the 2D Graphics window

1. Bring the 2D Graphics window to the front.
2. Zoom to Place1 ( ). Remember, in the 2D Graphics window, you have to use your mouse to center and zoom to an object!
3. Open Place1's ( ) Properties ( ).
4. Return to the 2D Graphics window.
5. Click the upper-left corner of the building.

Setting the height above ground of the facility

Set Place1 ( ) 30 feet above the ground to model the sensor on the roof of the building.

1. Return to Place1's ( ) Properties ( ).
2. Enter 30 ft in the Height Above Ground field.
3. Click OK .

Using the 3D Object Editor

Use the 3D Object Editor to define and modify the position of an area target, facility, place, aircraft, ground vehicle, ship, or target in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Orient from Top ( ) in the 3D Graphics window toolbar.
3. Zoom To Place1 ( ).
4. Open the 3D Editing Object drop-down list.
5. Select Place/Place1.
6. Click Object Edit Start/Accept ( ) to start the editing process.
7. Press the Shift key on your keyboard.
8. Left-click in the center of the building.
9. Click Object Edit Cancel ( ) because that is not where you want to object to go.

Fixing the location

1. Click Object Edit Start/Accept ( ) to start the editing process.
2. Press the Shift key on your keyboard.
3. Left-click in the upper-right corner of the building.
4. Click Object Edit Start/Accept ( ) to accept the change.

Adding a Sensor object

The **Sensor** ( ) object models the field of view and other properties of a sensing device attached to another STK object.

1. Insert a Sensor ( ) object using the Insert Default ( ) method.
2. Select White_Sands-SULF ( ) in the Select Object dialog box.

3. Click **OK** .

4. Rename Sensor1 () to WS_Sensor.

Adjusting the Sensor orientation

1. Open WS_Sensor's () Properties ().

2. Select the Basic - Definition page.

3. In the Simple Conic panel, enter 90 deg in the Cone Half Angle field.

4. Click **Apply** .

Setting constraints

1. Select the Constraints - Active page.

2. Note the currently used constraints.

   - Line of Sight: Access to the object is limited to lines of sight not obstructed by the ground, which in this instance is the central body.

   - Field-of-View (Sensors only): Access is denied if the associated object is not within the field of view as defined by the angle settings for the sensor.

3. Click Add new constraints () in the Active Constraints toolbar.

4. Select Range in the Constraint Name list when the Select Constraints to Add dialog box opens.

5. Click **Add** .

6. Click **Close** to close the Select Constraints to Add dialog box.

7. Select the Max check box in the Range panel.

8. Enter 1000 km in the Max field.

   Range is measured as the distance between the two objects.

9. Click **OK** .

Closing the scenario

Save your work and close out of your scenario.

1. Close any open windows except for the 2D and 3D Graphics windows.
2. Save ( ) your work.
3. Select the File menu in the Menu Bar.
4. Select Close to close your scenario without closing the STK application.

Summary

This tutorial began with understanding the purpose of the Insert STK Objects tool. Next, the Scenario ( ) object and its properties were discussed. This was followed by an in-depth discussion of commonly used object in the Insert STK Objects tool, how to insert objects into the scenario, and which method to use during that insertion.

Part 3: Access Reports and Graphs

> **Note:** Visit **help.agi.com/stk/#training/Day1Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators often need to determine the times one object can "access" (or see) another object. In addition, they need to impose constraints on accesses between objects to define what constitutes a valid access. These constraints could include elevation angle, sunlight or umbra restrictions, gimbal speed, range, and more. Engineers also require the ability to create reports and graphs that summarize static data.

Solution

With the STK software, you can determine accesses between objects and generate reports to summarize your data. Building on your fundamental understanding of the STK software, use two important tools in the STK application – the Access tool and the Report & Graph Manager – to solve this problem.

Creating the satellite tracking station

A teleport facility, which is used to track satellites, is located in Castle Rock, Colorado.

Inserting a new Facility object

Insert Castle Rock Teleport into the scenario as a Facility object.

1. In the Insert STK Objects tool ( ), select Facility ( ) in the Select An Object To Be Inserted list.
2. Select From Standard Object Database ( ) in the Select a Method list.

3. Click **Insert...** .
4. Enter Castle Rock in the Name field when the Search Standard Object Data dialog box opens.
5. Click **Search** .
6. Select the entry in the Results list whose Facility Name is Castle Rock Teleport and whose Network is INTELSAT.
7. Click **Insert** .
8. Click **Close** to close the Search Standard Object Data dialog box.

Viewing the tracking station in 3D

View the Castle Rock Teleport in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on Castle_Rock_Teleport ( ) in the Object Browser.
3. Select Zoom To in the shortcut menu.
4. Use your mouse to get a good view of Castle_Rock_Teleport ( ) and the surrounding terrain.

Streaming terrain from a Terrain Server

For the purposes of this scenario, disable Terrain Server.

1. Right-click on on your Scenario object ( ) in the Object Browser.
2. Select Properties ( ) in the shortcut menu.
3. Select the Basic - Terrain page when the Properties Browser opens.
4. Clear the Use terrain server for analysis check box in the Terrain Server panel.
5. Click **OK** to accept your changes and to close the Properties Browser.
6. Bring the 3D Graphics window to the front.

Editing Castle Rock Teleport's properties

Castle_Rock_Teleport now appears to be floating above the terrain surface by several kilometers. The STK application is still referencing the Facility object's altitude based on the facility's altitude from the Standard

Object Database.

1. Open Castle_Rock_Teleport's ( ) Properties ( ).
2. Select the Basic - Position page when the Properties Browser opens.
3. Select the Use terrain data check box in the Position panel.
4. Click **Apply** to accept your changes and to keep the Properties Browser open.
5. Return to the 3D Graphics window.

Understanding access

The simplest definition of "access" is the ability of one object to see another object during a period of time. An access is defined by two or more objects – a primary object and an associated object or objects – for which the access is computed.

Understanding access constraints

The first condition for access is geometric line of sight, meaning the ability to draw a straight line between the positions of two objects. Additionally, there may also exist other conditions for access, called constraints. Access constraints are imposed by you. You can restrict the Castle Rock Teleport Facility object's field of view by setting constraints on it.

1. Return to Castle_Rock_Teleport's ( ) Properties ( ).
2. Select the Constraints - Active page.
3. Click **OK** to close the Properties Browser without making any changes.

Attaching a Sensor object to Castle Rock Teleport

You can also restrict and visualize Castle_Rock_Teleport's field of view by using a Sensor object that models the properties of the antennas at the site.

1. Insert a Sensor ( ) object using the Insert Default ( ) method.
2. Select Castle_Rock_Teleport ( ) when the Select Object dialog box opens.
3. Click **OK** .
4. Right-click on Sensor1 ( ) in the Object Browser.

5. Select Rename in the shortcut menu.
6. Rename Sensor1 ( ) CR_FOV.

   CR_FOV is an acronym for Castle Rock field of view.

Setting the half angle of the sensor

A simple conic sensor pattern is defined by a simple cone angle.

1. Open CR_FOV's ( ) Properties ( ).
2. Select the Basic - Definition page.
3. Enter 90 deg in the Cone Half Angle field.
4. Click **Apply** to accept your changes and to keep the Properties Browser open.

Setting the Sensor's constraints

Set a Max Range constraint of 1,500 kilometers on the sensor.

1. Select the Constraints - Active page.
2. Click Add new constraints ( ) in the Active Constraints toolbar.
3. Type Range in the Search field when the Select Constraints to Add dialog box opens.
4. Select Range in the Constraint Name list.
5. Click **Add** .
6. Click **Close** to close the Select Constraints to Add dialog box.
7. Select the Max check box in the Constraint Properties Range panel.
8. Enter 1500 km in the Max field.
9. Click **OK** to accept your changes and to close the Properties Browser.

Viewing the sensor's field of view in 3D

View the sensor's constrained field of view in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom To Castle_Rock_Teleport ( ).
3. Zoom out enough to see CR_FOV's ( ) field of view.

Inserting OneWeb satellites

Use the Standard Object Database to propagate five OneWeb satellites into your scenario.

Inserting satellites from the Standard Object Database

You'll use a select subset of satellites to calculate access from your sensor. Use the options available in the Standard Object Database to limit your search to those OneWeb satellites with an inclination between 87 and 90 degrees.

1. Insert a Satellite ( ) object using the From Standard Object Database ( ) method.
2. Enter OneWeb in the Name or ID field when the Search Standard Object Data dialog box opens.
3. Scroll down to Inclination.
4. Select the Min check box.
5. Enter 87 deg in the Min field.
6. Select the Max check box.
7. Enter 90 deg in the Max field.
8. Click **Search** .

Selecting the most currently launched satellites

You will select the five most recently launched satellites.

1. In the Results list, click on the Space Surveillance Catalog Number column header.
2. You should see an upward-facing arrow.
3. Click on the arrow so it points downward.

   This should sort the list so that the newest satellites are now at the top.

4. Select the first five satellites whose Data Source is AGI's Standard Object Database.

> ⚠ **Important:** If you don't have an Internet connection, pick the first five satellites, but make sure your local install is using the most recent Two Line Element (TLE) files that are available. For information on how to obtain the most current TLE files, refer to the **Using External TLE and GPS Almanac Files tutorial**.

5. Click Insert .

6. Click Close to close the Search Standard Object Data dialog box after all five Satellite ( ) objects have been propagated.

Using the Access tool

The Access tool enables you to select the objects and the time period associated with an individual access, enables you to manage all defined accesses, and provides multiple methods to display your access results.

Computing access

You want to analyze if any OneWeb satellites pass through CR_FOV's field of view.

1. Click Access... ( ) in the STK Tools toolbar.

2. Click Select Object... to the right of the Access for field when the Access tool opens.

3. Select CR_FOV ( ) in the Select Object dialog box.

4. Click OK .

   "Access for" now shows Castle_Rock_Teleport-CR_FOV. This is the object from which you're looking.

5. Multi-select all the Satellite ( ) objects in the Associated Objects list.

6. Click Compute

Generating an Access report

In the Access tool's Reports panel, you can generate an Access report that provides access times between one object and one or more selected objects. Global statistics are provided if more than one object is selected.

1. Ensure all the satellites are selected in the Associated Objects list.
2. Click Access... in the Reports panel.
3. Scroll through the report to become familiar with the layout.
4. Leave the Access report and the Access tool open.

Generating an Access graph

In the Access tool's Graphs panel, you can generate an Access graph that provides access times between one object and one or more selected objects.

1. Return to the Access tool.
2. Click Access... in the Graphs panel.

   When you generate a graph, the zoom in function is automatically on.
3. Locate the first access in the graph.
4. Using your mouse, hold down the left mouse button and draw a box around the access.

   This can be done multiple times until the graph is filled with the one access.
5. Place the cursor at the beginning of the access. A text box will appear with information about the access start time.
6. When you are done, click Zoom Out ( ) until you see the whole graph.
7. Close the Access graph when you are finished.
8. Leave the Access tool open.

Generating an Azimuth Elevation Range (AER) report

In the Reports and Graphs panels, you can generate an AER report or an AER graph with azimuth, elevation, and range data.

1. Return to the Access tool.
2. Ensure all the satellites are selected in the Associated Objects list.
3. Click AER... in the Reports panel.

Since the access is taking place from an object on the ground, an azimuth of zero (0) degrees is True North. The elevation is based on the central body (the WGS84 ellipsoid). The range is calculated from the center point of the FROM object to the center point of the TO object. Remember, the Satellite objects must enter the Sensor object's field of view in order to be accessed.

4. Scroll through the report to become familiar with the layout.
5. Close the AER report once you are done.
6. Leave the Access tool open.

Extending CR_FOV's range

Extend the CR_FOV Sensor object's range to see how it affects your data.

1. Open CR_FOV's ( ) Properties ( ).
2. Select the Constraints - Active page when the Properties Browser opens.
3. Select Range in the Active Constraints list.
4. Enter 2000 km in the Range panel's Max field.
5. Click **OK** to accept your change and to close the Properties Browser.

Refreshing the Access report

Your Access report is showing the old data. Apply the new range constraint to the report.

1. Return to your Access report.
2. Click Refresh (F5) ( ) in the Access report toolbar. You also have the option of selecting the F5 key to refresh a report.
3. Compare your new data to your old data.
   - Do you have the same number of accesses?
   - Are your durations longer?

Changing the report's units

Change the Access report's time dimension to minutes.

1. Click Report Units ( ) in the report toolbar.
2. Select the Time Dimension in the Units Access dialog box.
3. Select Minutes (min) in the New Unit Value list.
4. Click OK .
5. The Duration is now reported in minutes instead of seconds.

Using quick reports

A quick report retains the customized object, time, and unit settings of the displayed report or graph. This enables you to easily re-create your output. Unlike .txt and .csv files, a quick report is saved inside of the STK application.

Saving a new quick report

Save your Access report as a new quick report.

1. Click Save as quick report ( ) in the Access report toolbar.
2. Close the Access report.
3. Close the Access tool.

Using the Quick Report Manager

The Quick Report Manager displays the entire list of quick reports that you have saved in your scenario. A quick report retains the customized object, time and unit settings of the displayed report or graph.

1. Click Quick Report Manager... ( ) in the Data Providers Toolbar.
2. Enter Sensor to OneWeb in the Name column when the Quick Report Manager opens.
3. Select the Enter key.

4. Clear the Show on Load check box if it's checked.
5. Click **Create** .

   This enables you to create the Access report without having to use the Access Tool.

6. Close the Access report.
7. Click **OK** to close the Quick Report Manager.

Viewing the quick report

With your quick report created, open it from the Quick Report Manager using the drop-down menu.

1. Open the Quick Report Manager ( ) drop-down menu.
2. Select Sensor to OneWeb ( ).
3. Leave the Access report open.

Using the Report & Graph Manager

You can generate data output for most STK objects using the Report & Graph Manager, which is available from the Analysis menu or the Data Providers toolbar.

Opening the Report & Graph Manager

Open the Report & Graph Manager and focus on data for a single satellite.

1. Click Report & Graph Manager... ( ) in the Data Providers toolbar.
2. Change the Object Type to Satellite in the upper-left corner of the Report & Graph Manager.
3. Select the OneWeb satellite you have been studying in the Object List.

   Multiple objects can be selected, but for this scenario, focus on your selected OneWeb satellite.

Managing report and graph styles

You can manage the report and graph styles.

1. Ensure (⊞) the Installed Styles ( ) folder in the Styles list is expanded.
2. Take a close look at the two entries for Classical Orbit Elements located in the Installed Styles ( ) folder.

   One is a graph style ( ) and one a report style ( ).
3. Notice the lock icons on each.

   The reports and graphs located in the Styles list are read only and cannot be customized. However, they can be duplicated, and those duplicates can be customized.
4. Select the Classical Orbit Elements ( ) report.
5. Click **Generate...** .
6. Take a look at the report.
7. Close the Classical Orbit Elements report when you are finished.
8. Return to the Report & Graph Manager.
9. Look at the very top of the Styles field.

   The Classical Orbit Elements ( ) report is now available in the My Favorites ( ) folder.

Understanding data providers, groups and elements

The content of any report or graph is generated from the selected data providers for the report or graph style. Data providers, data provider groups, and data provider elements are the organizing principles of the data provider functionality. The STK software provides hundreds of prebuilt reports and graphs using the data providers.

Understanding data providers by object

View the data providers used for the Classical Orbit Elements report.

1. Right-click on the Classical Orbit Elements ( ) report in the Installed Styles ( ) folder.
2. Select Duplicate ( ) in the shortcut menu.
3. Select the Content page when the Properties Browser opens.

   The left side shows all the data providers for the object type (in this case, a Satellite ( ) object) and on the right side are the Report Contents.

4. In the Report Contents list, select Classical Elements-J2000-Semi-major Axis.
5. Look in the Data Providers list.

   You can see the hierarchy of Data Provider - Data Provider Group - Data Provider Element: in this case, Classical Elements ( ) - J2000 ( ) - Semi-major Axis ( ).

Customizing reports

You can customize the properties of a static or dynamic report.

Removing the default data provider elements

Before adding any new data provider elements, remove some existing ones.

1. In the Report Contents list, select the following:
   - Classical Elements-J2000-Arg of Perigee
   - Classical Elements-J2000-True Anomaly
   - Classical Elements-J2000-Mean Anomaly
2. Click **Remove** .

Customizing the report's contents

As previously stated, you will replace argument of perigee, true anomaly and mean anomaly data provider elements with J2000 x, y, and z Cartesian Position data provider elements.

1. Remove the asterisk (*) at the top of the Data Providers list, in the Filter field.
2. Enter Cartesian in the Filter field.
3. Click **Filter** .

   This narrows down your choices to only those data providers containing Cartesian data providers, groups or elements.
4. Expand (⊞) the Cartesian Position ( ) data provider.
5. Expand (⊞) the J2000 ( ) data provider group.
6. Move ( ) the x, y, and z data providers ( ) to the Report Contents list.

7. If desired, use the up ( ) and down ( ) arrows to place elements where desired.
8. Click **OK** to accept your changes and to close the Properties Browser.

Generating a custom report

With your new report style saved, generate your custom report.

1. Expand (⊞) the My Styles ( ) folder in the Styles list.
2. Right-click on Classical Orbit Elements ( ).
3. Select Rename in the shortcut menu.
4. Rename Classical Orbit Elements ( ) My Classical Orbit Elements.
5. Click **Generate...**
6. Close the report when you are finished.

Summary

Using the Access tool, you computed an Access report, an Access graph, and then an Azimuth Elevation Range report between the CR_FOV Sensor object and the satellites. The Report & Graph Manager came next, where you learned what data providers, data provider groups, and data provider elements are, and how to create a custom report.

Part 4: Movies and Visual Data Files

> **Note:** Visit **help.agi.com/stk/#training/Day1Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem Statement

Engineers and operators often need resources to prepare visual presentations for briefings, talks, speeches and demonstrations. In this scenario, an aircraft is flying an important mission that you wish to simulate using a video and an STK Visual Data File.

Solution

Upon completion of this tutorial, you will learn how to:

- Create a simple movie that can be embedded in a PowerPoint presentation or played for an audience.
- Convert an STK scenario into a visual data file (VDF).
- Display the VDF in STK Viewer, which does not require an STK license.
- Create a first-rate briefing experience for both the presenter and the audience.

Post-typhoon aerial photography flight route

You will simulate an aircraft's flight route across Japanese airspace. Your aircraft will fly over the following islands and cities in the order shown:

- Naha, Okinawa
- Amami Oshima, Kagoshima
- Minamidaito, Okinawa
- Back to Naha, Okinawa

Inserting Naha Airport as a Place object

1. Select Place ( ) in the Insert STK Objects Tool.
2. Select the Search by Address ( ) method.

> **Note:** The Insert by Address option requires an internet connection. If you do not have an internet connection, you can select the Define Properties option and set the lat/lon manually.

3. Click Insert...
4. Type Naha Airport in the **Enter an address or other search criteria below** field in the STK: Insert by Address dialog box.
5. Select Naha Airport, Japan (latitude 26.19583, longitude 127.64583).
6. Click Insert Place(s) .

Inserting Amami Oshima as a Place object

1. Type Amami in the **Enter an address or other search criteria below** field.
2. Select Amami, Japan (latitude 28.37724 longitude 129.49374).
3. Click Insert Place(s) .

Inserting Minamidaito as a Place object

1. Type Minamidaito in the **Enter an address or other search criteria below** field.
2. Select Minamidaito, Japan (latitude 25.82889 longitude 131.23187).
3. Click Insert Place(s) .
4. Click Close to close the STK: Insert by Address dialog box.

Inserting an Aircraft object

You will insert an Aircraft ( ) object. Adjust the altitude, speed, turn radius, and attitude.

1. Insert an Aircraft ( ) object using the Insert Default ( ) method.
2. Right-click Aircraft1 ( ) in the Object Browser.
3. Select Rename.
4. Rename Aircraft1 ( ) to PhotoMission.
5. Bring the 2D Graphics window to the front.
6. Maximize the 2D Graphics window.
7. Zoom in to the 2D Graphics window so that all three Place ( ) objects are visible and centered.

Modifying the Aircraft object's properties

1. Right-click PhotoMission ( ) in the Object Browser.
2. Select Properties ( ).
3. Select the Basic - Route page when the Properties Browser opens.
4. Open the Reference drop-down menu in the Altitude Reference panel.
5. Select MSL (Mean Sea Level).
6. Bring the 2D Graphics window to the front by clicking the 2D Graphic... tab near the bottom of STK.

Creating waypoints for the Aircraft object

To keep things simple and focus on movie making and creating VDFs, you'll use the "clicking the 2D Graphics window" method to create waypoints for PhotoMission ( ). By clicking the 2D Graphics window, you won't fly to precise locations. You'll fly to the point you clicked. Start at Naha.

1. Click as close as you can to the Naha Place ( ) object symbol's point in the 2D Graphics window.
2. Click as close as you can to the Amami Place ( ) object symbol's point.
3. Click as close as you can to the Minamidaito Place ( ) object symbol's point.
4. Click as close as you can to the Naha Place ( ) object symbol's point.

   Due to the default turn radius, your waypoints won't be overhead of Amami or Minamidaito. You'll fix that shortly.
5. Return to PhotoMission's ( ) properties by clicking the PhotoMission... tab ( ) below the 2D Graphics window.

Adjusting the aircraft's altitude, speed, and turn radius

Adjust the altitude, speed, and turn radius.

1. Click **Set All...** .
2. Select the Altitude, Speed, and Turn Radius check boxes in the Set All Grid Values dialog box.
3. Set the following:

Option	Value
Altitude	10000 ft
Speed	200 mi/hr
Turn Radius	1 km

4. Click **OK** to close the Set All Grid Values dialog box.
5. Click **Apply** to save your changes and to keep the Properties Browser open.
6. Return to the 2D Graphics window.

Adjusting the attitude

Define the attitude profile for PhotoMission ( ) by using a coordinated turn.

1. Return to PhotoMission's ( ) properties ( ).
2. Select the Basic - Attitude page.
3. Open the Type drop-down menu in the Basic panel.
4. Select Coordinated Turn.
5. Click **Apply** to save your changes and to keep the Properties Browser open.

Changing the Aircraft object's model

1. Select the 3D Graphics - Model page.
2. Click the Model File ellipsis ( ) in the Model panel.
3. Select the commuter.glb file in the File dialog box.
4. Click **Open** .
5. Click **Apply** to save your changes and to keep the Properties Browser open.
6. Select the Basic - Route page.

Preparing the Scene

There are many ways to create a movie inside of STK. For the purposes of this movie, you'll use just one of them.

The second point (waypoint) is when PhotoMission ( ) is over Amami_Japan ( ). You will make a movie that begins ten (10) seconds prior to and ends ten (10) seconds after the waypoint's time. Jump to the waypoint's time. Your time will be different than the time showed in the example.

1. Go to the Time cell of point two.
2. Click in the cell to select the time.
3. Press Ctrl + C (copy) on your keyboard.
4. Highlight the time in the Current Scenario Time field in the Animation Toolbar.
5. Press Ctrl + V (paste) on your keyboard.
6. Press the Enter key on your keyboard.

Viewing in 3D

1. Bring the 3D Graphics window to the front
2. Right-click PhotoMission ( ) in the Object Browser.
3. Select Zoom To.
4. Using your mouse, set up the view so that you can see both PhotoMission ( ) and Amami_Japan ( ).

Setting Animation Time

Earlier, you set your animation time to the second waypoint, which is in the Current Scenario Time field. You will start the movie 10 (ten) seconds prior to that time. The following is an example of what to do:

- Assume the time in the Current Scenario Time field is **1 Mar 2023 23:56:20.123**.
- Round down ten seconds earlier to the nearest integer **1 Mar 2023 23:56:10.000**.

Now apply this to your scenario.

1. Round down the time by ten seconds in the Animation Toolbar Current Scenario Time field.
2. Press Enter on your keyboard.
3. Make sure you're satisfied with your view. Once you start recording the movie, do not touch the 3D Graphics window.

Creating a Stored View

1. Click Stored Views ( ) in the 3D Graphics window toolbar.
2. Click **New** in the Stored View: 3D Graphics 1 - Earth dialog box.
3. Set the View Name to "Movie Time".
4. Click **OK** to close the the Stored View: 3D Graphics 1 - Earth dialog box.

   By creating this view, if you were to Reset ( ) your scenario, you can quickly jump back to this time and view in the 3D Graphics window.

5. Click X Real-time Animation Mode ( ) in the Animation toolbar.

Using the Movie Timeline Plugin

The Movie Timeline Plugin helps you record animations from STK. You can also record a movie using 3D Graphics window properties where It is found on the Record Movie page. In this scenario, use the Movie Timeline Plugin. There are a lot of features in the Movie Timeline Plugin, but to keep things simple, you'll use the Record from the Movie Timeline button.

1. Open the View menu at the top of STK.
2. Select Toolbars in the drop-down menu.
3. Select Movie Timeline in the second drop-down menu. You should now see the Movie Timeline toolbar.
4. Click Record ( ) in the Movie Timeline tool bar.

Movie Wizard

The Movie Wizard will help you walk through important steps in creating your movie.

On the left side are pages starting with Filename & Format. For a straightforward way to make a movie, follow the sequence below from top to bottom.

Choosing a directory and format for your movie

Save your movie in your scenario directory using H.264 format.

1. Select the Filename & Format page.
2. Click Save as... in the **Choose the filename to use during movie recording** panel.
3. Browse to your scenario folder (e.g., C:\Users\<username>\Documents\STK_ODTK 13\STK_Movies_VDFs).
4. Change **File name** to any name you desire (e.g., First Movie.h264).
5. Click Save .

6. Leave the format as a H.264 (.mp4).
7. Click **Next** .

Selecting a graphics window to record

You can select which window you are recording. You're recording the 3D Graphics window.

1. Ensure the 1 - 3D Graphics 1 - Earth option is selected.
2. Click **Next** .

Changing the resolution

Here, you resize the 3D Graphics window using preset sizes.

1. Open the Preset drop-down menu in the **Choose the resolution of the recorded movie** panel.
2. Select Large.
3. Click **Next** .

Setting the video time and length

Take your time on this page. It's important to follow these steps:

1. Clear the black lock ( ) check box.
2. Copy (Ctrl + C) the time in the Current Scenario Time field.
3. Paste (Ctrl + V) it in the **Start time** field of the Movie time range section.
4. Highlight the time and units in the Shorter / Longer field of the Movie playback length section.
5. Enter the value 20 into the Shorter / Longer field.
6. Press the Tab key on your keyboard. This will automatically enter sec (seconds) and change the End Time in the Movie time range section to twenty (20) seconds past the Start time.
7. Select the black lock ( ) check box.
8. Click **Next** .

Setting the video size and quality

In the 3D Graphics window, aliasing is the process by which smooth curves and other lines become jagged because the resolution of the graphics device or file is not high enough to represent a smooth curve. Antialiasing is a technique for diminishing jagged stair-step-like lines that you want to be smooth.

1. Extend the Anti-aliasing drop-down menu to view the settings.
2. Leave the default setting of 3x3 - Good Quality.
3. In the **Recorded image quality, and estimated file size** field, change Quality to High Quality @ 1080. This adjusts the bitrate and changes the estimated maximum size of the recorded movie.
4. Click **Next** .

Starting your recording

Here you can view the summary and go back to make any changes you feel are necessary. Remember, it's a good idea not to manipulate the 3D Graphics window once the recording begins.

1. Click **Begin Record** ( ) once you are prepared to record. Sit back and wait until the recording is complete.
2. Click **Yes** when you are prompted to view the recording.
3. When you are finished admiring your Oscar-winning creation, close the media player and return to STK.
4. Click **Finish** in the Movie Wizard.

Resetting the scenario animation time

In the Movie Wizard, creating the start and stop time of your movie sets those times in the Scenario ( ) objects Basic - Time Animation field. It's a good idea to reset these time so that you can animate your complete analysis time (in this case four hours).

1. Open STK_Movies_VDFs ( ) properties ( ).
2. Select the Basic - Time page.
3. Select the following check boxes in the Animation panel:

- Use Analysis Start Time
- Use Analysis Stop Time

4. Click OK to select your changes and close the Properties Browser.

Converting a Visual Data File (VDF)

You can convert your STK 3D scenarios to Visual Data (.vdf) files so that you can do the following:

- **Display them in STK Viewer**:
- **Load and edit them in STK Viewer**:
- **Load, edit, and save them as a scenario file (.sc) in STK**:

Creating a VDF

The VDF Setup tool is small, but there's a lot going on.

1. Open the File menu at the top of STK.
2. Select VDF Setup...
3. Select the Exclude Install Files check box when the VDF Setup for Scenario STK_Movies_VDFs dialog box opens.
4. Click Yes for each time the Question window appears.
5. Look at Custom/User data. Use this to select the scenario files and non-STK files (.doc and .XLS files, for example) to be included with the VDF file.
6. Look at Quick Reports For STK Viewer. These reports will be available if using STK Viewer during your presentation.
7. Click Create VDF...
8. Ensure the **Save in** location points to your scenario folder.
9. Click Save .

Summary

You learned how to enable the Movie Timeline Plugin and use all of its functionality by recording a movie. Finally, you were introduced to the functionality of STK Viewer and its power in delivering a top-notch briefing to your audience.

Part 5: Introduction to Connect

> **Note:** Visit **help.agi.com/stk/#training/Day1Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Programmers, engineers, and operators often need resources to easily build applications that communicate with the STK software. Having a spreadsheet containing data, they need to transfer the data into and out of the STK application quickly and easily. Many of the engineers and operators are not programmers. They need a simple to understand language and syntax with which to create ways to efficiently populate objects into the STK application and extract data from or create automations with the STK application.

Solution

You will learn what an STK Connect command is and how to find and use the Connect Command Library. Using the STK software's *Integration* capability and the API Demo Utility, explore the Connect Command Library and become familiar with some simple connect commands used to quickly and easily populate objects into and obtain analytical data from an STK scenario.

Creating a new scenario

Create a new scenario.

1. Launch the STK application ( ).
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following in the STK: New Scenario Wizard:

Option	Value

Name	STK_Connect
Location	Default
Start	Default
Stop	Default

4. Click **OK** once you are done.
5. Click Save () once the scenario loads.
6. Close the Timeline View once the STK application opens.

Using the Web Browser

In the STK application, the Web Browser is a web browser that is an integrated part of the workspace.

1. Click New Integrated Web Browser () on the Default toolbar.
2. Click Browse () in the Web Browser - AGI - Resources web browser toolbar.
3. Click **Example HTML Utilities** in the Open dialog box.
4. Select the STK Automation folder.
5. Click **Open** .
6. Select the API Demo folder.
7. Click **Open** .
8. Select API Demo Utility.
9. Click **Open** .

Using the New command

The New command allows you to create a new scenario or add a new object to the current scenario. The Syntax is:

```
New <ApplicationPath> <ClassPath> <NewObjectName> {NewOptions}
```

1. Select Add facility in the In the API Demo Utility's Examples list.
2. Break down the string of text located in the Code Sandbox:

String of Text in Add Facility	Meaning
`New`	Connect command
`/`	ApplicationPath
`*/Facility`	ClassPath
`MyFacility_Con`	NewObjectName

3. Click **Run Code** .

A Facility ( ) object has been inserted into the scenario using its default properties.

Understanding the Connect Command Library

There is a whole section in STK Help devoted to Connect. You can use the commands in the Connect Command Library to easily build applications that communicate with the STK software.

1. Select the Help menu in the Menu Bar.
2. Select STK Help.
3. Select Integrating with STK in the table of contents.
4. Select Connect Command Library.

Viewing the Alphabetical Listing of Connect Commands

The Alphabetical listing includes all Connect commands, regardless of their groupings.

1. Select the Alphabetical Listing of Connect Commands link on the Connect Command Listings page.
2. Select N at the top of the page.
3. Select the New command.

The page describes the command, syntax, related commands, options, and examples. Whenever you're starting to write Connect commands, look at the examples. You'll have to change them to work with your object types and such, but it's a great place to start.

Viewing the SetPosition command page

You can use the SetPosition command to set the position of a facility, place, target or area target.

1. Return to the Alphabetical Listing of Connect Commands.
2. Select S at the top of the page.
3. Select SetPosition (Facility, Place, Target & Area Target).
4. Ensure you have the correct syntax:

```
SetPosition <ObjectPath> [{Type}] {CoordType} <Parameters>
```

5. Scroll down the page until you find the chart of coordinate types and parameters in the Description section.
6. Note the Geodetic coordinate type parameters:

```
<Lat> <Lon> {<Altitude> | Terrain} [MSL].
```

7. Return to the STK application.

You don't have to keep jumping back and forth between the STK application and the Connect Command Library. However, feel free to go there on your own as you proceed through the training.

Understanding the SetPosition command

Before you run the code, look at the SetPosition command in the API Demo Utility.

1. Select Modify facility in the In the API Demo Utility's Examples list.
2. Right-click on MyFacility_Con ( ) in the Object Browser.
3. Select Properties ( ).
4. Select the Basic - Position page.
5. Notice the SetPosition code in the Code Sandbox matches the Connect command syntax (e.g. Type, Latitude, Longitude, Altitude).
6. Click Cancel to close the Properties Browser.
7. Return to the API Demo Utility.

Modifying the SetPosition command

Update the default SetPosition code and examine the second command before running it.

1. Remove `0.0` (which is the altitude) at the end of the SetPosition Connect command.
2. Enter `Terrain` at the end of the Connect command.

   Since you have streaming terrain from Terrain Server, by removing the altitude and inserting `Terrain`, the Facility object will be placed on top of the terrain at the coordinates in the command.

   The second command is another New command. Note the path. It follows the object type to object name like you see it in the Object Browser. The New command inserts a Sensor ( ) object and attaches it to the MyFacility_Con ( ).
3. Click **Run Code** .

MyFacility_Con ( ) has new coordinates and an attached Sensor ( ) object named MyFacSensor_Con.

Inserting a Satellite object

Using the API Demo Utility to add a Satellite ( ) object to the scenario and name it MySat.

1. Select Add satellite In the API Demo Utility's Example list.
2. Change the Satellite ( ) object's name from `MySatellite_Con` to `MySat`.
3. Click **Run Code** .

MySat ( ) has been added to the scenario, but its orbit still needs to be propagated.

Propagating MySat

The SetState Classical command sets the orbit state of the satellite using classical coordinates. The Syntax is:

```
SetState <VehObjectPath> Classical {Propagator} {NoProp | {TimeInterval}} <StepSize>
{CoordSystem} "<OrbitEpoch>" <SemiMajorAxis> <Eccentricity> <Inclination>
<ArgOfPerigee> <RAAN> <MeanAnom>
```

1. Select Modify satellite in the API Demo Utility's Examples list.
2. Compare the text in the Code Sandbox to the syntax and change the following:

Option	Value
Object name	`MySat`
Inclination	`45`
RAAN	`180`

3. Click **Run Code** .

Inserting an Aircraft object

Use the API Demo Utility to insert a new Aircraft ( ) object.

1. Select Add aircraft in the API Demo Utility's Examples list.
2. Look at the code in the Code Sandbox.

   The New command inserts the Aircraft ( ) object named MyAircraft_Con. Then it sets the propagator to Great Arc, reference mean sea level (MSL) and create waypoints.

Command	Description
SetPropagator	Sets the propagator of a great arc vehicle.
AltitudeRef	Sets the altitude reference for a Great Arc vehicle.
AddWaypoint	Adds waypoints to a great arc vehicle.

3. Click **Run Code** .

Modifying MyAircraft_Con

Run the Modify aircraft example code to update MyAircraft_Con.

1. Select Modify aircraft in the API Demo Utility's Examples list.
2. Look at the code in the Code Sandbox.

The Waypoints command and the Clear WaypointOption clears the original waypoints and AddWaypoint inserts new waypoints.

Command	Description
Waypoints	Clears all waypoints from a great arc vehicle.

3. Click Run Code .

Computing access

Compute an access from MySat ( ) to MyFacility_Con ( ).

1. Select Compute access in the API Demo Utility's Examples list.
2. Look at the code in the Code Sandbox.

Command	Description
Access	Calculates access intervals between two objects.

3. Change the Satellite name from `MySatellite_Con` to `MySat`.
4. Click Run Code .
5. Bring the 2D Graphics window to the front. You will see access lines above MyFacility_Con ( ).
6. Look at the API Demo Utility - Output.

You can see the full path to each object followed by individual accesses.

Adding vectors

Add a vector which points from the Facility ( ) object to the Satellite object.

1. Select Add vectors in the API Demo Utility's Examples list.
2. Look at the code in the Code Sandbox.

Command	Description

VO SetVectorGeometry	Defines the display of geometric components in the 3D Graphics window.
VO View	Sets parameters for the view in a 3D window.

3. Change the Satellite name from `MySatellite` to `MySat` in the `VO SetVectorGeometry` code line.
4. View both lines of code.
5. Click **Run Code** .
6. Bring the 3D Graphics window to the front.
7. Note that the VO View command set the view in the 3D Graphics window to be centered on MyFacility_Con ( ).

   To better view the vector pointing from MyFacility_Con ( ) to MySat ( ), you might have to set your view in the 3D Graphics window so that you're under the terrain. The vector is only visible above the terrain when MyFacility_Con ( ) has an access to MySat ( ).
8. Click Start ( ) to animate the scenario.
9. Click Reset ( ) when finished.
10. Save ( ) your work.

Summary

This was an introduction to the STK Connect module and the Connect Command Library. You were introduced to the API Demo Utility which is just one of a few tools that come with the STK application, which you can use to send Connect commands.

- **STK ButtonTool**: The STK ButtonTool is a Perl script with a graphical user interface (GUI) that you can use to associate STK Connect commands with buttons.
- **Send_A_Connect_Command.htm**: Located in the Example HTML Utilities folder. This is a utility that allows you to send a Connect command to the STK application by typing it into a text field and submitting it. The Alphabetical Listing of Connect Commands can be displayed to help look up different commands.

Become Level 1 - STK Certified

Once you have completed Fundamentals training (the Level 1 - Beginner tutorials), you will be well prepared to complete the STK Level 1 - STK Certification test. The STK Certification is the first level of certification and validates your ability to perform fundamental skills needed to be productive with the Ansys Systems Tool Kit® (STK®) digital mission engineering software.

What's in the test?

The STK Certification test consists of a scenario development exercise; you have 30 days from the registration date to complete the test. The following objectives are tested:

- Modeling Your Systems – KML, Aircraft, Satellite, Sensor, Constraints
- Analying Your Systems – Access tool, Report & Graph Manager, Quick Reports
- Visualizing Your Data – 3D Models, Stored Views, Timeline View
- Extending the STK application – Connect and the STK Object Model
- Sharing Your Work – VDF, Movies, Snapshots

If you pass your STK Certification test, you will receive an STK Certification badge.

STK Level 2 - Advanced Tutorials

The Level 2 - Advanced tutorials are designed to take you through the Ansys Systems Tool Kit® (STK®) digital mission engineering software's advanced capabilities. They build off of the STK Level 1 - Beginner tutorials. You will take simulations from the STK Fundamentals training a step further with advanced analysis capabilities and tools to quantify and measure mission effectiveness.

Once you are Level 1 - STK Certified and have completed Comprehensive training (the Level 1 - Beginner and Level 2 - Advanced tutorials), you will be ready to take the Level 2 - STK Master Certification test.

Please note: one Level 2 - Advanced tutorial uses the Ansys ModelCenter® model-based systems engineering software, one tutorial uses the MATLAB programming platform, one tutorial uses Python, and one tutorial uses the EOIR capability, which is available with STK Premium. While these are very useful to have and understand, they are not required to become Level 2 - STK Master certified. If you do not have access to MATLAB or Python, you can open the example files in a text editor to view their syntax.

Tutorial	Capabilities	Required License	Required Install
Using Terrain, Chains, and Constellations Learn how to add terrain and imagery to a scenario and how to use Chain and Constellation objects in your analysis.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Customize Analysis with the Analysis Workbench Learn how to use the STK software's Analysis Workbench capability to build custom geometric, temporal, and logical operations through STK.	STK Pro, *Analysis Workbench*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Compute Coverage Over Regions Learn how to analyze global and regional coverage provided by various assets.	STK Pro, *Coverage*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Introduction to the AzEl Mask Tool and Volumetrics Learn how to use the AzEl Mask Tool and build a volumetric object.	STK Pro, *Analysis Workbench*, *Volumetric Analysis*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro

Performing Trade Studies with ModelCenter Learn how to use the STK Plugin for ModelCenter and the Ansys ModelCenter model-based systems engineering software to automate trade studies and parametric analyses.	STK Pro, *Analyzer*	STK Premium (Air or Space) or STK Enterprise	STK Pro + STK Premium
Introduction to Communications Learn how to use the STK software's Communications capability to simulate how transmitters and receivers work in the field.	STK Pro, *Communications*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Introduction to Radar Learn how to build radar system models to simulate and analyze system performance.	STK Pro, *Radar*, *Coverage*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Integrating STK with MATLAB Learn how to control the STK software through an external application like MATLAB.	STK Pro, *Integration*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Integrating STK with Python Learn how to control the STK appllication through an external application like Python.	STK Pro, *Integration*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Model Aircraft Missions with Aviator Learn how to model a sequence of curves parameterized by well-known performance characteristics of an aircraft.	STK Pro, *Aviator*	STK Premium (Air) or STK Enterprise	STK Pro

Introduction to the Advanced CAT Tool Learn how to use the STK software's Conjunction Analysis capability to avoid a satellite collision.	STK Pro, *Conjunction Analysis*	STK Premium (Space) or STK Enterprise	STK Pro
Design Trajectories with Astrogator Learn how to use the Ansys STK/Astrogator® capability to place a satellite in orbit.	STK Pro, *Astrogator*	STK Premium (Space) or STK Enterprise	STK Pro
Ground-based SSA with EOIR Learn how to set up an EOIR sensor in the STK application to track an object in space and determine the attitude profile of the target based on its light signature.	STK Pro, *EOIR*, *SatPro*	STK Premium (Space) or STK Enterprise	STK Pro + STK Premium

Part 6: Using Terrain, Chains, and Constellations

Note: Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Note: The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators require an expedient way to determine if local terrain is affecting visibility between ground sites and satellites for purposes such as communications, imaging, and general situational awareness. You are conducting a line-of-sight test between a constellation of GPS satellites, a ground-based test team located in mountainous terrain, a CubeSat communications satellite, and a base team located in Morton, Washington. The effects of the terrain must be taken into account. This test will be used to schedule times for the test team to transmit their location information to the base team, using GPS and a simple FM transmitter and relayed by way of the CubeSat satellite in low Earth orbit.

Solution

Use the STK application to analyze the impact of local terrain on accesses between the GPS constellation, the test team, a CubeSat satellite, and the base team. First, load a USGS Digital Elevation Model (DEM) file and convert it to a STK terrain inlay (.pdtt) file, which can be used to visualize terrain in the 3D Graphics window. Then, create a constellation of GPS satellites, use Place objects as the teams' locations, and insert a Satellite object for the CubeSat. Factor in the impact of local terrain using terrain and azimuth-elevation (AzEl) masks. Finally, model the communications links with a Chain object to create connections between all the nodes.

Creating a new scenario

Create a new scenario using the default 24-hour analysis period.

1. Launch the STK application ( ).
2. Click Create a Scenario in the Welcome to STK dialog box.

3. Enter the following in the STK: New Scenario Wizard:

Option	Value
Name	TerrainChainsConstellations
Start	Default
Stop	Default

4. Click **OK** when you finish.
5. Click Save (💾) when the scenario loads. A folder with the same name as your scenario is created for you.
6. Verify the scenario name and location in the Save As dialog box.
7. Click **Save** .

Locating the U.S. Geological Survey DEM file

Find and copy the preinstalled USGS Digital Elevation Model (DEM) file for use in your scenario. The USGS Digital Elevation Model is the standard set forth by the US Geological Survey (USGS) for handling digital elevation data for the United States and Puerto Rico.

> **Note:** The DEM file used in this tutorial is located inside a compressed (zipped) file in the install directory. Take a moment to copy the file out of the zipped file archive. Do not extract the entire archive. Follow the instructions in the tutorial to use the file.

1. Using Windows File Explorer, browse to the location of the DEM file in the install directory:
   - STK 12: C:\Program Files\AGI\STK 12\CodeSamples\CodeSamples.zip\SharedResources\Scenarios\Events
   - STK 13: C:\Program Files\AGI\STK_ODTK 13\CodeSamples\STKCodeSamples.zip\SharedResources\Scenarios\Events
2. Copy the file named hoquiam-e.dem.
3. Navigate to your scenario folder.
4. Paste the hoquiam-e.dem file into your scenario folder.
5. Close Windows File Explorer.

Using the DEM file for analysis

The STK application provides a foundation for analyzing and visualizing complex systems in the context of their missions. You'll use the DEM terrain data file you copied out of the zipped Code Samples file for both analytical and visual terrain in your scenario. The file contains digital elevation data for Mount St. Helens and the vicinity near the city of Hoquiam, Washington.

Turning off streaming terrain

By turning off streaming terrain, you're simulating what you'd see in a setting that doesn't have an internet connection.

1. Right-click on TerrainChainsContellations ( ) in the Object Browser.
2. Select Properties ( ) in the shortcut menu.
3. Select the Basic - Terrain page when the Properties Browser opens.
4. Clear the Use terrain server for analysis check box in the Terrain Server panel.
5. Click **Apply** to confirm your change and to keep the Properties Browser open.

Loading the local DEM file

Now, load the hoquiam-e.dem terrain data file on your local machine into the scenario for analysis.

1. Click **Add** in the Custom Analysis Terrain Sources panel.
2. Open the file type drop-down list when the Open dialog box opens.
3. Select USGS DEM (DEM).
4. Browse to your scenario folder
5. Select hoquiam-e.dem.
6. Click **Open** to select the file and to close the Open dialog box.
7. Click **OK** to confirm your selection and to close the Properties Browser.

Creating a terrain inlay using the Imagery and Terrain Converter

Use the Imagery and Terrain Converter to create a STK terrain inlay file (.pdtt) for a specific region. The .pdtt will have the same analytical terrain as the USGS DEM file. However, it can also be used to visualize the terrain in the 3D Graphics window.

Selecting the DEM file for conversion

Select the DEM file as the source of the terrain data.

1. Select the Utilities menu in the menu bar.
2. Select Imagery and Terrain Converter... in the Utilities menu.
3. Select the Terrain Region page when the Imagery and Terrain Converter opens.
4. Open the Terrain Source drop-down list in the Input Data panel.
5. Select the path to the hoquiam-e.dem file.

Setting the terrain output data

Specify the file location and the filename of the terrain file you will create.

1. Click the Directory ellipsis (...) in the Output Data panel.
2. Navigate to the location of your scenario (e.g. C:\Users\<username>\Documents\STK_ODTK 13\TerrainChainsConstellations when the Directory dialog box opens.
3. Click **Select Folder** to confirm your selection and to close the Directory dialog box.
4. Enter StHelensTerrain in the Filename field in the Output Data panel.
5. Click **Convert** .
6. Click **Close** to close the Imagery and Terrain Converter when finished.

Adding the terrain inlay with Globe Manager

The Globe Manager allows you to customize scenario globes with imagery and terrain data and to manage that data once it has been applied.

Opening Globe Manager

Open Globe Manager from the Globe Manager Toolbar in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Globe Manager ( ) in the Globe Manager toolbar.

Selecting the files to display in the 3D Graphics window

You can use Globe Manage to select imagery and terrain data for display in the 3D Graphics window.

1. Click Add Terrain/Imagery ( ) In the Hierarchy toolbar when Globe Manager opens.

   The Hierarchy tab is used to add central bodies, image, and terrain items to a scenario.

2. Select Add Terrain/Imagery... ( ) in the drop-down menu.
3. Open the Path drop-down list when the Globe Manager: Open Terrain and Imagery Data dialog box opens.
4. Select the path to your scenario (e.g. C:\Users\<username>\Documents\STK_ODTK 13\TerrainChainsConstellations).
5. Select the check box for StHelensTerrain.pdtt.
6. Click **Add** .
7. Click **No** when the Use Terrain for Analysis dialog box opens.

   You're already using the hoquiam-e.dem file for analysis.

Viewing the terrain inlay in the 3D Graphics window

Gain situational awareness by viewing the terrain inlay in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on StHelensTerrain.pdtt ( ) in the Globe Manager hierarchy.
3. Select Zoom To ( ) in the shortcut menu.
4. Use your mouse to view the image and surrounding terrain.

Inserting operational GPS satellites

Add the constellation of currently operational GPS satellites into your scenario.

Using the Standard Object Database

Use the Standard Object database to insert the GPS satellites in the STK application.

1. Return to the STK application.
2. Bring the Insert STK Objects tool ( ) to the front.
3. Select Satellite ( ) in the Select An Object To Be Inserted list.
4. Select From Standard Object Database ( ) in the Select A Method list.
5. Click **Insert. . .** .

Selecting operational GPS satellites

Search for operational GPS satellites using the online AGI Standard Object Data Service.

1. Clear the Data Sources - Local check box.
2. Enter GPS in the Name or ID field.
3. Select Operational in the Operational Status drop-down list.
4. Click **Search** .
5. Select the Create Constellation from Selected check box in the Insert Options panel.
6. Enter GPS_Sats in the Name field in the Insert Options panel.
7. Select all the satellites in the Results list.

8. Click Insert .

> **Note:** You can also insert the GPS satellites one at a time, if you find that to be easier. You'll have to make sure you click Yes in the dialog box that opens after clicking Insert to make sure you add all the GPS satellites to the existing GPS_Sats constellation.

9. When all the GPS satellites have propagated, click Close to close the Search Standard Object Data dialog box.

Viewing the GPS satellites assigned to the constellation

Open the Constellation object's definition page to view the objects that are grouped into the constellation.

1. Open GPS_Sats' ( ) Properties ( ).
2. Select the Basic - Definition page when the Properties Browser opens.

You can see that the GPS Satellite ( ) objects are placed in the Assigned Objects list.

Updating the GPS constellation's constraints

Constellation constraints allow you to specify the criteria to be used when constellations are combined with other objects in a chain. Each pair of objects in the chain can be thought of as creating access pairs with a "from" object and a "to" object. The constellation constraints allow you to specify different logical and parent ownership constraints depending on where the constellation sits in the chain, either as the "from" object or the "to" object. In this instance, you are "sending" from the GPS satellites to the test team. Therefore, you will use the "From" access position logical restriction.

1. Select the Constraints - Basic page.
2. Open the From access position drop-down list in the Logical Restriction panel.
3. Select At Least N.
4. Enter 4 in the At Least N field.

   Although you aren't analyzing dilution of precision or navigation accuracy, you are testing the link for proper accesses. Therefore, you want to make sure that the test team can access at least four GPS satellites at all times. Anything less than four will not be considered a successful access.

5. Click OK to confirm your changes and to close the Properties Browser.

Cleaning ground tracks and orbits from the 2D and 3D Graphics windows

You will override the default display of satellite ground tracks and orbits. You have multiple satellites propagated into your scenario. By clearing these two check boxes, your 2D and 3D Graphics windows will be easier to use for situational awareness.

1. Bring the 3D Graphics window to the front.
2. Zoom out until you can see all the GPS orbits and ground tracks on the surface of the Earth.
3. Open TerrainChainsContellations ( ) Properties ( ).
4. Select the 2D Graphics - Global Attributes page when the Properties Browser opens.
5. Clear the Show Ground Tracks / Routes check box in the Vehicles panel.
6. Clear the Show Orbits / Trajectories check box.
7. Click **OK** to confirm your changes and to close the Properties Browser.

Inserting the test team's location

Insert a Place object, which will simulate the location of the test team.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Place ( ) object using the Insert Default ( ) method.
3. Right-click on Place1 ( ) in the Object Browser.
4. Select Rename in the shortcut menu.
5. Rename Place1 ( ) TestTeam.

Updating the test team's position

The test team is located in very mountainous terrain. Update TestTeam's position to reflect this.

1. Open TestTeam's ( ) Properties ( ).
2. Select the Basic - Position page when the Properties Browser opens.
3. Set the following options in the Position panel:

Option	Value
Latitude	46.304 deg
Longitude	-122.321 deg
Height Above Ground	6 ft

   Height Above Ground represents the height of Test Team's antenna if you were analyzing communication devices.

4. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Using a terrain mask constraint

Use the terrain mask constraint in your analysis. The STK application constrains access to the object to which access is being calculated by any terrain data in the line of sight. The terrain mask constraint determines instantaneous visibility based on detecting intersections of the instantaneous line of sight with the terrain surface.

1. Select the Constraints - Active page.
2. Click Add new constraints ( ) in the Active Constraints toolbar.
3. Select Terrain Mask in the Constraint Name list when the Select Constraints to Add dialog box opens.
4. Click **Add** .
5. Click **Close** to close the Select Constraints to Add dialog box.
6. Click **OK** to confirm your selection and to close the Properties Browser.

Inserting the base team's location

The base team is located in the city of Morton, Washington.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Place ( ) object using the From City Database ( ) method.
3. Enter Morton in the Name field when the Search Standard Object Data dialog box opens.
4. Click **Search** .
5. Select Morton - Washington in the Results list.
6. Click **Insert** .
7. Click **Close** to close the Search Standard Object Data dialog box.

Raising the base team's height above the ground

Although you aren't using an actual antenna in your analysis, the location of the antenna on the base team's building is located on the building's roof, which is 25 feet above ground level.

1. Open Morton's ( ) Properties ( ).
2. Select the Basic - Position page when the Properties Browser opens.
3. Enter 25 ft in the Height Above Ground field in the Position panel.
4. Click **Apply** to confirm your change and to keep the Properties Browser open.

Using an azimuth-elevation mask in your analysis

Using the azimuth-elevation (AzEl) mask is another way of using analytical terrain in your analysis. The AzElMask properties, which are a part of the Basic properties for facilities, places and targets, enable you to define an AzEl mask for the facility, place, or target. When computing the AzEl Mask from terrain, terrain blockage is only modeled up to the ground distance specified by the maximum range that was considered when generating the mask. If the AzEl Mask constraint is used when doing access to an object, and the ground distance to the object is larger than the maximum range that was considered when computing the mask, then the mask may fail to correctly model the terrain blockage. The AzEl Mask constraint leverages a provided or computed AzEl Mask to determine visibility. The mask may be computed from terrain information to be representative of terrain-based visibility restrictions.

You can construct terrain-based AzEl masks by extending a number of rays in directions of constant azimuth outwards from the facility, place, or target location. Obstruction information is stored along each ray. During visibility computations, the STK software uses obstruction information from the two rays that bound the current direction of interest to compute an interpolated visibility metric.

Defining Morton's AzEl mask

Define an AzEl mask for Morton through its Properties.

1. Select the Basic - AzElMask page.
2. Set the following options:

Option	Value
Use	Terrain Data
Max range to consider	160 km
Use Mask for Access Constraint	Selected

3. Click **Apply** to confirm your changes and to keep the Properties Browser open.

   Using Terrain Data automatically creates and stores an AzEl mask file, which is an ASCII text file that is formatted for compatibility with the STK software and ends in an .aem extension, into your scenario folder. Selecting Use Mask for Access Constraint enables the AzEl Mask constraint located on the Constraints - Active page. Using the AzElMask constraint constrains access to a 360-degree field of view around the object being constrained.

Displaying Morton's AzEl mask

For situational awareness, you can display the AzEl mask in both the 2D Graphics and 3D Graphics windows at a specified number of steps from the minimum to the maximum range from Morton's antenna.

1. Select the 2D Graphics - AzElMask page.
2. Set the following properties in the At Range panel:

Option	Value
Show	Selected
Number of Steps	16
Minimum Range	0 km
Maximum Range	160 km

3. Click **OK** to confirm your changes and to close the Properties Browser.

Decluttering the 3D Graphics window

Although it's not required, you can adjust the properties of the 3D Graphics window to have better situational awareness. Objects located on the surface of the terrain could be covered by the terrain, which makes them unreadable. You can fix this by making a change to the 3D Graphics window's properties.

1. Bring the 3D Graphics window to the front.
2. Click Properties ( ) on the 3D Window Defaults toolbar.
3. Select the Details page when the Properties Browser opens.
4. Select the Enable check box in the Label Declutter panel.
5. Click **OK** to confirm your selection and to close the Properties Browser.

Viewing the AzEl mask in the 3D Graphics window

View the AzEl mask in context in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on Morton ( ) in the Object Browser.
3. Select Zoom To in the shortcut menu.
4. Using your mouse, zoom out until you can see the visual representation of the AzEl mask.

Adding a CubeSat communications satellite

When looking at the AzEl mask in the 3D Graphics window, it's clear that the base team cannot access test team directly through line-of-sight communication. In order for the test team to access the base team, a CubeSat communications satellite, which is located in a low Earth orbit (LEO), is needed.

Inserting a Satellite object

Insert a Satellite object to model the CubeSat satellite.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Satellite ( ) object using the Insert Default ( ) method.
3. Rename Satellite1 ( ) CubeSat.

Propagating the CubeSat satellite

You can set the orbital parameters of the CubeSat satellite by updating the CubeSat's Satellite object's properties.

1. Open CubeSat's ( ) Properties ( ).
2. Select the Basic - Orbit page when the Properties Browser opens.
3. Enter the following orbital parameters:

Option	Value
Semimajor Axis	6997 km
Eccentricity	0.02
Inclination	64.8 deg
Argument of Perigee	267 deg
RAAN	2 deg
True Anomaly	302 deg

4. Click **OK** to propagate CubeSat ( ) and to close the Properties Browser.

Building a Chain object

You are now ready to test access starting from the GPS satellite constellation and ending at Morton. You need a Chain object to test all the links in the access. "Links" in the Chain can be individual objects like satellites, sensors, and places, or grouped objects, like constellations. By defining a Start object, an End object, and sets of paired object connections, the STK application can compute times when one object can access another through connections to one or more other objects.

Inserting the Chain object

Assign objects to the chain and define the order in which objects are accessed.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Chain ( ) object using the Insert Default ( ) method.
3. Rename Chain1 ( ) GPS_to_Morton.

Defining the start and end objects

Start by choosing the start object and end object in your chain.

1. Open GPS_to_Morton's ( ) Properties ( ).
2. Select the Basic - Definition page when the Properties Browser opens.
3. Click the Start Object ellipsis (...).
4. Select GPS_Sats ( ) when the Select Object dialog box opens.
5. Click **OK** to close the Select Object dialog box.
6. Click the End Object ellipsis (...).
7. Select Morton ( ) when the Select Object dialog box opens.
8. Click **OK** to close the Select Object dialog box.

Creating the Chain object's first connection

After you choose the start and end objects in your chain, you need to build the chain's connections. It doesn't matter in which order you place the connections in the Connections list. What matters is the From Object must be able to access the To Object.

1. Click **Add** in the Connections panel.
2. Click the From Object ellipsis (...).
3. Select GPS_Sats ( ) when the Select Object dialog box opens.
4. Click **OK** to close the Select Object dialog box.

5. Click the To Object ellipsis (...).
6. Select TestTeam ( ) when the Select Object dialog box opens.
7. Click OK to close the Select Object dialog box.

Creating the Chain object's second connection

When the test team accesses at least four GPS satellites, they will uplink their fix to the CubeSat satellite.

1. Click Extend in the Connections panel.
2. Click the To Object ellipsis (...).
3. Select CubeSat ( ) when the Select Object dialog box opens.
4. Click OK to close the Select Object dialog box.

Creating the Chain object's final connection

The CubeSat satellite will downlink test team's fix to the base team located in Morton.

1. Click Extend in the Connections panel.
2. Click the To Object ellipsis (...).
3. Select Morton ( ) when the Select Object dialog box opens.
4. Click OK to close the Select Object dialog box.
5. Click OK to confirm your changes and to close the Properties Browser.

Computing Chain accesses

You are ready to compute the Chain object's accesses.

1. Select GPS_to_Morton ( ) in the Object Browser.
2. Select the Chain menu.
3. Select Compute Accesses.

Generating a Complete Chain Access report

Generate a Complete Chain Access to report to analyze whether or not you have accesses during your scenario analysis period. A complete chain access is a time interval during which the chain is completed. Complete chain access intervals are computed by overlapping all the strand accesses.

1. Right-click on GPS_to_Morton ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select the Complete Chain Access ( ) report in the Installed Styles folder ( ) in the Styles panel when the Report & Graph Manager opens.
4. Click **Generate...** .

You could use this report to view the windows of opportunity you'll have to transmit the test team's location to the base team in Morton.

Viewing the accesses in the 3D Graphics window

You can view a complete chain access in the 3D Graphics window directly from the Complete Chain Access report.

1. Right-click on the Start Time of the longest access duration.
2. Select Start Time in the shortcut menu.
3. Select Set Animation Time in the Start Time submenu.
4. Bring the 3D Graphics window to the front.
5. Zoom to Morton ( ).
6. Using your mouse, maneuver your view to get an idea of all the connections in your chain.

You can see the downlink accesses from at least four GPS satellites to the test team. Next, you can see the uplink from the test team to the CubeSat satellite. Finally you can see the downlink from the CubeSat satellite to Morton.

Summary

You conducted a line-of-sight test between a constellation of GPS satellites, a ground-based test team located in mountainous terrain, a CubeSat and a base team located in Morton, Washington. You loaded a USGS DEM file to analyze the impact of local terrain on accesses between the GPS constellation, the test team, the CubeSat and the base team. Using the Terrain Region Converter, you changed the DEM file into a terrain inlay file and used that to visualize the terrain in the 3D Graphics window and to create an azimuth-elevation mask. Next, you created a constellation of GPS satellites, used Place objects to represent the teams' locations, and used a Chain object to create connections between all the objects. Finally, you visualized the complete chain access linking the components together and generated a Complete Chain Access report, which could be used for further mission planning.

Part 7: Customize Analysis with the Analysis Workbench

Note: Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Important: This tutorial requires STK 12.9 or newer to complete in its entirety. If you have an earlier version of STK, you can view a

Note: The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and technicians require additional capabilities when using the STK application to create custom functions and calculations relative to times, positions, and reference frames. You work at a ground station that is tracking a geosynchronous satellite. The ground station's sensor must be monitored for its safety when its boresight is within 30 degrees of the Sun so that it won't be damaged. You require data that informs you when the sensor's boresight is within that 30-degree range and when it's outside of that same 30-degree range.

Solution

Use a combination of STK Pro and the STK software's *Analysis Workbench* capability to create vectors, custom angles, calculations, time components and temporal constraints to determine when the sensor can safely track the satellite.

Creating a new scenario

First, you must create a new STK scenario and then build from there.

1. Launch the STK application ().
2. Click Create a Scenario in the Welcome to STK dialog box.

3. Enter the following in the STK: New Scenario Wizard:

Option	Value
Name	STK_AnalysisWorkbench
Location	Default
Start	Today
Stop	+ 5 days

By using Today as the Start time, every time you open the scenario, it will automatically adjust to the current date, using midnight local time and its equivalent Universal Time Coordinated Gregorian (UTCG) date unless your chose a different time unit for your scenario. This is useful when you want to track Satellite objects that you have inserted into the scenario.

4. Click **OK** when you finish.
5. Click Save ( ) when the scenario loads.

   The STK software creates a folder with the same name as your scenario for you.

6. Verify the scenario name and location in the Save As dialog box.
7. Click **Save** .

Disabling streaming terrain

Streaming terrain is not required for this analysis. Disable the Terrain Server.

1. Right-click on STK_AnalysisWorkbench ( ) in the Object Browser.
2. Select Properties ( ) in the shortcut menu.
3. Select the Basic - Terrain page when the Properties Browser opens.
4. Clear the Use terrain server for analysis check box in the Terrain Server panel.
5. Click **OK** to confirm your change and to close the Properties Browser.

Modeling the ground station

The satellite tracking ground station is located at the Kaena Point Space Force Station on Oahu, Hawaii. Model the station with a Facility object. A Facility object models a ground station or other facility on the surface of the central body.

Inserting the Facility object

Insert a new Facility object from your locally installed Standard Object Database.

1. Bring the Insert STK Objects tool () to the front.
2. Select Facility () in the Select An Object To Be Inserted list.
3. Select From Standard Object Database () in the Select A Method list.
4. Click Insert... .
5. If online operations are enabled, clear the Data Sources: Online check box when the Search Standard Object Data dialog box opens.
6. Enter Kaena In the Name field located under Data Sources.
7. Click Search .
8. Select Kaena Point Radar STDN KPTQ in the Results list.
9. Click Insert .
10. Click Close to close the Search Standard Object Data dialog box.

Adding the facility's lighting intervals to the Timeline View

Lighting intervals are the periods of full lighting (sunlight), partial lighting (penumbra) and zero lighting (umbra). The Timeline View is used to display and operate on time components in your scenario. You can use the Timeline View toolbar to adjust the timeline display and add rows of time components. Add Kaena_Point_ Radar_STDN_KPTQ's lighting intervals to the Timeline View.

1. Click Add Time Components () on the Timeline View toolbar.
2. Select Kaena_Point_Radar_STDN_KPTQ () in the Objects list when the Select Timeline Component dialog box opens.

3. Select LightingIntervals ( ) in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
4. Click **OK** to confirm your selection and to close the Select Timeline Component dialog box.

Viewing Kaena_Point_Radar_STDN_KPTQ in the 3D Graphics window

Use the Timeline View to view Kaena_Point_Radar_STDN_KPTQ in sunlight.

1. Right-click on Kaena_Point_Radar_STDN_KPTQ () in the Object Browser.
2. Select Zoom To in the shortcut menu.
3. Move the Timeline View's gray pointer () to a Sunlight interval so you can better see the facility in the 3D Graphics window.

Note that the facility seems to be floating above the terrain. When you bring a Facility object into your STK scenario using the From Standard Object Database method, the Facility object is automatically placed at the actual altitude of the terrain at that location.

Adjusting the altitude of Kaena_Point_Radar_STDN_KPTQ

Since you're using the WGS84 ellipsoid's surface in your analysis, Kaena_Point_Radar_STDN_KPTQ's altitude needs to be modified. Adjust it so that it sits on the surface of the WGS84 ellipsoid by updating the facility's Position properties.

1. Open Kaena_Point_Radar_STDN_KPTQ's () Properties ().
2. Select the Basic - Position page when the Properties Browser opens.
3. Select the Use terrain data check box in the Position panel.
4. Click **OK** to confirm your selection and to close the Properties Browser.

Kaena_Point_Radar_STDN_KPTQ is now sitting on the surface of the WGS84 ellipsoid.

Inserting a Satellite object

Insert a Satellite object, which will be tracked by Kaena_Point_Radar_STDN_KPTQ. In this scenario you will build a nominal Satellite object using the Orbit Wizard.

Inserting a satellite from the Orbit Wizard

The Orbit Wizard is a satellite-level tool designed to assist you in creating any one of several standard orbits, or designing your own satellite orbit. The configurable options available will depend on the orbit type selected. If you were tracking an actual satellite you would want to propagate the satellite using its TLE or an external ephemeris file with an orbit epoch matching your scenario's analysis period.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Satellite ( ) object using the Orbit Wizard ( ) method.
3. Set the following options when the Orbit Wizard opens:

Option	Value
Type	Geosynchronous
Satellite Name	GEO_Sat
Subsatellite Point	-150 deg

4. Click **OK** to propagate GEO_Sat ( ) and to close the Orbit Wizard.

Updating GEO_Sat's orbit epoch.

It's important to note that should you open this scenario in the future using GEO_Sat, you will need to match the orbit epoch to the day you open the scenario. You can change it by updating GEO_Sat's Basic - Orbit properties.

1. Open GEO_Sat's ( ) Properties ( ).
2. Select the Basic - Orbit page when the Properties Browser opens.
3. Locate the Orbit Epoch field.
4. Open the drop-down menu ( ) next to Orbit Epoch.
5. Select Set to Today in the drop-down menu.
6. Click **OK** to confirm your selection and to close the Properties Browser.

The time at which the established orbital elements are true will now update with your scenario time, should you choose to reopen it.

Computing access

Calculate the times Kaena_Point_Radar_STDN_KPTQ can access, or see, GEO_Sat using the Access tool. This access interval will be used as a component when using the Analysis Workbench.

1. Right-click on Kaena_Point_Radar_STDN_KPTQ ( ) in the Object Browser.
2. Select Access... ( ) in the shortcut menu.
3. Select GEO_Sat ( ) Associated Objects list when the Access tool opens.
4. Click Compute .
5. Look at the Timeline View.

   There is one continuous access interval between Kaena_Point_Radar_STDN_KPTQ and GEO_Sat.
6. Click **Close** to close the Access tool.

Displaying installed components from the Analysis Workbench

The *Analysis Workbench* capability comprises four application-wide tools (the Vector Geometry tool, the Time tool, the Calculation tool, and the Spatial Analysis tool) that you can use to create custom components and insert them into your scenarios. They are designed to streamline, organize, and extend the fundamental computational capabilities of the STK software. You can use these tools to create components that suit your needs.

The Analysis Workbench capability also contributes an extensive list of predefined components to the STK application that you can use immediately. Any of the components created by you or already installed can be used to report, graph, create data files, and visualize 2D and 3D time-dynamic data. You can design your computational tasks by combining different types of components from the Analysis Workbench.

Among the available components are the Facility object's Lighting intervals, which you added earlier, and its Sun Vector. Visualize this vector to aid in the design of your computation.

Displaying Kaena_Point_Radar_STDN_KPTQ's Sun vector

Vectors define directions in 3D space as well as magnitude. The Sun Vector is the apparent position of the Sun with respect to the object as a function of time. In this scenario, the Sun vector is anchored to tracking station's center point and targets the Sun. Enable the display of the vector by updating Kaena_Point_Radar_STDN_KPTQ's 3D Graphics - Vector properties.

1. Open Kaena_Point_Radar_STDN_KPTQ's ( ) Properties ( ).
2. Select the 3D Graphics - Vector page when the Properties Browser opens.
3. Select the Vectors tab.
4. Select the Sun Vector Show check box.
5. Click **Apply** to confirm your selection and keep the Properties Browser open.

Viewing the Sun Vector

View the Sun Vector in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom to Kaena_Point_Radar_STDN_KPTQ ( ).
3. Zoom out far enough so that you can see the Sun vector.

When you animate the scenario, the Sun vector will always follow the Sun.

Adding a To Vector

A To Vector is a displacement vector between origin and destination object points. To Vectors are automatically generated by STK for all objects in your scenario. The vectors are stored in a separate folder labeled "To Vectors," unless a vector with the same name already exists (Earth or Sun). In scenarios with many objects, the To Vectors folder can be very large for each object. Enable the display of the facility's To Vector to GEO_Sat.

1. Return to Kaena_Point_Radar_STDN_KPTQ's ( ) 3D Graphics - Vector Properties ( ).
2. Click **Add...** .

   The Geometry Component Add/Edit windows enable you to specify various properties of a component that you have added or selected for modification.

3. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Add Components dialog box opens.
4. Expand (⊞) the To Vectors ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
5. Select GEO_Sat ( ).
6. Click **OK** to confirm your selection and to close the Add Components dialog box.

Notice that GEO_Sat Vector is now added to the Vectors list, and the Show check box is already selected.

Showing the vector's magnitude

If Show Magnitude is selected, the magnitude (distance) value is displayed on the selected vector. This is only available for vectors. Turn on the magnitude for GEO_Sat's To Vector.

1. Ensure GEO_Sat Vector is highlighted in the Vectors list.
2. Select the Show Magnitude check box below the Vectors list.
3. Click **Apply** to confirm your selection and to keep the Properties Browser open.
4. Bring the 3D Graphics window to the front.

Notice that the GEO_Sat To Vector now shows how far away GEO_Sat is located from Kaena_Point_Radar_STDN_KPTQ. When you animate the scenario, the GEO_Sat To vector will always point to GEO_Sat.

Using the Vector Geometry tool

The *Analysis Workbench* capability's Vector Geometry tool creates geometry components with time-varying placement or orientation in 3D space, including Vectors ( ), Axes ( ), Points ( ), Systems ( ), Angles ( ), and Planes ( ). Use the Vector Geometry tool to create an angle between Sun vector and the GEO_Sat To vector. For the purposes of this scenario, whenever the angle is 30 degrees or less, the satellite tracking system needs to be monitored.

Opening the Analysis Workbench

Access the Vector Geometry tool by opening the Analysis Workbench.

1. Right-click on Kaena_Point_Radar_STDN_KPTQ ( ) in the Object Browser.
2. Select Analysis Workbench... ( ) in the shortcut menu.
3. Select the Vector Geometry tab when the Analysis Workbench opens.

Creating a Between Vectors angle

Create an angle between the Sun Vector and the GEO_Sat Vector using a Between Vectors angle type. A Between Vectors Angle type defines an angle between two vectors.

1. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list on the left side of the browser.
2. Click Create new Angle ( ) in the Vector Geometry toolbar.
3. Ensure the Type is set to Between Vectors when the Add Geometry Component dialog box opens.
4. Enter Tracking Angle in the Name field.

Selecting the From Vector

Select GEO_Sat's To Vector as the angle's From Vector. The custom angle, Tracking Angle, will be measured starting from GEO_Sat's To Vector.

1. Click the From Vector ellipsis (...).
2. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Reference Vector dialog box opens.
3. Expand (⊞) the To Vectors ( ) folder in the Vectors for: Kaena_Point_Radar_STDN_KPTQ list.
4. Select GEO_Sat ( ).
5. Click **OK** to confirm your selection and to close the Select Reference Vector dialog box.

Selecting the To Vector

Use the Sun Vector as the To Vector.

1. Notice that Kaena_Point_Radar_STDN_KPTQ Sun is already selected for the To Vector.
2. Click **OK** to confirm your changes and to close the Add Geometry Component dialog box.
3. Keep the Analysis Workbench ( ) open.

Displaying the tracking angle

Custom angles can be viewed in the 3D Graphics window. Add the custom tracking angle to Kaena_Point_Radar_STDN_KPTQ's 3D Graphics - Vector Properties page.

1. Return to Kaena_Point_Radar_STDN_KPTQ's ( ) 3D Graphics - Vector Properties ( ).
2. Select the Angles tab.
3. Click **Add...** .

4. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Add Components dialog box opens.

5. Select TrackingAngle ( ), located in the My Components ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.

6. Click OK to confirm you selection and to close the Add Components dialog box.

7. Click OK to confirm your changes and to close the Properties Browser.

Viewing the tracking angle

View your custom tracking angle in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.

2. Adjust your view so that you can see the Sun Vector, the To Vector and the Tracking Angle.

3. Move the Timeline View's gray pointer ( ) to animate the scenario.

Notice that as the scenario animates, the angle increases and decreases, updating dynamically.

Using the Calculation tool

Use the Calculation tool to determine when the tracking angle is 30 degrees or less. The Calculation tool defines components that produce time-varying computational results that can be reported, graphed, transformed, and analyzed further.

Opening the Calculation tool

Open the Calculation tool from the Analysis Workbench.

1. Return to Analysis Workbench.

2. Select the Calculation tab to open the Calculation tool.

Defining a Scalar calculation

You will create a Scalar calculation to return the tracking angle values. A Scalar defines components that produce scalar time-varying calculations. Scalar calculation components also have the ability to return minimum, maximum, mean, and standard deviation values.

1. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list.
2. Click Create new Scalar Calculation ( ) in the Calculation toolbar.
3. Ensure the Type is set to Angle when the Add Calculation Component dialog box opens.

   The default scalar calculation type is Angle. The Angle type is an angular displacement specified by an Angle component from the Vector Geometry tool.
4. Enter Scalar Tracking Angle in the Name field.
5. Click the Input Angle ellipsis (...).
6. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Reference Angle dialog box opens.
7. Select TrackingAngle ( ), located in the My Components ( ) folder in the Angles for: Kaena_Point_Radar_STDN_KPTQ list.
8. Click OK to confirm your selection and to close the Select Reference Vector dialog box.
9. Click OK to confirm your changes and to close the Add Calculation Component dialog box.

Defining a Condition

A condition defines a scalar calculation that is considered to be satisfied when it is positive and not satisfied when it is negative. Create a Scalar Bounds Condition to report when the tracking angle is 30 degrees or less. Scalar Bounds define a condition by combining a specified Scalar component with scalar bounds.

1. Return to the Calculation tool.
2. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list.
3. Click Create new Condition ( ) in the Calculation toolbar.
4. Ensure the Type is set to Scalar Bounds when the Add Calculation Component dialog box opens.

   The default Calculation type is Scalar Bounds.
5. Enter Below 30 Degrees in the Name field.

Selecting the Reference Scalar Calculation

Select the tracking angle scalar component you created earlier as the Reference Scalar Calculation.

1. Click the Scalar ellipsis (...).
2. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Reference Scalar Calculation dialog box opens.
3. Select Scalar_Tracking_Angle ( ), located in the My Components ( ) folder in the Scalar Calculations for: Kaena_Point_Radar_STDN_KPTQ list.
4. Click OK to confirm your selection and to close the Select Reference Scalar Calculation dialog box.

Defining the Scalar Bounds' operation

Define the calculation condition of the Scalar Bounds to be 30 degrees or less by choosing an associated operation.

1. Open the Operation drop-down list.
2. Select Below Maximum.
3. Enter 30 deg in the Maximum field.

   When the values are at 30 degrees or below, the calculation will be satisfied.
4. Click OK to confirm your changes and to close the Add Calculation Component dialog box.

Creating a report

You can create reports and graphs directly inside the Calculation tool. Create an Access Intervals by Time report showing the satisfaction intervals when the tracking angle is below 30 degrees.

1. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list.
2. Expand (⊞) Below_30_Degrees ( ), located in the My Components ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
3. Right-click on SatisfactionIntervals ( ).
4. Select Report... in the shortcut menu.

   The report shows when the tracking angle is 30 degrees or less during the entire scenario interval.
5. Close ( ) the report.
6. Keep the Analysis Workbench open.

Viewing the access intervals in the Timeline View

Use the Timeline View to visualize the times Kaena_Point_Radar_STDN_KPTQ has access to GEO_Sat and the times the tracking angle is less than 30 degrees.

1. Click Add Time Components ( ) in the Timeline View toolbar.
2. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Timeline Component dialog box opens.
3. Expand (⊞) Below_30_Degrees ( ), located in the My Components ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
4. Select SatisfactionIntervals ( ).
5. Click **OK** to confirm your selection and to close the Select Timeline Component dialog box.
6. Look in the Timeline View to view the intervals when the tracking angle at 30 degrees or below.

Using the Time tool

Time is fundamental to most computations in STK and is used in reporting and graphing as well as in static and dynamic visualizations. Use the Time tool to create components that deal with time-related quantities.

Opening the Time tool

Open the Time tool from the Analysis Workbench.

1. Return to the Analysis Workbench.
2. Select the Time tab.

Creating an Interval List

An Interval List Time component type defines components that produce an ordered list of time intervals. Create an Interval List to show when Kaena_Point_Radar_STDN_KPTQ has access to GEO_Sat while the tracking angle is above 30 degrees by merging the access intervals with the Below_30_Degrees satisfaction intervals. Subtract the Below_30_Degrees satisfaction intervals from the access intervals to define optimal tracking opportunities.

1. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list.
2. Click Create new Interval List ( ) in the Time toolbar.
3. Click Type **Select...** when the Add Time Component dialog box opens.
4. Select Merged ( ) in the Select Component Type list when the Select Component Type dialog box opens.

   A Merged Interval list contains intervals merged from multiple Interval or Interval List time components.
5. Click **OK** to confirm your selection and to close the Select Component Type dialog box.
6. Enter Optimal Tracking Times in the Name field.

Defining the merging operation

Define the merge operation as Minus. The Minus operation is available when there are only two time components in the merge list. This will subtract the below 30-degree satisfaction intervals from the accesses to define optimal tracking opportunities.

1. Open the Operation drop-down list.
2. Select MINUS.

Removing the default time components

Remove the default time components.

1. Select both Time Components in the Time Components list.
2. Click **Remove** .

Defining the Time Components

In this instance, there are two time components you need for your merged interval list: the accesses between the Kaena_Point_Radar_STDN_KPTQ and GEO_Sat and the Below_30_Degrees satisfaction intervals.

1. Click **Add...** .
2. Select Facility-Kaena_Point_Radar_STDN_KPTQ-To-Satellite-GEO_Sat ( ) in the Objects list when the Select Time Intervals dialog box opens.

3. Select AccessIntervals ( ), located in the Installed Components ( ) folder in the Components for: Facility-Kaena_Point_Radar_STDN_KPTQ-To-Satellite-GEO_Sat list.
4. Click **OK** to confirm your selection and to close the Select Time Intervals dialog box.
5. Click **Add...** .
6. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Time Intervals dialog box opens.
7. Expand (⊞) Below_30_Degrees ( ), located in the My Components ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
8. Select SatisfactionIntervals ( ).
9. Click **OK** to confirm your selection and to close the Select Time Intervals dialog box.
10. Click **OK** to confirm your changes and close the Add Time Component dialog box.

Viewing the Optimal Tracking Times Interval List in the Timeline View

You now have all the components for your analysis completed. View them in the Timeline View.

1. Click Add Time Components ( ) in the Timeline View toolbar.
2. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Timeline Component dialog box opens.
3. Select Optimal_Tracking_Times ( ), located in the My Components ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
4. Click **OK** to confirm your selection and to close the Select Timeline Component dialog box.
5. Look at the Timeline View to see the all the time components.
6. Return to the Analysis Workbench.
7. Click **Close** to close the Analysis Workbench.

Modeling a sensor outage using components from the Analysis Workbench

You can use the custom components your created using the Analysis Workbench with other objects in the scenario for a variety of applications. Use your optimal tracking times Interval List to model a potential outage on a Sensor object attached to when the tracking angle goes below 30 degrees.

Attaching a Sensor object to Kaena_Point_Radar_STDN_KPTQ

Attach a Sensor object to Kaena_Point_Radar_STDN_KPTQ.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Sensor ( ) object using the Define Properties ( ) method.
3. Select Kaena_Point_Radar_STDN_KPTQ ( ) when the Select Object dialog box opens.
4. Click **OK** to confirm your selection and to close the Select Object dialog box.

Modeling a limited field of view

Set a limited field of view for the Sensor object to provide situational awareness. The Definition page enables you to set parameters defining a sensor's field of view.

1. Select the Basic - Definition page when the Properties Browser opens.
2. Keep the default Simple Conic Sensor Type.
3. Enter 5 deg in the Cone Half Angle field.
4. Click **Apply** to confirm your change and to keep the Properties Browser open.

Targeting GEO_Sat

Use the Targeted pointing type to point the Sensor object at GEO_Sat.

1. Select the Basic - Pointing page.
2. Open the Pointing Type drop-down list.
3. Select Targeted.
4. Select GEO_Sat ( ) in the Available Targets list.
5. Move ( ) GEO_Sat ( ) to the Assigned Objects list.
6. Click **OK** to confirm your changes and to close the Properties Browser.
7. Rename Sensor1 ( ) Tracking_Sensor.

Computing access

Compute an access between the Tracking_Sensor and GEO_Sat.

1. Right-click on Tracking_Sensor ( ) in the Object Browser.
2. Select Access... ( ) in the shortcut menu.
3. Select GEO_Sat ( ) in the Associated Objects list when the Access tool opens.
4. Click Compute .

Generating an Access report

You can generate several types of reports and graphs that enable you to view specific types of access data directly from the Access tool. Generate an Access report.

1. Click **Access...** in the Reports panel.

   Notice that there is one continuous access between Tracking_Sensor and GEO_Sat.
2. Return to the Access tool.
3. Click **Close** to close the Access tool.
4. Keep the Access report open.

Adding a Temporal constraint

You want to know when Tracking_Sensor accesses GEO_Sat, but you want to remove access times from your report when Tracking_Sensor's boresight is within 30 degrees of the Sun. You can do this by adding a Temporal constraint to Tracking_Sensor using the custom components created in your scenario. Temporal constraints enable you to impose time-based constraints on an object.

1. Open Tracking_Sensor's ( ) Properties ( ).
2. Select the Constraints - Active page when the Properties Browser opens.
3. Click Add new constraints ( ) in the Active Constraints toolbar.
4. Clear the All Categories check box in the Filter by Category list when the Select Constraints to Add dialog box opens.
5. Select the Temporal check box.

6. Select Intervals in the Constraint Name list.
7. Click **Add** .
8. Click **Close** to close the Select Constraints to Add dialog box.

Setting the Constraint Properties

Set the constraint to use the merged interval list you created earlier.

1. Open the Source drop-down list in the Constraint Properties section.
2. Choose Select time component.
3. Click **Select...** .
4. Select Kaena_Point_Radar_STDN_KPTQ ( ) in the Objects list when the Select Interval, Interval List or Interval Collection dialog box opens.
5. Select Optimal_Tracking_Times ( ), located in the My Components ( ) folder in the Components for: Kaena_Point_Radar_STDN_KPTQ list.
6. Click **OK** to confirm your selection and to close the Select Interval, Interval List or Interval Collection dialog box.
7. Notice that the Exclude Time Intervals check box is cleared.

   The Access tool will only report accesses during the optimal tracking times. If you select the Exclude Time Intervals check box, then the Access report will show when the tracking angle is below 30 degrees.
8. Click **OK** to confirm your changes and to close the Properties Browser.

Refreshing the Access report

The current Access report has one long access period for five days. Refresh the report so that it shows only those times that the tracking angle is above 30 degrees. These will be your optimal tracking times.

1. Return to the Access report.
2. Click Refresh (F5) ( ) in the Access report toolbar.

   You should only see those time that the tracking angle is above 30 degrees. If you look at the Timeline View, you'll see that the access between Tracking_Sensor and GEO_Sat matches the Optimal Tracking Times time component.
3. Close the Access report.

Viewing Tracking_Sensor in the 3D Graphics window

You can watch Tracking_Sensor turn on and off in the 3D Graphics window using the Timeline View.

1. Bring the 3D Graphics window to the front.
2. Zoom to Kaena_Point_Radar_STDN_KPTQ ( ).
3. Use your mouse to zoom out far enough so that you can see Tracking_Sensor ( ).
4. Move the Timeline View's gray pointer ( ) over a couple of days while watching Tracking_Sensor ( ) in the 3D Graphics window.

Tracking_Sensor will turn on during optimal tracking times and turn off during periods when the tracking angle is below 30 degrees.

Part 8: Compute Coverage Over Regions

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators need to analyze global or regional coverage from one asset or from a collection of assets. They might need to account for constraints such as sunlight and terrain. They may need to simply analyze if a sensor footprint passes over a specific ground location, or they may need to analyze a large area of ground or space and determine communication bit error rates in terms of area of interest, age of data, dilution of precision, etc. In this lesson, you will analyze three satellites and their sensor footprints on the Earth's surface. You need to determine what percentage of the Earth's surface is seen by all three sensors during a 24-hour period while taking into consideration sunlight, umbra, and how many times points on the ground are accessed. Then, you need to determine how long one satellite sensor covers Canada and the continental United States during the same 24-hour period.

Solution

Use STK to model Earth-observing payloads attached to sensors located in three different orbits. Use STK's ***Coverage*** capability to model and analyze the quality and quantity of coverage provided by the three payloads to determine the following:

- The percentage of the Earth's surface the satellite payloads survey during a 24-hour period and daylight hours only
- How many times the satellite payloads survey points on the ground during a 24-hour period and daylight hours only
- How long points on the ground are seen in Canada and the continental United States

---

Create a new scenario

Create a new scenario.

1. Launch STK ( ).

2. Click Create a Scenario ( ) in the Welcome to STK window.

3. Enter the following in the STK: New Scenario Wizard:

Option	Value
Name	STK_Coverage
Location	Default
Start	15 Mar 2024 16:00:00.000 UTCG
Stop	16 Mar 2024 16:00:00.000 UTCG

4. Click **OK** when you finish.

5. Click Save ( ) when the scenario loads. A folder with the same name as your scenario is created for you in the location specified above.

6. Verify the scenario name and location in the Save As dialog box.

7. Click **Save** .

Viewing both the 2D and 3D Graphics windows

You can view coverage in both the 2D and 3D Graphics windows. Placing them side by side makes this simple to do.

1. Close ( ) the Timeline View at the bottom of STK.

2. Extend the Window menu.

3. Select Tile Vertically to evenly space the windows side-by-side in the integrated workspace.

Turning off Terrain Server

Analytical and visual terrain is not required in this analysis. Turn off the Terrain Server.

1. Right-click STK_Coverage ( ) in the Object Browser.

2. Select Properties ( ).

3. Select the Basic - Terrain page when the Properties Browser opens.
4. Clear the Use terrain server for analysis check box.
5. Click **OK** to accept your changes and to close the Properties Browser.

Creating a satellite in a circular orbit

Insert the first Satellite ( ) and place it in a circular orbit. Circular orbits have a constant radius.

1. Select Satellite ( ) in the Insert STK Objects tool.
2. Select the Orbit Wizard ( ) method.
3. Click **Insert...** .
4. Set the following in the Orbit Wizard:

Option	Value
Type	Circular
Satellite Name	Circ_Sat
Inclination	55 deg
Altitude	700 km
RAAN:	-105 deg

5. Click **OK** to propagate the Satellite ( ) object using the default J4Perturbation propagator and to close the Orbit Wizard.

Creating a satellite in a repeating ground trace orbit

Insert the second Satellite ( ) and place it in a repeating ground trace orbit. Orbits with repeating ground traces are useful when you want identical viewing conditions at different times to detect changes. You can have the ground trace repeat every day or interweave from day to day before repeating.

1. Insert a Satellite ( ) object using the Orbit Wizard ( ) method.
2. Set the following in the Orbit Wizard:

Option	Value
Type	Repeating Ground Trace
Satellite Name	Repeat_Sat

3. Click OK .

Creating a satellite in a sun-synchronous orbit

Insert the third Satellite ( ) and place it in a sun-synchronous orbit. These orbits are designed to utilize the effect of the Earth's oblateness, causing the orbit plane to precess at a rate equal to the mean orbital rate of the Earth around the Sun. Sun synchronous orbits have the property that their nodes maintain constant local mean solar times.

1. Insert a Satellite ( ) object using the Orbit Wizard ( ) method.
2. Set the following in the Orbit Wizard:

Option	Value
Type	Sun Synchronous
Satellite Name	Sun_Sat

3. Click OK .

Adding payloads

All three satellites use the same sensor type and size. STK enables you to copy and paste objects from one object to another. You will set up the first sensor and then copy and paste it to the other two. All you'll have to do is rename them or STK will use the same name with a one-up number.

Inserting a Sensor object

Attach a Sensor ( ) object to Circ_Sat ( ). Define a rectangular sensor with a 20-degree vertical half-angle and 10-degree horizontal half-angle.

1. Insert a Sensor ( ) object using the Define Properties ( ) method.
2. Select Circ_Sat ( ) in the Select Object dialog box.
3. Click OK .
4. Select the Basic - Definition page when the Properties Browser opens.
5. Set the following:

Option	Value
Sensor Type	Rectangular
Vertical Half Angle	20 deg
Horizontal Half Angle	10 deg

6. Click OK to accept your changes and to close the Properties Browser.

Reusing the Sensor object

Copy and paste the rectangular sensor onto the other two satellites.

1. Select Sensor1 ( ) in the Object Browser .
2. Click Copy ( ) in the Object Browser toolbar.
3. Select Repeat_Sat ( ) in the Object Browser.
4. Click Paste ( ) in the Object Browser toolbar.
5. Select Sun_Sat ( ) in the Object Browser.
6. Click Paste ( ) in the Object Browser toolbar.

Renaming the Sensor Objects

Rename the three sensors.

1. Right-click on Sensor1 ( ) in the Object Browser.
2. Select Rename in the shortcut menu.
3. Rename Sensor1 ( ) to Circ_Sens.

4. Press Enter on your keyboard.
5. Repeat the steps above to rename Sensor2 () to Repeat_Sens and Sensor3 () to Sun_Sens.

Adding a Coverage Definition object

The Coverage Definition object defines a coverage area for analysis.

Inserting a Coverage Definition object

Insert a Coverage Definition () object into your scenario.

1. Insert a Coverage Definition () object using the Insert Default () method.
2. Rename CoverageDefinition1 () to World_Cov.

Defining the coverage grid

Define the coverage grid to be global, with a 4-degree point granularity.

1. Open World_Cov's () Properties ().
2. Select the Basic - Grid page when the Properties Browser opens.
3. Open the Type: shortcut menu in the Grid Area of Interest panel.
4. Select Global.
5. Enter 4 deg in the Lat/Lon field located in the Point Granularity panel.
6. Click Apply to accept your changes and to keep the Properties Browser open.
7. Select the 3D Graphics window to bring it to the front. Notice the grid covers the entire globe and the grid spacing is 4 degrees.

Assigning coverage assets

Assets properties allow you to specify the STK objects used to provide coverage. Define the three sensors as the Coverage Assets.

1. Return to World_Cov's () properties ().
2. Select the Basic - Assets page.
3. Expand () Circ_Sat (), Repeat_Sat () and Sun_Sat () in the Assets list.
4. Select Circ_Sens (), Repeat_Sens () and Sun_Sens ().
5. Click **Assign** .
6. Click **Apply** to accept your changes and to keep the Properties Browser open.

Turning off automatic recomputation of accesses

STK automatically recomputes accesses every time you update an object on which the coverage definition depends (such as an asset). If you want control as to when STK computes coverage, you need to turn this off.

1. Select the Basic - Advanced page.
2. Clear the Automatically Recompute Accesses check box.
3. Click **OK** to accept your changes and to close the Properties Browser.

Using the Compute Accesses tool

The ultimate goal of coverage is to analyze accesses to an area by using assigned assets and applying necessary limitations upon those accesses. Compute coverage with the Compute Accesses tool.

1. Select World_Cov () in the Object Browser.
2. Extend the CoverageDefinition menu.
3. Select Compute Accesses.

   You can view the progress bar in the lower-right corner of STK.

Generating a coverage by latitude graph

A coverage by latitude graph uses the Coverage by Latitude data provider. This analyzes coverage for each latitude in the selected range at intervals depending on the selected resolution. It uses the Latitude and Percent Time Covered elements.

1. Right-click on World_Cov ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select the Coverage By Latitude graph ( ) in the Installed Styles list.
4. Click **Generate...** .

   The Coverage By Latitude graph is a quick way to see the percentage of time covered by latitude.
5. Close ( ) the graph.
6. Click **Close** to close the Report & Graph Manager.

Figure of Merit

STK enables you to specify the method by which the quality of coverage is measured using a Figure Of Merit ( ) object.

Inserting a Figure of Merit

Insert a Figure of Merit object.

1. Insert a Figure Of Merit ( ) object using the Insert Default ( ) method.
2. Select World_Cov ( ) in the Select Object dialog box.
3. Click **OK** .
4. Rename FigureOfMerit1 ( ) to Simple_Cov.

Measuring Simple Coverage

Simple Coverage measures whether or not a point is accessible by any of the assigned assets during the analysis period. Areas on the map that are shaded mean that the surface point is seen by at least one sensor footprint during the 24-hour analysis period.

1. Open Simple_Cov's ( ) Properties ( ).
2. Select the Basic - Definition page when the Properties Browser opens.
3. Look at the Definition - Type. It defaults to Simple Coverage.

4. Click Cancel to close the Properties Browser without making any changes.
5. Bring the 2D Graphics window to the front.

   Your 2D Graphics window may display different colors than the image above. You can change the color in Simple_Cov's ( ) 2D Graphics - Static properties page.

Reporting Percent Satisfied

Generate a Percent Satisfied report. This presents the percentage of the total grid area where the static value of the Figure Of Merit meets the specified satisfaction criterion. In this scenario, the report shows the percentage of the globe that is accessible by any of the three sensors during the analysis period.

1. Right-click Simple_Cov ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select the Percent Satisfied report ( ) in the Installed Styles list.
4. Click Generate... .
5. Note the % Satisfied value at the bottom of the report (e.g., ~77%).

   Approximately 77% of the globe is covered by at least one of the three sensors during the analysis time period.
6. Close the report and the Report & Graph Manager when finished.

Creating a constraint

Once you have defined the grid area, you can specify an object class or a specific object for the points within the grid. You are interested in the percentage of coverage during periods of direct sun.

Creating a Target object

Insert a Target ( ) object and define a direct sun lighting constraint.

1. Insert a Target ( ) object using the Define Properties ( ) method.
2. Select the Constraints - Active page when the Properties Browser opens.

3. Click Add new constraints ( ) in the Active Constraints toolbar.
4. Type Lighting in the Search: field when the Select Constraints to Add dialog box opens.
5. Select Lighting in the Constraint Name list.
6. Click **Add** .
7. Click **Close** to close the Select Constraints to Add dialog box.
8. Look at the Lighting Constraint Name. DirectSun is the default Value.
9. Click **OK** to accept your changes and to close the Properties Browser.
10. Rename Target1 ( ) to Constraint_Template.
11. Clear the check box next to Constraint_Template ( ) in the Object Browser. There is no need to see Constraint_Template ( ) on the 2D or 3D Graphics windows.

Applying the grid constraint

Set the Constraint_Template ( ) as the grid constraint.

1. Open World_Cov's ( ) Properties ( ).
2. Select the Basic - Grid page when the Properties Browser opens.
3. Click **Grid Constraint Options...** in the Grid Definition panel.
4. You can see that the Reference Constraint Class defaults to Target in the Grid Constraint Options dialog box.

   For all object classes, the Basic properties of the object, excluding positional information, are applied to the grid points.
5. Select the Use Object Instance check box.
6. Select Constraint_Template in the list.
7. Click **OK** to close the Grid Constraints Options dialog box.
8. Click **OK** to accept your changes and to close the Properties Browser.

Recomputing accesses

Since a change was made to the coverage grid, you need to recompute accesses.

1. Select World_Cov () in the Object Browser.
2. Extend the CoverageDefinition menu.
3. Select Compute Accesses.
4. Bring the 2D Graphics window to the front when finished.

Determining Coverage Loss

It's obvious from the view in the 2D Graphics window that you have less coverage using a Direct Sun constraint. Due to the analysis time period, there is more sunlight in the northern hemisphere that you can see on the map. Generate a Percent Satisfied report to determine the percent coverage loss.

1. Right-click Simple_Cov () in the Object Browser.
2. Select Report & Graph Manager... () in the shortcut menu.
3. Select the Percent Satisfied report () in My Favorites
4. Click **Generate...** .
5. Note the % Satisfied value at the bottom of the report (e.g., ~51 %).

   Since coverage is now being analyzed during periods of direct sun and not being analyzed during periods of umbra, there's a significant loss of coverage.
6. Close report and the Report & Graph Manager when finished.

Measuring Number of Accesses

Insert a new Figure of Merit to measure Number of Accesses. This will measure the number of independent accesses of points.

1. Clear the check box next to Simple_Cov () in the Object Browser.
2. Insert a Figure Of Merit () object using the Define Properties () method.
3. Select World_Cov () in the Select Object dialog box.
4. Click **OK** .
5. Select the Basic - Definition page when the Properties Browser opens.

6. Open the Type: shortcut menu in the Definition panel.
7. Select Number Of Accesses.
8. Keep Compute set to the default Total.

   Total computes the total number of accesses over the coverage interval without considering overlapping accesses.
9. Click OK to accept your changes and to close the Properties Browser.
10. Rename FigureOfMerit2 ( ) to Num_Access.

Generating a Grid Stats report

Generate a Grid Stats report to see the smallest to largest number of accesses to any point in the grid.

1. Right-click Num_Access ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select the Grid Stats report ( ) in the Installed Styles list.
4. Click Generate... .
5. Note the Maximum value (e.g., 6).

   That means at least one point in the grid was accessed on six (6) different occasions during the analysis period.
6. Note the Minimum value (e.g., 0).

   That means at least one point in the grid was never accessed by any of the three sensors during the analysis period.
7. Close the report and the Report & Graph Manager when finished.
8. Clear the check box next to World_Cov ( ) in the Object Browser.

Creating an Area Target object for coverage

You can focus the coverage grid in a confined location using Area Target ( ) objects. For this part of the analysis, you will focus on the continental United States and mainland Canada. You will ignore islands and territories.

Insert an Area Target ( ) to model Canada and the Continental United States.

1. Insert an Area Target ( ) object using the Select Countries and US States ( ) method.
2. Clear the US States check box in the List Selections panel when the Select Countries And US States dialog box opens.
3. Use the Ctrl key to select Canada and United_States_of_America in the countries list on the left. You can insert them individually if desired.
4. Click **Insert** .
5. Click **Close** to close the Select Countries And US States dialog box.

Defining a new Coverage Definition

You want to determine coverage time inside the boundaries of the Area Target ( ) objects. You will use Repeat_Sens ( ) as the asset. No constraints are necessary.

Inserting a Coverage Definition

Insert a new Coverage Definition ( ).

1. Insert a Coverage Definition ( ) object using the Insert Default ( ) method.
2. Rename CoverageDefinition2 ( ) to Country_Cov.

Define Coverage Grid

Define the coverage area of interest to be Canada and the continental United States using the Area Targets ( ).

1. Open Country_Cov's ( ) properties ( ).
2. Select the Basic - Grid page when the Properties Browser opens.
3. Open the Type: shortcut menu in the Grid Area of Interest panel.
4. Select Custom Regions.
5. Open the next shortcut menu below Type:.

6. Select Area Targets.
7. Use the Ctrl key to select Canda and United_States_of_America in the Area Targets list on the left.
8. Move ( ) Canada ( ) and United_States_of_America ( ) from the Area Targets list to the Selected Regions list.
9. Enter 1 deg in the Lat/Lon field located in the Point Granularity field.
10. Click Apply to accept your changes and to keep the Properties Browser open.
11. Bring the 2D Graphics window to the front.
12. Zoom In ( ) so that you only see Canada and United States.

    Your 2D Graphics window may display different colors than the image above. The grid point colors can be changed by changing Country_Cov's ( ) color in the Object Browser or on Country_Cov's ( ) 2D Graphics - Attribute properties page.
13. If desired, you can Zoom In ( ) and center the 3D Graphics window to the same view as the 2D Graphics window.

Assigning a coverage asset

Define Repeat_Sens ( ) as the Coverage Asset.

1. Return to Country_Cov's ( ) properties ( ).
2. Select the Basic - Assets page.
3. Expand (⊞) Repeat_Sat ( ) in the Assets list.
4. Select Repeat_Sens ( ) in the Assets list.
5. Click Assign .
6. Click Apply to accept your changes and to keep the Properties Browser open.

Turning off grid display

You will display Figure Of Merit graphics, so turn off the grid point display.

1. Select the 2D Graphics - Attributes page.
2. Clear the Show Points check box in the Grid panel.

This turns off the visual grid inside the Area Target ( ) object. Analytically, they're still there.

3. Click **OK** to accept your changes and to close the Properties Browser.

Computing accesses

You can compute accesses from the Object Browser vice the CoverageDefinition menu.

1. Right-click Country_Cov ( ) in the Object Browser.
2. Select CoverageDefinition in the shortcut menu.
3. Select Compute Accesses in the next shortcut menu.

Defining a Figure Of Merit object

Use a Figure Of Merit ( ) object to determine the amount of time during which the grid points are covered by Repeat_Sens ( ).

Inserting a Figure of Merit

Insert a Figure Of Merit ( ) to measure the quality of coverage of Country_Cov ( ).

1. Insert a Figure Of Merit ( ) object using the Insert Default ( ) method.
2. Select Country_Cov ( ) in the Select Object dialog box.
3. Click **OK** .
4. Rename the FigureOfMerit3 ( ) to Cov_Time.

Measuring Coverage Time

Set the Figure Of Merit to Coverage Time. It will measure the amount of time during which grid points are covered.

1. Open Cov_Time's ( ) properties ( ).
2. Select the Basic - Definition page when the Properties Browser opens.
3. Open the Type: shortcut menu in the Definition panel.
4. Select Coverage Time.
5. Open the Compute: shortcut menu.
6. Select Total.
7. Click Apply to accept your changes and to keep the Properties Browser open.

Generating a Grid Stats report

Generate a report to determine the minimum and maximum amount of time any of the grid points are covered by Repeat_Sens ( ).

1. Right-click Cov_Time ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select Grid Stats report ( ) in the My Favorites list.
4. Click Generate... .
5. Note the Maximum (sec) (e.g., ~49 seconds).

   That means at least one point in the grid was accessed for a total of 49 seconds during the analysis period.
6. Return to Cov_Time's ( ) properties ( ).

Defining static graphics for the Figure Of Merit

Define static graphics for the Figure Of Merit on the 2D Graphics - Static page.

1. Select the 2D Graphics - Static page.
2. Enter 30 in the % Translucency: field in the Show Points As panel.
3. Select the Show Contours option in the Display Metric panel.
4. Click Remove All in the Level Attributes panel.

5. Set the following in the Level Adding panel:

Option	Value
Start	0 sec
Stop	round down to the nearest integer from the Maximum (sec) value in the Grid Stats report (e.g. 49 sec)
Step	5 sec

6. Click **Add Levels**.
7. Open the Start Color: shortcut menu in the Level Attributes panel.
8. Select red.
9. Open the End Color: shortcut menu
10. Select blue.

    Points with no coverage will be red and any points at or above your highest Level Attribute value will be blue.
11. Select the Natural Neighbor option in the Contour Interpolation (points must be filled) panel.

    Color is applied smoothly over all points in the grid to differentiate contour levels.
12. Click **Apply** to accept your changes and to keep the Properties Browser open.

Set 2D and 3D Graphics windows legends

Once you have set the contours for coverage, you can set the display of the contour key, or legend.

1. Click **Legend...** in the Level Attributes panel.
2. Click **OK** to close Cov_Time's ( ) properties ( ).
3. Click **Layout...** in the Static Legend for Cov_Time dialog box.
4. Set the following on the Figure of Merit Legend Layout dialog box:

Option	Value
2D Graphics Window - Show at Pixel Location	on
3D Graphics Window - Show at Pixel Location	on

Text Options - Title:	Coverage Time (seconds)
Text Options - Number Of Decimal Digits:	0
Range Color Options - Color Square Width (pixels):	50

5. Click OK to close the Figure of Merit Legend Layout dialog box.
6. Close () the Static Legend for Cov_Time dialog box.
7. Bring the 2D and 3D Graphics windows to the front.
8. Note Snap Frame () in both the 2D and 3D Graphics windows. You could use Snap Frame to take a picture of the map to place in a PowerPoint slide or in a document.

Summary

You placed three Satellite () objects into the scenario using circular, repeating ground trace and sun-synchronous orbits. All three satellites had similar Sensor () objects. You built a Coverage Definition () object and set the grid definition to global. Using all three Sensor () objects, you computed access. Using a Figure Of Merit () object, you learned about Simple Coverage and how to analyze the percentage of the Earth accessed during a twenty four hour period. Next, you loaded a Target () object at the Prime Meridian (default location), and set a direct sun constraint. Returning to the Coverage Definition () object, you applied the constraint across the coverage grid and recalculated coverage to see how the direct sun constraint affected the percentage of coverage. The next step was to add a second Figure Of Merit () object and determine the number of accesses against each point in the coverage grid. Switching gears, you loaded two Area Target () objects which outlined the continental United States and Canada. With a new Coverage Definition () object, you focused coverage inside of the Area Target () objects using only the Sensor () object attached to the satellite in the repeating ground trace orbit. This time, you configured a new Figure Of Merit () object to focus on how long each point in the coverage grid was covered by the Sensor () object. Finally, you learned how to apply static contours to both the 2D and 3D Graphics windows.

Part 9: Introduction to the AzEl Mask Tool and Volumetrics

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem

Engineers and operators require a quick way to determine if the Earth, local terrain, nature, and man-made structures affect visibility between ground sites and satellites for a variety of purposes, such as communications, imaging, radar, and general situational awareness. A country's space program is planning to install a satellite tracking radar in an area that has distant hills and a large mountain range. A communication antenna, enclosed by a large radome, will be constructed close by. You want to determine how much impact the Earth, terrain and the radome will have on the radar's field of view.

Solution

Using the STK software, insert Facility objects to simulate the radar and communications sites. Use a local terrain file for analysis and determine access times between the radar's field of view and several Earth-observing satellites. Use the AzEl Mask tool to determine if the communications site's radome further degrades the radar's access times to the satellites. Insert an Area Target object to outline the approximate maximum distance that a satellite can be observed by the radar site. Use the *Analysis Workbench* capability's Spatial Analysis tool and the *Volumetric Analysis* capability to build and analyze a Volumetric object to determine how much of the radar's field of view is blocked by the Earth, terrain, and the radome at a specified distances and altitudes.

Creating a new scenario

Create a new scenario, then build from there.

1. Launch the STK application ().
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following in the STK: New Scenario Wizard:

Option	Value
Name	AzElMask_Volumetrics
Location	Default
Start	15 Mar 2024 12:00:00.000 UTCG
Stop	+ 1 day

4. Click OK when you finish.
5. Click Save () when the scenario loads. The STK application creates a folder with the same name as your scenario in the location specified above.
6. Verify the scenario name and location.
7. Click Save .

Turning off streaming terrain

Since you will use a local analytical terrain file in this analysis, disable Terrain Server.

1. Right-click on AzElMask_Volumetrics's () in the Object Browser.
2. Select Properties ().
3. Select the Basic - Terrain page.
4. Clear the Use terrain server for analysis check box in the Terrain Server panel.
5. Click OK to accept the change and close the Properties Browser.

Adding analytical and visual terrain

Load local terrain data, in the form of an STK Terrain File (*.pdtt) located in the STK software install directory, for analysis and situational awareness using the Globe Manager.

1. Bring the 3D Graphics window to the front.
2. Click Globe Manager ( ) in the 3D Graphic window's Globe Manager toolbar.
3. Click Add Terrain/Imagery ( ) in the Globe Manager Hierarchy toolbar when Globe Manager opens.
4. Select Add Terrain/Imagery... ( ) in the drop-down menu.
5. Click the Path ellipsis (...) when the Globe Manager: Open Terrain and Imagery Data dialog box opens.
6. Navigate to <Install Dir>\Data\Resources\stktraining\imagery (for example, C:\Program Files\AGI\STK_ODTK 13\Data\Resources\stktraining\imagery) in the Select Image Directory list when the Browse For Folder dialog box opens.
7. Click **OK** to select the directory and to close the Browse For Folder dialog box.
8. Select the check box for RaistingStation.pdtt.
9. Click **Add** .
10. Click **Yes** to enable terrain for analysis and to close the Use Terrain for Analysis prompt.

Decluttering the 3D Graphics window labels

Enable Label Declutter to separate the labels on objects that are in close proximity for better identification in small areas.

1. Bring the 3D Graphics window to the front.
2. Click Properties ( ) in the 3D Window Defaults toolbar.
3. Select the Details page.
4. Select the Enable check box in the Label Declutter panel.
5. Click **OK** to confirm your selection and to close the Properties Browser.

Modeling the radar site

The radar site is located in the Alpine Foreland in Southern Germany.

Inserting a new Facility object

Use a Facility object to model the radar site.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Select Facility ( ) in the Select An Object To Be Inserted list.
3. Select Insert Default ( ) in the Select A Method list.
4. Click **Insert...**
5. Right-click on Facility1 ( ) in the Object Browser.
6. Select Rename in the shortcut menu.
7. Rename Facility1 ( ) Radar_Site.

Moving the radar site to its location

Update the Facility object's position properties to place it in Germany.

1. Open Radar_Site's ( ) Properties ( ).
2. Select the Basic - Position page.
3. Set the following position options:

Option	Value
Latitude	47.8996 deg
Longitude	11.1142 deg

4. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Defining the facility's Azimuth-Elevation Mask

Define an Azimuth -Elevation Mask (AzEl Mask) to use the local terrain analytically. An AzEl mask represents obstructions to visibility from the point of view of a Facility, Place, Target, or Sensor object. The AzEl mask is built by observing in all directions around the object.

1. Select the Basic - AzElMask page.
2. Set the following options:

Option	Value
Use	Terrain Data
Max range to consider	160 km
Use Mask for Access Constraint	Selected

   Selecting the Use Mask for Access Constraint check box enables the Az-El Mask constraint. The AzEl Mask constraint leverages an AzEl Mask to determine visibility. The mask may be computed from terrain information to be representative of terrain based visibility restrictions. Your terrain data does not include structures such as buildings.

3. Click **OK** to confirm your changes and to close the Properties Browser.

Modeling the radar site's tracking radar

The facility is equipped with a tracking radar.

Using a Sensor object to define the radar's field of view

Use a Sensor object to simulate the radar system's field of view (FOV) from the site.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Sensor ( ) object using the Insert Default ( ) method.
3. Select Radar_Site ( ) in the Select Object dialog box.
4. Click **OK** .
5. Rename Sensor1 ( ) Radar_FOV.

Modeling the radar's field of view

Use a Complex Conic sensor pattern to model the radar's field of view. Complex Conic sensor patterns are defined by the inner and outer half angles and minimum and maximum clock angles of the sensor's cone.

1. Open Radar_FOV's (![]) Properties (![]).
2. Select the Basic - Definition page.
3. Open the Sensor Type drop-down list.
4. Select Complex Conic.
5. Enter 180 deg in the Outer field in the Half Angles panel.
6. Click **Apply** to accept your changes and to keep the Properties Browser open.

By setting the Half Angles - Outer value (that is, the vertical angle) to 180 deg and leaving the Clock Angles (or horizontal angle) values at the default, you've created a 360-degree field of view.

Raising the antenna's position

The radar's antenna is positioned twenty feet above the ground's surface. The Sensor Location properties enable you to position a sensor with respect to its parent object. The Facility object's positive (+) Z body points to the center of the Earth; if you want to move the Sensor object up, you have to use a negative (-) Z value.

1. Select the Basic - Location page.
2. Open the Location Type drop-down list.
3. Select Fixed.
4. Enter -20 ft in the Z field in the Fixed Location panel.
5. Click **Apply** to accept your changes and to keep the Properties Browser open.

Using the AzEl Mask constraint

An object that is the child of another, such as a sensor attached to a facility, does not automatically inherit constraints imposed from its parent, but you can set constraints on the Sensor object to use its parent object's AzEl Mask.

1. Select the Constraints - Active page.
2. Click Add new constraints (![]) in the Active Constraints toolbar.
3. Select Az-El Mask in the Constraint Name list when the Select Constraints to Add dialog box appears.
4. Click **Add** .
5. Click **Close** to close the Select Constraints to Add dialog box.
6. Click **Apply** to accept your changes and to keep the Properties Browser open.

Visualizing the AzEl Mask

The sensor's 2D Graphics Projection properties control the display of sensor projection graphics in the 2D and 3D Graphics windows. In order to visualize the constraints that the Sensor object is using, you have to define which constraints can be used to modify the sensor's field of view.

1. Select the 2D Graphics - Projection page.
2. Select the Use Constraints check box in the Field of View panel.
3. Select AzElMask in the list.
4. Click **Apply** to accept your changes and to keep the Properties Browser open.

Defining the 3D Graphics Projection properties

The 3D Graphics Projection properties are used to control the display of a sensor's cone and extension distances into 3D space. Extension distances define the length of a sensor's projection. For a constant space projection, enter the projection length in the Space Projection field. In this case, the distance is computed so that the projection of the outermost point on the contour along the bore sight is equal to the distance entered. Note this is a visualization property, not an analytical property.

1. Select the 3D Graphics - Projection page.
2. Enter 50 km in the Space Projection field in the Extension Distances panel.
3. Click **OK** to confirm your changes and to close the Properties Browser.

Viewing the radar antenna's field of view

View the projected field of view of the radar antenna in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Home View ( ) in the 3D Graphics toolbar.
3. Change you view so that you can see Radar_FOV's ( ) field of view.

If you were only using the Line of Sight constraint, the sensor's field of view would be circular. In this instance, you are taking into account the central body (Earth) and terrain, which is causing the blockage of the sensor's field of view.

Inserting the satellites

The radar site's primary purpose is to track three Technology Development satellites.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Satellite ( ) object using the From Standard Object Database ( ) method.
3. Set the following search criteria when the Search Standard Object Database dialog box opens:

Option	Value
Owner	Germany
Mission	Technology Development

4. Click **Search** .
5. Multi-select MAROC-TUBSAT, BeeSat and DLR-Tubsat in the Results list.
6. Click **Insert** .
7. Click **Close** to exit the Standard Object Database tool after the satellites are propagated.

Determining benchmark access to the satellites

Determine the total time each satellite appears within the sensor's field of view. That is considered the total access duration. You will use this value as a benchmark to see if the radome affects accesses to the satellites.

Computing access

The STK application computes access durations by determining the time intervals over which all applicable constraints are satisfied.

1. Right-click on Radar_FOV ( ) in the Object Browser.
2. Select Access... ( ) in the shortcut menu.

3. Select all three Satellite (icon) objects in the Associated Objects list when the Access tool opens.
4. Click Compute .

Generating an Access report

An Access report reports a list of the access intervals.

1. Click **Access...** in the Reports panel.
2. Scroll to the bottom of the report.
3. Note the value of the Total Duration in the Global Statistics section (for example, approximately 15,100 seconds).

Saving the Access report as an external text file

Save the Access report outside of the STK application. This is a safe way to retain the original analysis values.

1. Return to the Access report.
2. Click Save as text (icon) in the Access report toolbar.
3. Ensure your scenario folder displays in the in Address bar in the Save Report dialog box.
4. Enter Sensor to Satellites Terrain Only in the File name field.
5. Click **Save** .
6. Close the Access report.
7. Close the Access tool.

Modeling the communications radome

Construction crews plan to build a large communications site radome less than one-half kilometer from the proposed radar site. You want to model the radome to analyze potential interference.

Inserting another facility object

Use another Facility object as the radar site location.

1. Bring the Insert STK Objects tool () to the front.
2. Insert a Facility () object using the Insert Default () method.
3. Rename Facility2 () Comm_Radome.
4. Open Comm_Radome's () Properties ().
5. Select the Basic - Position page.
6. Set the following position options:

Option	Value
Latitude	47.899 deg
Longitude	11.1113 deg

7. Click OK to confirm your changes and to close the Properties Browser.

Viewing the communications radome in relation to the radar field of view

View the radome in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on Comm_Radome () in the Object Browser.
3. Select Zoom To in the shortcut menu.
4. Change your view so that you can see both the communications site radome and the and radar site.
5. Use your mouse to set the view so that you can see how Radar_FOV's () projection cuts through Comm_Radome's () 3D Model.

If you look closely, you can see that the sensor's projection cuts through the communications site's radome. The Facility object's 3D Graphics model is not being used as an obstruction during analysis. In order to use it as an obscuring object, you must use the AzEl Mask tool.

> **Note:** In this scenario, you're using a Facility object to simulate the communications site's radome. If you were actually performing this analysis against an actual building, you would need to consider creating your own 3D model built to specifications.

Using the AzEl Mask tool

Use the AzEl Mask tool to create a body mask file (.bmsk) that can be used in access computations and visualization. Body mask files provide data used to restrict visibility to a sensor. The term "body masking" refers to line of sight obstruction caused by the three dimensional model of the parent object of the sensor or other objects in the scenario. The body masking files contain obscuration contours which are generated based on six views generated from the point of view of an observer at the location of the sensor.

Opening the AzEl Mask tool

You can access the AzEl Mask tool from the Sensor menu in the Menu Bar.

1. Maximize your 3D Graphics window.
2. Select Radar_FOV ( ) in the Object Browser.
3. Select the Sensor menu in the Menu Bar at the top of the STK application.
4. Select AzEl Mask... in the Sensor menu.

Preparing the AzEl Mask tool

Start by setting up the AzEl Mask tool prior to creating a body mask file. The tool's Az/El Mask View window allows you to see the obscuring objects in the six views used in generating the contours. The views will be shown in successive fashion when the Compute button is clicked. The tool's AzEl Mask dialog box enables you to identify obscuring objects and define the instant in time at which obscuration contours are computed. Set Comm_Radome as the obscuring object and the window dimension to 500.

1. Move the AzEl Mask dialog box (AzElMask for Radar_FOV) to the right so that it isn't on top of the Az/El Mask View window.
2. Select Comm_Radome ( ) in the AzEl Mask window's Obscuring Objects list.
3. Set the Window Dim value to 500 in the Data panel.
4. Click **Apply** .
5. Click **Compute...**
6. Ensure your .bmsk file is being saved in your scenario folder when the Select Body Mask File dialog box opens. Use the default file name.

7. Click **Save** .
8. Close the AzEl Mask dialog box and the Az/El Mask View window when the computation is complete.

Constraining the sensor with the AzEl Mask

Use the body mask file as the sensor AzEl mask and apply it as the as an Access constraint on the sensor.

1. Open Radar_FOV's ( ) Properties ( ).
2. Select the Basic - Sensor AzEl Mask page.
3. Open the Use drop-down list.
4. Select MaskFile.
5. Click the Mask File ellipsis (...).
6. Browse to your scenario folder, if required, when the Select File dialog box opens.
7. Select Radar_FOV.bmsk in the list.
8. Click **Open** .
9. Select the Use Mask for Access Constraint check box.
10. Click **Apply** to accept your changes and to keep the Properties Browser open.

Visualizing the sensor AzEl Mask

In order to visualize the Sensor AzEl Mask constraint, follow the same procedure as you did to visualize the terrain Az-El Mask.

1. Select the 2D Graphics - Projection page.
2. Leave AzElMask selected in the Field of View - Use Constraints panel.
3. Scroll down the Field of View - Constraints list until you locate SensorAzElMask.
4. Use Ctrl + click to select SensorAzElMask in the list in addition to AzElMask, which you selected earlier.
5. Click **OK** to confirm your changes and to close the Properties Browser.

Viewing the sensor AzEl Mask in the 3D Graphics window

You can view the constrained FOV in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom To Radar_Site ( ).
3. Change your view so that you can see Radar_Site ( ), Comm_Radome ( ), and Radar_FOV's ( ) field of view being affected by Comm_Radome ( ).

Determining changes to the total access duration

Determine the total time each satellite appears within the sensor's field of view now that the sensor FOV takes obscuration by the radome into account using the Sensor AzEl mask.

1. Right-click on Radar_FOV ( ) in the Object Browser.
2. Select Access... ( ) in the shortcut menu.
3. Select all three Satellite ( ) objects in the Access tool's Associated Objects list.
4. Click Access... in the Reports panel.
5. Scroll to the bottom of the report.
6. Note the Total Duration value in the Global Statistics section (for example, approximately 15,000 seconds).

Comparing the access data

Compare access data between the total access durations when using just the AzEl mask and when using boht th terrain AzEl mask and the facility body mask file. The best result would be that you don't lose any of the total access duration.

1. Open Windows File Explorer.
2. Browse to your scenario folder (e.g. C:\Users\<username>\Documents\STK_ODTK 13\AzElMask_Volumetrics).

3. Open the Sensor to Satellites Terrain Only.txt file.
4. Return to the STK application.
5. Compare the Global Statistics Total Duration time in the text file to the Total Duration time in the Access Report.
6. Close both reports and the Access tool when finished.

Using Volumetric Analysis and the Analysis Workbench to analyze the radar's field of view

Based on the curvature of the Earth and that the satellites being tracked by the radar are in a low Earth orbit (LEO), you want to determine how much of the radar's field of view is blocked by the Earth itself, surrounding terrain, and the radome within a 3,000-kilometer radius from the radar site. In order to determine how much the radar field of view is affected, you will use an Area Target object in conjunction with the *Volumetric Analysis* capability and the *Analysis Workbench* capability's Time tool and Spatial Analysis tool to analyze a 3D volume of space inside the portion of a spherical sector between 10 and 700 kilometers in altitude from the radar site. Volumetric Analysis involves evaluation of spatial conditions and calculations across a number of grid points. This analysis is configured in a Volumetric object. A Volumetric object defines a 3D grid of points using various coordinate definitions and the conditions and calculations that depend on locations in 3D space. It evaluates these conditions and calculations across grid points. The Spatial Calculations and Volume grids are defined using the Spatial Analysis tool in the Analysis Workbench. The Spatial Analysis tool enables you to create calculations and conditions that depend on locations in 3D space which are, in turn, provided by user-definable Volume grids.

Inserting an Area Target object

Insert an Area Target () object to model the 3,000-kilometer radius analysis area on the globe.

1. Bring the Insert STK Objects tool () to the front.
2. Insert an Area Target () object using the Area Target Wizard () method.
3. Set the following options in the Area Target Wizard ():

Option	Value
Name	OpsArea
Area Type	Ellipse

Semi-Major Axis	3000 km
Semi-Minor Axis	3000 km
Centroid - Latitude	47.8996 deg
Centroid - Longitude	11.1142 deg

4. Click **OK** to confirm your changes and to close the Area Target Wizard.

Viewing the Area Target object in the 3D Graphics window

View the Area Target in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Home View ( ) in the 3D Graphics window toolbar.
3. Move your view so that you can see OpsArea ( ).

Opening the Time tool

There is no need to calculate the radar's field of view for the entire 24-hour analysis period. Use the *Analysis Workbench* capability's Time tool to create a one-second fixed time interval. You will use this interval when analyzing the radar's field of view.

1. Right-click on AzElMask_Volumetrics ( ) in the Object Browser.
2. Select Analysis Workbench... ( ) in the shortcut menu.
3. Select the Time tab when the Analysis Workbench opens.
4. Ensure AzElMask_Volumetrics ( ) is selected in the Object list on the left.

Creating a Fixed Interval

An Interval component produces a single interval of time. You want a to create a one-second Fixed Interval to use in your analysis.

1. Click Create new Interval ( ).
2. Click **Select...** for Type when the Add Time Component dialog box opens.

3. Select Fixed Interval ( ) in the Select Component Type list when the Select Component Dialog box opens.
4. Click **OK** to confirm your selection and to close the Select Component Dialog box.
5. Enter One_Second in the Name field.
6. Enter 15 Mar 2024 12:00:01.000 UTCG in the Stop Time field.
7. Click **OK** to confirm your changes and to close the Add Time Component dialog box.

Creating a Cartographic Volume grid

Use the Spatial Analysis tool create two Volume grids: a Cartographic Reference grid and a Constrained grid. The Reference grid will be used to create a 3D volume of space encompassing the Area Target object's sphere between 10 to 700 kilometers in altitude. First, you must create your Reference grid.

1. Select the Spatial Analysis tab at the top of the Analysis Workbench.
2. Select OpsArea ( ) in the object list.
3. Click Create new Volume Grid ( ).
4. Ensure the Type is Cartographic in the Add Spatial Analysis Component dialog box.

   A Cartographic grid uses latitude, longitude and altitude based on a central body reference ellipsoid.
5. Enter Ref_Grid in the Name field.
6. Click **Set Grid Values...**
7. Set the following values in the Altitude panel when the Grid Values dialog box opens:

Option	Value
Minimum	10 km
Maximum	700 km
Number of Steps	20

   The number of steps determines how many grid points are added to the volume for computation and analysis.
8. Click **OK** to close the Grid Values dialog box.
9. Click **OK** to close the Add Spatial Analysis Component dialog box.

Creating a Constrained Volume grid

A Constrained Volume grid is one in which the grid points from the reference grid are available only when a specified spatial condition is satisfied.

1. Select OpsArea () in the Object list.
2. Click Create New Volume Grid ( ).
3. Enter OpsArea_Constrained in the Name field when the Add Spatial Analysis Component dialog box opens.
4. Click **Select...** for Type.
5. Select Constrained ( ) in the Select Component Type list when the Select Component Type dialog box opens.
6. Click **OK** to confirm your selection and to close the Select Component Type dialog box.

Selecting the Reference grid

Select the Cartographic grid you created previously as the Reference grid.

1. Click the Reference Grid ellipsis (...).
2. Select OpsArea () in the Object list when the Select Reference Volume Grid dialog box opens.
3. Select Ref_Grid ( ) in the Volume Grids for: OpsArea list.
4. Click **OK** to close the Select Reference Volume Grid dialog box.

Choosing the spatial condition

Select the Radar_FOV's () Visibility as the spatial condition used for the volumetric analysis.

1. Click the Spatial Condition ellipsis (...) when you return to the Select Reference Spatial Condition dialog box.
2. Select Radar_FOV () in the object list when the Select Reference Spatial Condition dialog box opens.
3. Select Visibility ( ) in the Spatial Conditions for: Radar_FOV list.

This will apply the constrained visibility of Radar_FOV ( ) to the 3D Volume grid.

4. Click OK to close the Select Reference Spatial Condition dialog box.
5. Click OK to close the Add Spatial Analysis Component dialog box.
6. Click Close to close the Analysis Workbench.

Inserting a Volumetric object

Insert a new Volumetric object into your scenario.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Volumetric ( ) object using the Insert Default ( ) method.

Defining the Volume Grid

The default Volume grid encircles the Earth up to an altitude of 1,000 kilometers. Change the grid's definition to use your constrained grid.

1. Open the Volumetric1's ( ) Properties ( ).
2. Select the Basic - Definition page.
3. Click the Volume Grid ellipsis (...).
4. Select OpsArea ( ) in the Object list when the Select Volume Grid for Volumetric1 dialog box opens.
5. Select OpsArea_Constrained ( ) in the Volume Grids for: OpsArea list.
6. Click OK to close the Select Volume Grid for Volumetric1 dialog box.
7. Click Apply to accept your changes and to keep the Properties Browser open.

Selecting the Spatial Calculation

A Spatial Calculation is a scalar calculation that depends on both time and location.

1. Select the Spatial Calculation check box.
2. Click the Spatial Calculation ellipsis (...).

3. Select OpsArea ( ) in the Object list when the Select Spatial Calculation for Volumetric1 dialog box opens.
4. Select Altitude ( ) in the Spatial Calculations for: OpsArea list.
5. Click **OK** to close the Select Spatial Calculation for Volumetric1 dialog box.
6. Click **Apply** to accept your changes and to keep the Properties Browser open.

Selecting the analysis Interval

To evaluate the Spatial Calculation on your Volume grid, update the Volumetric object's Interval properties to apply the one-second Fixed Interval you created using the Time tool.

1. Select the Basic - Interval page.
2. Click the Analysis Interval ellipsis (...).
3. Select AzElMask_Volumetrics ( ) in the object list when the Select Interval or List dialog box opens.
4. Select One_Second ( ) in the Components for: AzElMask_Volumetrics list.
5. Click **OK** to close the Select Interval or List dialog box.
6. Click **Apply** to accept your selection and to keep the Properties Browser open.
7. Save ( ) your scenario.

Computing visibility inside the grid

With your analysis configured, compute the volumetric. Your best-case scenario would have 100 percent visibility inside the grid. However, the curvature of the Earth, terrain, and the radome will reduce this percentage.

1. Select Volumetric1 ( ) in the Object Browser.
2. Select the Volumetric menu in the Menu Bar.
3. Select Compute.

Generating a Satisfaction Volume report

Generate a report that shows how much of the radar's field of view is visible. The Satisfaction Volume report style reports the Volume grid properties and the percentage of volume of the entire grid that has been active at some moment up to the current time.

1. Right-click on Volumetric1 () in the Object Browser.
2. Select Report & Graph Manager... () in the shortcut menu.
3. Select the Satisfaction Volume () report in the Installed Styles () folder when the Report & Graph Manager opens.
4. Click **Generate...** .

   You can see in the report that the radar field is approximately 44 percent satisfied. Remember, you computed this starting at a low altitude of only 10 kilometers; there will be more losses at lower altitudes due to the central body (Earth), terrain and the radome.
5. Close the Satisfaction Volume report and the Report & Graph Manager when finished.

Displaying visibility inside the grid

The Volumetric object's 3D Graphics Grid properties are used to define the 3D Graphics Volumetric grid properties for the Volumetric Definition.

1. Return to Volumetric1's () Properties.
2. Select the 3D Graphics - Grid page.
3. Clear the Show Grid check box.

   This will make the grid look better for a briefing or presentation. If you were analyzing something like Bit Error Rates per grid point, you might leave this on. By clicking on a point in the 3D Graphics window, you will receive a value for that point. In this scenario, they are on or off.
4. Click **Apply** .

Adding Spatial Calculation levels

Update the Volumetric object's 3D Graphics Volume properties to show Spatial Calculation levels. These levels represent straight line distances from the parent object.

1. Select the 3D Graphics - Volume page.
2. Select the Spatial Calculation Levels option.
3. Click **Insert Evenly Spaced Values...** at the bottom of the page.

4. Set the following when the Insert Evenly Spaced Values dialog box opens:

Option	Value
Units	km
Start Value	10
Stop Value	700
Step Size	100

5. Click **Create Values** .

Adjusting the translucency

You can adjust translucency of the colors in order to make the levels stick out or fade depending on your desired view. In this case, you want to be able to see the lower altitude colors more than the higher altitude colors. The Earth, terrain, and the communications radome affect the lower-altitude colors more than the higher altitude colors. You can use the Translucency slider or manually type in the percentage. In this case, type them in.

1. Set the following in the % column, which is located in the Fill Levels list:

Value km	%
10	10
110	20
210	40
310	40
410	50
510	50
610	60
700	60

2. Click **Apply** .

Displaying a 3D Graphics Legend

The Volumetric object's 3D Graphics Legends properties allows you to place a legend in the 3D Graphics window. which explains what the colors of your contours mean.

1. Select the 3D Graphics - Legends page.
2. Select the Fill Legend tab.
3. Set the following options:

Option	Value
Show Legend	Selected
Text Options - Title	Altitude (km)
Text Options - Number Of Decimal Digits	0
Range Color Options - Color Square Width (pixels)	40

4. Click OK .

Viewing the Volumetric contours

1. Bring the 3D Graphics window to the front.
2. Use your mouse to change your view to get an idea of how obscurations affect the different levels of altitude.

Summary

You loaded analytical terrain into your scenario that covers the area in which the radar site is going to be built. You then used a Sensor object to create the field of view of the radar. You propagated two satellites and generated an access between the sensor and the satellites to create a benchmark access time. Next, you placed a Facility object where a new communications radome will be constructed. You used the AzEl Mask tool to determine if the radome affects the sensor's field of view. Generating another Access report, you determined that the radome will affect your overall access time to the satellites. Next, you used the Analysis Workbench Time tool to create a one second interval to be used with a Volumetric object's compute time. Then, using the Analysis Workbench Spatial Analysis tool, you created a reference grid inside of an Area Target object and a constrained grid which then applied the Sensor object's constraints to the 3D volume of space. You determined that a significant amount of your volume is obscured by the Earth, terrain and the radome.

Part 10: Performing Trade Studies with ModelCenter

Note: Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Important: Required product install: The Ansys ModelCenter® model-based systems engineering software and the STK Plugin for ModelCenter are required to complete this tutorial.

Important: ModelCenter installation prerequisites: The ModelCenter software requires the installation of a 64-bit version of Java, a 64-bit implementation of Python 3.x, and the installation of the thrift and six Python packages. See the ModelCenter Installation Prerequisites for more information.

Note: This tutorial was written using version 2026 R1 of the Ansys ModelCenter® model-based systems engineering software.

Note: The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators require an easy way to determine how a satellite's orbital parameters, such as its semi-major axis, eccentricity, inclination, argument of perigee, right ascension of the ascending node, and true anomaly, affect the ability a sensor or camera on board to view the surface of the Earth. You want to quickly run multiple trade studies to determine how the inclination, eccentricity, and the combination of both will effect the percentage of coverage over the entire Earth during a 24-hour analysis period.

Solution

Use the STK software's *Coverage* capability and the *Analyzer* capability, which is part of the Ansys ModelCenter® model-based systems engineering software, to perform parametric and carpet plot trade studies, which will determine the combination of inclination and eccentricity that provides the highest percentage of global coverage.

Creating a new scenario

First, you must create a new scenario, then build from there.

1. Launch the STK application ( ).
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following in the New Scenario Wizard:

Option	Value
Name	STK_ModelCenter
Location	Default
Start	8 Jun 2026 16:00:00.000
Stop	9 Jun 2026 16:00:00.000

4. Click OK when you finish.
5. Click Save ( ) when the scenario loads. The STK software creates a folder with the same name as your scenario for you.
6. Verify the scenario name and location in the Save As window.
7. Click Save .

Cleaning up your workspace

You can view coverage in both the 2D and 3D Graphics windows. Closing the Timeline View and placing the graphics windows side by side makes this simple to do.

1. Close ( ) the Timeline View at the bottom of the STK application window.
2. Select the Window menu in the Menu Bar.
3. Select Tile Vertically.

Your 2D and 3D Graphics windows will fill the available workspace.

Turning off streaming terrain

Analytical and visual terrain is not required in this analysis, so you can turn off streaming terrain from the Terrain Server.

1. Open STK_ModelCenter's ( ) Properties ( ).
2. Select the Basic - Terrain page.
3. Clear the Use terrain server for analysis check box in the Terrain Server panel.
4. Click OK to accept your changes and to close the Properties Browser.

Designing an initial orbit

Use the Orbit Wizard to insert a Satellite object and place it in a retrograde orbit.

1. Bring the Insert STK Objects Tool ( ) to the front.
2. Insert a Satellite ( ) object using the Orbit Wizard ( ) method.
3. Select the Orbit Designer Type when the Orbit Wizard opens.

   With the Orbit Designer, you can create any orbit you wish.
4. Enter MySat for the Satellite Name.
5. Set the following Orbital Elements:

Option	Value
Semimajor Axis	10600 km
Eccentricity	0.363
Inclination	116 deg
Argument of Perigee	270 deg
RAAN	104 deg
True Anomaly	90 deg

6. Click OK to propagate the satellite and to close the Orbit Wizard.

Attaching a Sensor object to the satellite

Use a Sensor object to model a camera with a 20-degree field of view.

1. Insert a Sensor ( ) object using the Define Properties ( ) method.
2. Select MySat ( ) in the Select Object dialog box.
3. Click **OK** to confirm your selection and to close the Select Object dialog box.
4. Note that the default Sensor Type is set to Simple Conic when the Properties Browser opens.

   A Simple Conic sensor pattern is defined by a simple cone angle, measured in terms of the cone half angle.
5. Set the Cone Half Angle to 10 deg.
6. Click **OK** to accept your change and to close the Properties Browser.
7. Rename Sensor1 ( ) Camera_View.

Using the Coverage capability

The STK software's *Coverage* capability allows you to analyze the global or regional coverage provided by one or more assets while considering all available accesses. To address area coverage, the *Coverage* capability provides you with two STK object classes: Coverage Definition objects and Figure of Merit objects.

Inserting a Coverage Definition object

You want to assess coverage over the entire globe. To do this, you will use a Coverage Definition object to model the area that you will analyze. Coverage Definition objects enable you to define and maintain an area of coverage, to define the STK objects providing coverage for the area (such as satellites, aircraft and sensors), to define the time period of interest, and to calculate accesses to the region.

1. Insert a Coverage Definition ( ) object using the Insert Default ( ) method.
2. Rename CoverageDefinition1 ( ) Global_Grid.

Defining the Coverage Grid Area of Interest

Coverage analyses are based on the accessibility of assets (that is, the objects that provide coverage) to the geographical areas of interest. For analysis purposes, the geographical areas of interest are further refined using regions and points defined by the STK application. Points have specific geographical locations and are used in the computation of asset availability. Regions are closed boundaries that contain sets of points. The combination of the geographical area, the regions within that area, and the points within each region is called the coverage grid. For this scenario, create a grid that covers the entire globe.

1. Open Global_Grid's ( ) Properties ( ).
2. On the Basic - Grid page, change Grid Area of Interest - Type to Global.
3. Click **Apply** to accept your change and to keep the Properties Browser open.

Selecting the coverage assets

Assign the satellite's camera as your coverage asset. Assets properties enable you to specify the STK objects used to provide coverage.

1. Select the Basic - Assets page.
2. Expand (⊞) MySat ( ) in the Assets list.
3. Select Camera_View ( ).
4. Click **Assign** .
5. Click **OK** to accept your changes and to close the Properties Browser.

Inserting a Figure of Merit object

The STK software enables you to specify the method by which the quality of coverage is measured using a Figure Of Merit object. The default figure of merit type is Simple Coverage. Simple Coverage measures whether a point is accessible by any of the assigned assets.

1. Insert a Figure Of Merit ( ) object using the Insert Default ( ) method.
2. Select Global_Grid ( ) when the Select Object dialog box opens.
3. Click **OK** .
4. Rename FigureOfMerit1 ( ) Simple_Cov.

Computing accesses

The ultimate goal of coverage is to analyze accesses to an area using assigned assets. Compute accesses by opening the Compute Accesses tool.

1. Right-click on Global_Grid ( ) in the Object Browser.
2. Select CoverageDefinition in the shortcut menu.
3. Select Compute Accesses in the CoverageDefinition submenu.
4. Bring the 2D Graphics window to the front.
5. Review the satellite's areas of coverage across the globe.

Note that coverage is more complete at higher latitudes.

Creating a Percent Satisfied report

You're interested in the percent of the earth's surface covered as defined by your Simple Coverage definition, meaning the percentage of the coverage grid area covered by at least one asset at some point during the coverage interval. To find this static value, generate a Percent Satisfied report in the Report & Graph Manager. Percent Satisfied reports the percentage of the coverage grid area which is satisfied; it's computed by summing the areas associated with all satisfied grid points, dividing by the total grid area, and multiplying by 100.

1. Right-click on Simple_Cov ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select the Percent Satisfied ( ) report style in the Installed Styles ( ) folder in the Styles list.
4. Click **Generate. . .** .
5. Note the % Satisfied value at the bottom of the report.

   The value you see is the scenario benchmark, which, in this case, is approximately 49 percent.
6. When finished, close the report and the Report & Graph Manager.

Closing out of the STK application

With your baseline scenario configured, save your scenario and close out of the STK application in preparation for the next step.

1. Save ( ) your scenario.
2. Close any open reports, the Report & Graph Manager, and any open tools.
3. Close the STK application.

Creating a new ModelCenter project

The Ansys ModelCenter software is designed for multidisciplinary analysis and design optimization. It allows you to automate complex workflows, integrate multiple engineering tools, and optimize designs within model-based systems engineering frameworks. By wrapping analysis programs like the STK software and running them in an automated fashion, the ModelCenter software makes the design process more efficient, saves engineering time, and reduces the chances for error in the design process.

1. Open the ModelCenter ( ) application.
2. Click **Start a New Model** in the Welcome to ModelCenter dialog box.
3. Click **Process** when the What type of model would you like to create? dialog box opens.
4. Navigate to your scenario folder (for example, C:\Users\<username>\Documents\STK_ODTK 13\STK_ModelCenter.
5. Enter STK_ModelCenter in the File name field.
6. Ensure the Save as type is set to the ModelCenter Model (Zip) (*.pxcz).
7. Click **Save** .

Configuring the STK Plugin for ModelCenter

The STK Plugin for ModelCenter is a component plugin that allows the ModelCenter application to directly link to the STK application. It imports a copy of the desired Scenario and opens it in an instance of the STK application. Instantiate the plugin by adding it to your workflow's analysis view from the Server Browser.

1. Select favorites (★) in the Server Browser at the bottom of the window.

   The Server Browser resides at the bottom of the ModelCenter window and is used to browse for components that can be used in ModelCenter.

2. Click and drag the STK component () into the dashed circle underneath "Drop items here to build the model" in your workflow's Analysis View.

3. Select STK_ModelCenter.sc when the Open STK Scenario file dialog box opens.

4. Click **Open** .

5. After a few moments, the STK Analyzer window will open.

> **Note:** The STK_ModelCenter scenario file will open in the STK application in the background.

Specifying the variables for analysis with Analyzer

Use the STK Analyzer window to configure the input and output variables available for further analysis with the *Analyzer* capability.

Selecting the input variables

Your studies will focus on only two of the six orbital elements you set at the beginning of the lesson – inclination and eccentricity. You need to select input and output variables from the main STK Analyzer window to pass to the ModelCenter software's Trade Study tools.

1. Select MySat () in the STK Variables tree.

   When you select an object in the STK Variables tree, all possible input variable candidates for that object are listed under the General tab and the Active Constraints tab in the STK Property Variables panel.

2. Expand (⊞) the Propagator (TwoBody) () property in the STK Property Variables tree.

3. Select Inclination ().

4. Move () Inclination () to the Analyzer Variables list.

5. Select Eccentricity ().

6. Move () Eccentricity () to the Analyzer Variables list.

Note that both Eccentricity and Inclination are listed as Inputs in the Analyzer Variables list.

Selecting the output variable

The same data providers that are available in the Report & Graph Manager are available in the Data Provider Variables tree. Select Percent Satisfied as the Output variable.

1. Expand (⊞) Global_Grid ( ) in the STK Variables tree.
2. Select Simple_Cov ( ).
3. Expand (⊞) the Static Satisfaction ( ) data provider.
4. Select the Percent Satisfied ( ) data provider element.
5. Move ( ) Percent Satisfied ( ) into the Analyzer Variables list.

   Note that Percent Satisfied is listed under Outputs in the Analyzer Variables list.
6. Click **OK** to confirm your selections and to close the STK Analyzer window.

   This will also close the STK application, which had been running in the background.

Determining the impact of satellite inclination on the percentage satisfied

The first study you will perform varies inclination and its effect on global coverage.

Using the Parametric Study tool

The Parametric Study tool runs a workflow through a sweep of values for some input variable. You can plot the resulting data to view trends. Set up a parametric study using inclination as the design variable and percent satisfied as the response.

1. Expand (⊞) all the elements in the Component Tree.
2. Click Parametric Study ( ) in the Standard toolbar.
3. Click and drag Inclination ( ) from the Component Tree to the Design Variable field when the Parametric Study tool opens.
4. Set the following Design Variable values:

Option	Value

starting value	112
ending value	120
step size	1

Note the number of samples is automatically set to 9, calculated from the values you set.

5. Click and drag Percent_Satisfied ( ) from the Component Tree to the Responses field.
6. Click **Run...** .

Clicking **Run...** will open the Data Explorer, which is a tool used by Trade Study tools to display data collected from a Model. While data are being collected, the Data Explorer displays a progress meter, a halt button, and the data.

Reviewing the Table page data

The Table page of the Data Explorer displays trade study data in a tabular form. It is the default window that is present for all trade studies. Cells are shaded differently depending on the associated variable's state. Input variables are shown with green text, valid values are displayed with black text, invalid values are displayed with gray text, and modified values are displayed with blue text. From the table it is possible to view and edit all values in your trade study and even to add and remove whole runs.

1. Close the 2D Scatter Plot that opened when the trade study finished running.
2. Bring the Table page to the front when all runs finish.
3. Examine the results in the Data Explorer table.

Notice nine runs were performed in one-degree increments from 112 degrees to 120 degrees inclination. Notice the second row shows the global coverage percentage for each change in inclination, and that an inclination of 120 degrees provides the highest percentage of global coverage.

Creating a 2D Line Plot

For this study, you will create a 2D Line Plot. A 2D Line Plot displays an X-Y plot for variables in your model. Any variable in the workflow can be plotted against any other variable.

1. Close the 2D Scatter Plot that opened when the trade study finished running.
2. Click Add View ( ) on the Table Page toolbar.
3. Select 2D Line Plot ( ) in the drop-down list.

Setting options for the axes

Use the Axes tab to set options for the axes.

1. Click Axes in the Plot Options menu.

   The Axes tab is used to set options for the plot's axes.

2. Select the Ticks tab.

   The Ticks tab is used to set the display of ticks along the axes.

3. Change the Max # value to 20.
4. Click anywhere on the plot to close the Plot Options menu.
5. Review the 2D Line Plot.

Looking at the 2D Line Plot, you see that inclination of 120 degrees, basing the analysis on one-degree increments, gives you the best choice for global coverage during the 24-hour analysis period.

Closing out your trade study

Close out your trade study to prepare for the next section.

1. Close the 2D Line Plot and the Table page when you are finished.
2. Click **No** when prompted to close your trade study without saving.
3. Leave the Parametric Study tool open.

Studying the satellite's eccentricity

Eccentricity could have an impact on the sensor's footprint. Note that you have to take into consideration the possibility of the satellite colliding with the Earth's surface when changing its eccentricity.

Running a new Parametric Study

Set up a second Parametric Study with eccentricity as the Design Variable and Percent Satisfied as the response.

1. Click and drag Eccentricity (icon) from the Component Tree to the Design Variable field when the Parametric Study tool opens.

   This will replace Inclination as the Design Variable.

2. Set the following Design Variable values:

Option	Value
starting value	0.362
ending value	0.364
number of samples	10

3. Click **Run...** .

Creating a 2D Line Plot

Create a 2D Line Plot to investigate the data.

1. Close the 2D Scatter Plot that automatically opened after the trade study finished running.
2. Click Add View (icon) on the Table Page toolbar.
3. Select 2D Line Plot (icon) in the drop-down list.
4. Click Axes in the Plot Options menu.
5. Select the Ticks tab.
6. Change the Max # value to 40.
7. Click anywhere on the plot to close the Plot Options menu.
8. Hover over one of the design points. The Design Tooltip appears.

   The Design Tooltip allows you to quickly examine individual design points.

The small change in eccentricity didn't have too much impact on global coverage. 0.362000 through 0.36245 provided a steady value of 49.25 percent satisfied; after that, there is a drop in the satisfaction percentage.

Closing out your trade study

Close out your trade study to prepare for the next section.

1. Close the 2D Line Plot and the Table page when you are finished.
2. Click **No** when prompted to close your trade study without saving.

3. Close the Parametric Study tool.

Using the Carpet Plot tool to study inclination and eccentricity together

A Carpet Plot is a means of displaying data dependent on two variables in a format that makes interpretation easier than normal multiple curve plots. A Carpet Plot can be thought of as a multidimensional Parametric Study. Setting the design variables in a Carpet Plot is similar to using the Parametric Study tool, except you can study two variables simultaneously instead of one.

Creating a new Carpet Plot

Set up a Carpet Plot with inclination and eccentricity as the Design Variables and Percent Satisfied as the response.

1. Click Carpet Plot ( ) on the Analyzer toolbar to access the Carpet Plot tool.
2. Click and drag Inclination ( ) from the Component Tree to the first Design Variables field when the Carpet Plot tool opens.
3. Set the following Inclination Design Variable values:

Option	Value
From	119
To	121
Step Size	1

4. Click and drag Eccentricity ( ) from the Component Tree the second Design Variables field.
5. Set the following Eccentricity Design Variable values:

Option	Value
From	0.361
To	0.365
Step Size	0.001

6. Click and drag Percent_Satisfied ( ) from the Component Tree to the Responses field.
7. Click **Run...** .

Configuring the Carpet Plot's axes

Using the Carpet Plot tool, look for the best combination of inclination and eccentricity. First, make the Carpet Plot easier to read by adjusting its axes, then review the plot.

1. Bring the Carpet Plot to the front.
2. Click Axes in the Plot Options menu.
3. Select the Lines tab.
4. Change the Grid Lines value to 10.
5. Click anywhere on the plot to close the Plot Options menu.
6. Review the plot.

The original benchmark of global coverage was approximately 49 percent. In this study, an inclination of 119.6 degrees and an eccentricity of 0.361 provided the best percentage of global coverage of approximately 52 percent.

> **Note:** You can find the exact value for this run in the Data Explorer table.

Running your model

Confirm the data by running your model.

1. Close out any open plots, tools, and the Data Explorer window.
2. Click No when prompted to close your trade study without saving.
3. Click on the Value field for Inclination in the Component Tree.
4. Enter 119.6.
5. Click on the Value field for Eccentricity in the Component Tree.
6. Enter 0.361.
7. Click Run ( ) in the Standard Toolbar.
8. Note the progress bar on the STK_ODTK13 component as ModelCenter runs your model.
9. When it is complete, note that the value for Percent_Satisfied has been recalculated, and that its icon has changed from to , indicating the output is valid.

By running your model, you can see that Percent_Satisfied has been updated to 51.9328, confirming the results from your Carpet Plot Study.

Summary

You began the scenario by placing a Satellite object in a retrograde orbit. You attached a Sensor object to it. The sensor had a 20-degree field of view and was orientated to point straight down below the satellite to the Earth's surface. Using a Coverage Definition object, you created a global coverage grid and assigned the Sensor object as the coverage asset. Using a Figure Of Merit object with a Simple Coverage definition, you determined that approximately 49 percent of the Earth's surface was accessed by them sensor object during a 24-hour analysis period. The satellite's original inclination was 116 degrees and its eccentricity was 0.363. Using the STK Plugin for ModelCenter and the ModelCenter software, you ran two Parametric studies, changing inclination and eccentricity, and studied the effects on global coverage. You ended the analysis by performed a Carpet Plot study, which determined the best combination of inclination and eccentricity that provided the highest percentage of global coverage. Your final value for the inclination was 119.6 degrees and the eccentricity was 0.361. This combination raised global coverage percentage from the benchmark of approximately 49 percent to approximately 52 percent.

Part 11: Introduction to Communications

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators require a fast and easy way to set up, analyze, and optimize communication systems prior to employing them in the field. They want to simulate transmitters and receivers for a link budget analysis.

Solution

Use the STK® *Communications* capability and the host of analysis tools it provides to simulate how transmitters and receivers work in the field.

Using the starter scenario (*.vdf file)

To speed things up and enable you to focus on this lesson's main goal, you will use a partially created scenario. The partially created scenario is saved as a visual data file (VDF) in your STK install.

Retrieving the starter scenario

1. Launch the STK ( ) application.
2. Click Open a Scenario in the Welcome to STK dialog box.
3. Go to <Install Dir>\Data\Resources\stktraining\VDFs.
4. Select STK_Communications.vdf.
5. Click Open .

Visual data files versus Scenario files

You must make sure that you save your work in the STK application as a scenario file (.sc) and not a visual data file (.vdf) by selecting Save As from the STK File menu. A VDF is a compressed version of an STK scenario, which makes them great for sending your work in the STK application to others. However, you should use a scenario file while working with the STK application on your machine.

If you open a VDF file, the STK application keeps it as a VDF and does not automatically convert it to a scenario file. This means that the STK application does not change the file type of your scenario when you launch your scenario. You need to convert the VDF to a Scenario file using Save As.

Saving a VDF file as a Scenario file

Use Save As from the STK File menu to convert the VDF file that you opened into a scenario file.

1. Select Save As... in the File menu.
2. Select the STK User folder in the navigation pane.
3. Right-click in the file and folder browser.
4. Select New > Folder in the shortcut menu.
5. Rename New Folder to match the title of the scenario.
6. Open the folder you just created.
7. Enter the name of the folder into the **File name** field. This will be the Scenario object's name.
8. Open the **Save as type** drop-down menu.
9. Select Scenario Files (*.sc).
10. Click **Save** .

Analyzing a communications system

In this scenario, you will analyze communications between a geosynchronous satellite transmitter and a communications ground site receiver.

Selecting the relevant scenario objects

The starter scenario you loaded contains most of the objects required for your analysis. You will only use a portion of the available objects in the Object Browser in this tutorial, not all of them. There are extra objects because you can use this same scenario to complete other focused, feature-specific lessons about STK *Communications*.

1. Select the check box for the following objects in the Object Browser:
    - GEO_Sat_West ( )
    - Communication_Site ( )
2. Click Save ( ).

Understanding the Communications capability

The STK *Communications* capability simulates the performance of communications systems in the context of their missions. With *Communications*, you can model all the physical components of a system including the radio frequency (RF) environment, assess the impacts of a wide variety of conditions and interference, conduct comprehensive link budget analyses, create reports of the results, and visualize them with 2D and 3D graphics.

Modeling a Simple Transmitter

Simulate the geosynchronous satellite's transmitter using a Transmitter object employing a Simple Transmitter model. The Simple Transmitter model is convenient when you do not have all the information necessary to model the transmitter in detail (such as during the system engineering process). The Simple Transmitter model uses an isotropic, omnidirectional antenna, which is an ideal spherical pattern antenna with constant gain.

Using a Simple Transmitter model, you can set up the RF carrier frequency, the EIRP, and data rate of the transmitter.

- **EIRP** is the effective isotropic radiated power at the output of the transmit antenna. EIRP is expressed as the product of the power of the transmitting antenna and its gain.
- **Data rate** is a compound dimension with data bits and time as simple dimensions.

Inserting a Transmitter object

Insert a Transmitter ( ) object and attach it to GEO_Sat_West ( ).

1. Select Transmitter ( ) in the Insert STK Objects Tool.
2. Select Insert Default ( ) as the method.
3. Click Insert...
4. Select GEO_Sat_West ( ) in the Select Object dialog box.
5. Click OK .
6. Right-click on Transmitter1 ( ) in the Object Browser.
7. Select Rename.
8. Rename Transmitter1 ( ) to Downlink_Tx.

Configuring the Simple Transmitter model

Set up the RF carrier frequency, the EIRP, and data rate of the transmitter.

1. Right-click on Downlink_Tx ( ) in the Object Browser.
2. Select Properties ( ).
3. Select the Basic - Definition page when the Properties Browser opens.
4. Look at the Transmitter Model field. Simple Transmitter Model is the default model.
5. Select the Model Specs tab.
6. Enter the following options:

Option	Value
Frequency	5 GHz
EIRP	5 kW
Data Rate	1 Mb/sec

7. Click Apply to accept your changes and to keep the Properties Browser open.

Adding a modulator to the transmitter

Communications allows you to select from multiple modulators, including user-defined modulators. Bi-phase shift keying (BPSK) is the default modulator in the STK application.

Change that to quadrature phase shift keying (QPSK). QPSK converts digital bits into pairs; this decreases the data bit rate to half, which allows space for other users on the same channel.

1. Select the Modulator tab.
2. Open the Name drop-down list.
3. Select QPSK.
4. Click **OK** to accept your changes and to close the Properties Browser.

Modeling a Simple Receiver

Simulate the ground communications site's receiver using a Receiver object employing a Simple Receiver model. The Simple Receiver model is convenient when you do not have all the information necessary to model the receiver in detail (such as during the system engineering process).

The Simple Receiver model uses an isotropic antenna which you cannot change. Set G/T (gain divided by the system noise temperature in kelvins) which expresses the performance of an entire receiver system.

Inserting a Receiver object

Insert a Receiver ( ) object and attach it to Communication_Site ( ).

1. Insert a Receiver ( ) object using the Insert Default ( ) method.
2. Select Communication_Site ( ) in the Select Object dialog box.
3. Click **OK** .
4. Rename Receiver1 ( ) to Downlink_Rx.

Configuring the Simple Receiver model

Set the G/T to express the performance of the entire receiver system.

1. Open Downlink_Rx's ( ) Properties ( ).
2. Select the Basic - Definition page.
3. Look at the Receiver Model field. Simple Receiver Model is the default model.
4. Select the Model Specs tab.
5. Enter 6 dB/K in the G/T field.

   Notice that Auto Track is turned on. The Frequency Auto Track option allows a receiver to track and lock onto the transmitter's carrier frequency with which it is currently linking, including any Doppler shift.

Adding a demodulator to the receiver

Communications enables you to select from a number of demodulators, including user-defined demodulators. Recall that your transmitter is using QPSK.

1. Select the Demodulator tab.

   Notice that Auto-select Demodulator is turned on. If selected (which it is by default), the receiver automatically selects a demodulator that matches the modulation of the incoming signal. If this option is not selected, you must specify the type of demodulator. If the incoming signal's modulation does not match the modulation type of the selected demodulator, the STK application will set the Bit Error Rate (BER) to 0.5.

2. Click **OK** to accept your changes and to close the Properties Browser.

Creating a Simple Link Budget report

A link budget created in the Access tool is known as a Simple Link Budget. The Link Budget report is a specialized Access report for basic link budget analysis and is available using the Link Budget button in the Reports frame of the Access tool.

1. Right-click on Downlink_Rx ( ) in the Object Browser
2. Select Access... ( ) in the shortcut menu.
3. Expand (⊞) GEO_Sat_West ( ) in the Associated Objects list once the Access Tool opens.
4. Select Downlink_Tx ( ).
5. Click Compute .

6. Click **Link Budget...** in the Reports frame.
7. Take some time to look at the Simple Link Budget report.

Changes in data in columns such BER (Bit Error Rate) are likely caused by the satellite's inclination change which increases the range between the ground site and the satellite.

Summary

This was a quick introduction to *Communications*. You designed a preliminary system using Simple Transmitter and Receiver models and their inherited isotropic antennas. The system contained a very basic, one-way communications link between a geosynchronous satellite and a communications ground site. Based on your analysis, you determined that a viable communication link could be established between the satellite and the ground site.

Part 12: Introduction to Radar

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators need to determine how various radar settings will affect its ability to track different sized targets. They want to know how the following settings affect a radar's ability to track multiple target types:

- Radar Cross Section
- Pulse Repetition Frequency
- Gain
- Pulse integration

Solution

Use STK Pro and STK's *Radar* capability to:

- Create an airfield radar site
- Model an airport surveillance radar
- Build a monostatic radar
- Test various settings against multiple targets
- Determine probability of detection

Creating a new scenario

Create a new scenario.

1. Launch STK ( ).
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following in the New Scenario Wizard:

Option	Value
Name	STK_Radar
Location	Default
Start	Default day, month and year. Set the time to 03:00:00.000 UTCG
Stop	+30 min

4. When finished, click OK .
5. When the scenario loads, click Save ( ). A folder with the same name as your scenario is created for you in the location specified above.
6. Verify the scenario name and location in the Save As window.
7. Click Save .

Turning off Terrain Server

This is an introduction to *Radar*. Terrain will not be used in this analysis.

1. Right click on STK_Radar ( ) in the Object Browser.
2. Select Properties ( ) in the shortcut menu.
3. Select the Basic - Terrain page when the Properties Browser opens.
4. Clear the Use terrain server for analysis check box.
5. Click OK to accept the changes and close the Properties Browser.

Turning on Label Declutter

Turn on Label Declutter to reposition object labels so they do not obstruct one another while in close proximity.

1. Bring the 3D Graphics window to the front.
2. Click Properties ( ) in the 3D Window Defaults toolbar.
3. Select the Details page when the Properties Browser opens.
4. Select the Enable check box in the Label Declutter panel.
5. Click OK to accept changes and close the Properties Browser.

Inserting the target aircraft

Insert an Aircraft ( ) object. We will use the aircraft to analyze the airfield surveillance radar.

1. Select Aircraft ( ) in the Insert STK Objects tool.
2. Select the Insert Default ( ) method.
3. Click Insert... .
4. Right-click on Aircraft1 ( ) in the Object Browser.
5. Select Rename in the shortcut menu.
6. Rename the Aircraft1 ( ) to Target_Acft.

Creating the target aircraft's flight route

Create Target_Acft's ( ) route, then modify it's altitude and speed.

1. Open Target_Acft's ( ) properties ( ).
2. Select the Basic - Route page when the Properties Browser opens.
3. Click Insert Point two times.
4. Set the following:

Waypoint	Latitude	Longitude
One	37 deg	139.7 deg
Two	34 deg	139.1 deg

5. Click **Set All...** .
6. Select the Altitude: and Speed: check boxes in the Set All Grid Values dialog box.
7. Set the following:

Option	Value
Altitude	25000 ft
Speed	330 mi/hr

8. Click **OK** to close the Set All Grid Values dialog box.
9. Click **Apply** to accept the changes and keep the Properties Browser open.

Changing the target aircraft's 3D model

You can select a realistic 3D model for your Aircraft ( ) object.

1. Select the 3D Graphics - Model page.
2. Click the Model File: ellipsis ( ).
3. Select c-130_hercules.glb in the File dialog box.
4. Click **Open** .
5. Click **Apply** to accept the changes and keep the Properties Browser open.

Specifying the radar cross section

Before setting up and constraining a radar system, *Radar* allows you to specify an important property of a potential radar target - its radar cross section (RCS). Use the RCS of a popular four-engined turboprop transport aircraft.

1. Select the RF - Radar Cross Section page.
2. Clear the Inherit check box.

   This allows you to set the RCS settings for the Aircraft ( ) object instead of inheriting the settings from the Scenario ( ) object.
3. Enter 19 dBsm (decibels referenced to a square meter) in the Constant RCS Value: field.

Ideally, you would want to use an Aspect Dependent RCS file. Since you don't have one, you will use a constant value. The constant value we set is the RCS of a sphere that radiates isotropically.

4. Click **Apply** to accept the changes and keep the Properties Browser open.

Displaying radar cross section graphics

The 3D Graphics RCS page allows you to control the 3D display of Radar Cross Section contour lines.

1. Select the 3D Graphics - Radar Cross Section page.
2. Select the Show Volume check box in the Volume Graphics panel.
3. Click **OK** to accept the changes and close the Properties Browser.
4. Bring the 3D Graphics window to the front.
5. Right-click on Target_Acft ( ) in the Object Browser.
6. Select Zoom To in the shortcut menu.
7. Use your mouse to zoom out until you can see the RCS sphere.

Inserting the radar site

Use a Place ( ) object as the radar site location.

1. Insert a Place ( ) object using the Insert Default ( ) method.
2. Rename Place1 ( ) to Radar_Site.

Defining the radar site's location

Define the location of Radar_Site ( ) and raise its height above ground 50 ft to model the radar antenna height.

1. Open Radar_Site's ( ) properties ( ).
2. Select the Basic - Position page when the Properties Browser opens.

3. Set the following in the Position panel:

Option	Value
Latitude	35.75174 deg
Longitude	139.35621 deg
Height Above Ground	50 ft

4. Click OK to accept the changes and close the Properties Browser.

Raising the Place ( ) object 50 feet above the ground simulates the height of the radar antenna.

5. Bring the 3D Graphics window to the front.
6. Zoom To Radar_Site ( ).
7. Use your mouse to obtain situational awareness of the radar site's location.

Inserting the antenna servo system

Insert a Sensor ( ) object to simulate a servo system for antenna positioning. In STK, you could create a spinning sensor to simulate a spinning radar antenna normally seen at an airfield. However, you will lock the sensor onto the aircraft and constrain the sensor to point in a limited area. This simulates the actual field of view of the airfield surveillance radar both horizontally and vertically.

1. Insert a Sensor ( ) object using the Insert Default ( ) method.
2. Select Radar_Site ( ) in the Select Object dialog box.
3. Click OK .
4. Rename Sensor1 ( ) to Servo_System.

Defining the sensor field of view

Define Servo_System's ( ) field of view using a Simple Conic sensor pattern. You will use the sensor's field of view for situational awareness when Servo_System( ) points the antenna at Target_Acft ( ).

1. Open Servo_System's ( ) properties ( ).
2. Select the Basic - Definition page when the Properties Browser opens.
3. Enter 2 deg in the Cone Half Angle: field in the Simple Conic panel.
4. Click **Apply** to accept the changes and keep the Properties Browser open.

Targeting the aircraft

Use the Targeted pointing type to point Servo_System ( ) to Target_Acft( ).

1. Select the Basic - Pointing page.
2. Open the Pointing Type: shortcut menu.
3. Select Targeted.
4. Move ( ) Target_Acft ( ) from the Available Targets list to the Assigned Targets list.
5. Click **Apply** to accept the changes and keep the Properties Browser open.

Setting range and elevation angle constraints

There are many types of radar systems. A typical airport surveillance radar's nominal range is 60 miles and the elevation angle of the beam can track from 0 to 30 degrees. Anything higher than 30 degrees is the cone of silence in which the radar cannot track the aircraft. Extend the Servo_System's ( ) maximum range further than 60 miles in order to lock onto the aircraft when it's above the horizon.

Adding the range and elevation constraints

Add range and elevation angle to the Active Constraints list.

1. Select the Constraints - Active page.
2. Click Add new constraints ( ) in the Active Constraints toolbar.
3. Using the Ctrl key on your keyboard, select the following in the Constraint Name list in the Select Constraints to Add dialog box:

- Elevation Angle
- Range

4. Click **Add** .
5. Click **Close** to close the Select Constraints to Add dialog box.

Setting the max values

Set the max values for range and elevation angle constraints.

1. Select Elevation Angle in the Active Constraints list.
2. Select the Max: check box in the Constraint Properties - Elevation Angle panel.
3. Enter 30 deg in the Max: field.
4. Select Range in the Active Constraints list.
5. Select the Max: check box in the Constraint Properties - Range panel.
6. Enter 150 km in the Max: field.
7. Click **OK** to accept the changes and close the Properties Browser.

Generating an azimuth elevation range report

Generate an azimuth-elevation-range (AER) report to see what affect your constraints have on your accesses.

1. Right click on Servo_System ( ) in the Object Browser.
2. Select Access... ( ) in the shortcut menu.
3. Select Target_Acft ( ) in the Associated Objects list when the Access Tool opens.
4. Click Compute .
5. Click **AER...** in the Reports panel.
6. Look at the Elevation (deg) column.
7. Notice that the first access ends and the second access begins at an approximate elevation angle of 30 degrees.

   There is a break in access when the elevation angle exceeds 30 degrees due to the modeled cone of silence.

8. Close the AER report. and the Access Tool when finished.

Looking at the sensor's field of view

Animate through the scenario to get a visual idea of when Servo_System ( ) tracks Target_Acft ( ).

1. Bring the 3D Graphics window to the front.
2. Click Reset ( ) in the Animation Toolbar.
3. Right-click on Radar_Site ( ).
4. Select Zoom To.
5. Use your mouse to zoom out until you can see the entire aircraft flight route, the radar site, and the sensor's field of view.
6. Click Decrease Time Step ( ) in the Animation Toolbar until Time Step: is 1.00 sec.
7. Click Start ( ) in the Animation Toolbar to animate the scenario.
8. Watch the animation. You can see the sensor turn off when the elevation angle exceeds 30 degrees, and turn back on when it returns to 30 degrees.
9. Click Reset ( ) in the Animation Toolbar when finished.

Inserting an airport surveillance radar

Insert a *Radar* ( ) object to create an airport surveillance radar. We will model actual airport surveillance radar specifications that are easily available to the public.

1. Insert a Radar ( ) object using the Insert Default ( ) method.
2. Select Servo_System ( ) in the Select Object dialog box.
3. Click OK .
4. Rename Radar1 ( ) to ASR.

Modeling a monostatic radar

Model a **Monostatic** radar with a **Search/Track** mode. This will model a common antenna for both transmitting and receiving, and detect and track point targets.

1. Open ASR's ( ) properties ( ).
2. Select the Basic - Definition page when the Properties Browser opens.
3. Notice that Radar System defaults to Monostatic.
4. Select the Mode tab.
5. Notice that Radar Monostatic Mode defaults to Search Track.

Defining the waveform

The waveform in your system will use a fixed pulse repetition frequency (PRF), with a PRF of ~1000 Hz. Radar systems often use **multiple pulse integration** to increase the signal-to-noise ratio. The PRF is the number of pulses of a repeating signal in a specific time unit. After producing a brief transmission pulse, the transmitter is turned off in order for the receiver to hear the reflections of that signal off of targets.

1. Select the Waveform sub-tab.
2. Notice that Waveform defaults to Fixed PRF.
3. Select the Pulse Definition sub-sub-tab.
4. Notice that the PRF option is selected and the default value is 0.001 MHz.
5. Keep that value since your system's PRF is ~1000 Hz.

Defining the pulse width

Pulse width is the width of the transmitted pulse (the uncompressed RF bandwidth can also be taken as the inverse of the pulse width). Set the pulse width to one microsecond.

1. Open the Pulse Width shortcut menu ( ).
2. Select usec.

3. Enter 1 usec in the Pulse Width field.
4. Click Apply to accept the changes and keep the Properties Browser open.

Defining the antenna model

Model the antenna using the **cosine squared aperture rectangular antenna** pattern. The antenna transmit frequency for this radar is between 2.7-2.9 GHz.

1. Select the Antenna tab.
2. Select the Model Specs sub-tab.
3. Click the Antenna Model Component Selector ().
4. Select Cosine Squared Aperture Rectangular () in the Antenna Models list when the Select Component dialog box opens.
5. Click OK to close the Select Component dialog box.
6. Select the Use Beamwidth option.
7. Set the following:

Option	Value
X Dim Beamwidth	5 deg
Y Dim Beamwidth	1.4 deg
Design Frequency	2.8 GHz
Main-lobe Gain	34 dB
Efficiency	55 %

8. Click Apply to accept the changes and keep the Properties Browser open.

Defining the radar transmitter

The transmitter has a frequency range of 2.7-2.9 GHz, a peak power of 20 kW.

1. Select the Transmitter tab.
2. Select the Frequency option.

3. Enter 2.8 GHz in the Frequency field.
4. Enter 20 kW in the Power: field.
5. Click **Apply** to accept the changes and keep the Properties Browser open.

Setting the polarization

An ASR system can use linear or circular polarization. You will model linear polarization.

1. Select the Polarization sub-tab.
2. Select the Use check box.
3. Keep the default setting of Linear.
4. Click **Apply** to accept the changes and keep the Properties Browser open.

Setting the radar receiver's polarization

You don't have specific values regarding the low noise amplifier settings. These would be applied on the Receiver's Specs sub-tab. However, you know the polarization and want to add the receiver's system noise temperature. Let's set the polarization model type to Linear now.

1. Select the Receiver tab.
2. Select the Polarization sub-tab.
3. Select the Use check box
4. Keep the default setting of Linear.
5. Click **Apply** to accept the changes and keep the Properties Browser open.

Adding the radar receiver's system noise temperature

Next, add the receiver's system noise temperature to your analysis. You will compute system noise temperature using the default values, and take into account Sun and Cosmic Background noise.

1. Select the System Noise Temperature sub-tab.
2. Select the Compute option.

3. Select the Compute option in the Antenna Noise panel.
4. Select the Sun check box.
5. Select the Cosmic Background check box.
6. Click **OK** to accept the changes and close the Properties Browser.
7. Save ( ) your scenario.

Determining probability of detection

You will base the probability of detection (Pdet) on a value of 0.800000 or higher, one (1) being the highest value. You will also look at signal-to-noise ratio (SNR) and pulse integration. You will start by determining the Pdet of the large turboprop aircraft. Then, you will change Target_Acft's ( ) constant RCS value to simulate a medium-sized aircraft, then a small-sized aircraft, and then a bird. Finally, you'll load a notional Aspect Dependent RCS file to see the difference between that and the constant value RCS sphere.

Computing access

Compute access between ASR ( ) and Target_Acft ( ).

1. Right-click on ASR ( ) in the Object Browser.
2. Select Access... ( ) in the shortcut menu.
3. Select Target_Acft ( ) in the Associated Objects list when the Access Tool opens.
4. Click Compute .

Generating a Radar SearchTrack report

Now that you calculated access between ASR ( ) and Target_Acft ( ), generate a Radar SearchTrack report.

1. Click **Report & Graph Manager...** .
2. Select the Radar SearchTrack report () when the Report & Graph Manager opens.
3. Click **Generate...** .
4. Click **Show Step Value** when the report opens.
5. Enter 30 sec in the Step: field.
6. Press Enter on your keyboard.

Understanding the data

The content of a report or graph is generated from the selected data providers for the report or graph style. The data provider you'll focus on in this analysis is Radar SearchTrack.

Observing Pdet

Look at the difference between S/T Pdet1 and S/T Integrated Pdet in the report. S/T Pdet1 is based off of a single pulse. S/T Integrated PDet uses multiple pulses.

1. Look at the first line in the report.
2. Locate the two columns S/T Pdet1 and S/T Integrated Pdet.
3. Note the difference in the values.

   Pulse integration improves the ability of the radar to detect targets by combining the returns from multiple pulses. You can see this in the S/T Pulses Integrated column in the report.
4. Notice that overall tracking is good when using pulse integration (Pdet of 0.8 or higher).
5. Keep the report open.

Observing SNR

Look at the difference between S/T SNR1 (dB) and S/T Integrated SNR (dB) in the report. S/T SNR1 (dB) is based on a single pulse and S/T Integrated SNR (dB) on pulse integration.

1. Locate the two columns S/T SNR1 (dB) and S/T Integrated SNR (dB).
2. Note the differences in the values.
3. Again, the pulse integration allows for a better SNR.

Simulating a medium-sized aircraft

Next, simulate a medium-sized aircraft.

1. Open Target_Acft's ( ) properties ( ).
2. Select the RF - Radar Cross Section page.
3. Enter 10 dBsm in the Constant RCS Value: field.
4. Click **Apply** to accept the changes and keep the Properties Browser open.
5. Return to the Radar SearchTrack report.
6. Click Refresh (F5) ( ) in the report's toolbar.
7. Note the S/T Pdet1, S/T Integrated Pdet, S/T SNR1 (dB), and S/T Integrated SNR (dB) changes.

   The radar's ability to track this aircraft has diminished due to the aircraft's smaller RCS.

Simulating a small-sized aircraft

Simulate a small sized aircraft.

1. Return to Target_Acft's ( ) properties ( )..
2. Enter zero (0) dBsm in the Constant RCS Value: field.
3. Click **Apply** to accept the changes and keep the Properties Browser open.
4. Return to the Radar SearchTrack report.
5. Click Refresh (F5) ( ) in the report's toolbar.
6. Note the S/T Pdet1, S/T Integrated Pdet, S/T SNR1 (dB), and S/T Integrated SNR (dB) changes.

   The radar's ability to track this aircraft has again diminished due to the aircraft's smaller RCS.

Simulating birds and stealth

Simulate a bird and a large, somewhat stealthy aircraft.

1. Return to Target_Acft's ( ) properties ( ).
2. Enter -20 dBsm in the Constant RCS Value: field.
3. Click **Apply** to accept the changes and keep the Properties Browser open.
4. Return to the Radar SearchTrack report.
5. Click Refresh (F5) ( ) in the report's toolbar.
6. Note the S/T Pdet1, S/T Integrated Pdet, S/T SNR1 (dB), and S/T Integrated SNR (dB) changes.

   Looking at the results, your system might need a different frequency or more power.

Using Aspect Dependent RCS files

If you have an Aspect Dependent RCS file built for a specific target aircraft, your data will be much more realistic.

Loading an external file

Load an installed Aspect Dependent RCS file.

1. Return to Target_Acft's ( ) properties ( ).
2. Open the Compute Type: shortcut menu.
3. Select External File.
4. Click the Filename: ellipsis ( ).
5. Browse to <STK install folder>\Data\Resources\stktraining\samples\SeaRangeResources\X-47B
6. Select X-47B_Notional_Sample.rcs in the Select File dialog box.
7. Click **Open** .

8. Click **Reload** .
9. Click **OK** to accept the changes and close the Properties Browser.

Visualizing the RCS pattern

View the RCS pattern in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom To Target_Acft ( ).
3. Use your mouse to get a good view of the aspect dependent RCS pattern.

Viewing the data

Refresh the Radar SearchTrack report to see the changes in SNR, PDet and Pulse Integration.

1. Return to the Radar SearchTrack report.
2. Click Refresh (F5) ( ) in the report's toolbar.
3. Note the S/T Pdet1, S/T Integrated Pdet, S/T SNR1 (dB), and S/T Integrated SNR (dB) changes.

   Depending on the reflection from the aircraft back to the radar, you could see fluctuation in your values. This is noticeable in the S/T Pulses Integrated column.

Viewing RCS data in a graph

Use the RCS graph style to visualize changes to RCS Decibel (dBsm). Note the cone of silence in the middle of the graph.

1. Return to the Report & Graph Manager.
2. Ensure the Object Type: is set to Access.
3. Select Place-Radar_Site-Sensor-Servo_System-Radar-ASR-To-Aircraft-Target_Acft in the Object Type: list.
4. Select the Radar RCS graph ( ) in the Installed Styles folder.
5. Click **Generate...** .

6. Click Show Step Value .Show Step Value At the top of the graph.
7. Change the Step: value to 1 sec.
8. Press Enter on the keyboard or click Refresh ( ) at the top of the report.
9. Save ( ) your scenario.

Summary

You created a scenario that used the surface of the WGS84 as the central body obstruction. You created a simple flight route of an aircraft and changed its RCS value to simulate a large, four-engined turboprop using a constant analytical RCS value. You created an airfield radar site and inserted a Sensor to create a servo system that was used to steer a radar antenna pattern inside its field of view in order to analyze various targets. You built a Radar using specifications typically found on air surveillance radars. You analyzed Pdet values for large, medium, small, and very small targets focusing on Pdet, SNR, and Pulse Integration. Finally, you used a notional aspect dependent RCS file that demonstrated both analytical and visual differences when compared to a constant RCS sphere.

Part 13: Integrating STK with MATLAB

Note: Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Important: Required Software Install: The MATLAB programming platform is required to complete this tutorial. Version R2024b was used in this tutorial. Your version may have different tools and procedures.

Problem statement

Developers, engineers and operators need to do the following tasks quickly and easily:

- Automate repetitive tasks from outside of the STK application
- Integrate the STK software with other applications, such as MATLAB, Microsoft Excel, and Python
- Develop custom applications
- Leverage the AGI Object Model collection of COM libraries from other applications

You want to be able to create, drive, and extract data from the STK application using MATLAB. To do so, you need to understand how to access the STK Object Model and Connect Application Programming Interfaces (APIs). You also need a basic understanding of the structure and contents of the STK Programming Help.

Solution

Use the STK software's *Integration* capability and the MATLAB programming platform to create a new instance of the STK application through the COM interface and expose the object root. Next, build a new scenario, populating it with a Satellite object and a Target object. Create an access between the Satellite object and the Target object, extracting access times from the STK software and moving them into MATLAB. Use the correct data providers, both groups and elements, to determine a Precision Pass value at a specific time in the scenario. Learn the benefits of the STK Programming Help.

Using the STK Programming Help

The STK Programming Help offers a wide variety of options to automate and customize STK and to integrate its technology into other applications. This tutorial covers only a small portion of the STK Programming Help, but you get a great starting point from which to branch out. You can search and browse the STK Programming Help as you would the STK Desktop Help. See the Using the Help page for more information.

1. Open your preferred internet browser.
2. Go to **help.agi.com**.
3. Click STK Programming Help.

Integrating with other applications

The STK software's *Integration* capability enables you to automate repetitive tasks from outside the STK application, integrate the STK software with other applications such as MATLAB and Microsoft Excel, develop custom applications, and leverage the AGI Object Model collection of COM libraries from other applications. Follow these steps to learn more about these possibilities.

1. Select the Select the Right Technology Help topic in the table of contents of the STK Programming Help.
2. Select the Integrate Technologies Help topic.
3. Click the clickable decision tree link on the Integrate With Other Applications page.
4. Take some time to view the page and click some of the icons.

There are many clickable decision trees available in STK Programming Help. You can use the trees to quickly locate help pages that you need to be successful when integrating with the STK software.

Automating repetitive tasks

To automate repetitive tasks in the STK application, you can use HTML, Connect, and the STK Object Model to build tools that are accessible from within the STK application. You can find more information on this in the STK Programming Help by following these steps:

1. Under the Select the Right Technology help topic, select Automate Tasks.
2. Click the clickable decision tree link on the Automate Repetitive Tasks topic page.
3. Take some time to view the page and click some of the icons.

Extending AGI products

AGI provides a variety of ways to extend its products. These extensibility mechanisms come under two distinct categories: user interface extensibility and engine extensibility. You can find more information on these in the STK Programming Help by following these steps:

1. Under Select the Right Technology, click Extend AGI Products.
2. Click the clickable decision tree link on the Extend AGI Products Using Plugins and Custom User Interfaces topic page.
3. Take some time to view the page and click some of the icons.

Visualizing with the STK software

STK X enables developers to add advanced 2D and 3D visualization with the STK software's and analytical capabilities to applications with little effort. You can find more information on this in the STK Programming Help by following these steps:

1. In the table of contents, go to Library Reference > Controls > STK X.
2. Take some time to familiarize yourself with the contents of this page.

Using core libraries

The STK software API consists of two major subsystems: Connect and the STK Object Model.

1. In the STK Programming Help, select the Using Core Libraries Help topic in the table of contents.
2. Note the advantages and disadvantages of using the Connect and Object Model libraries.

Choosing between Connect or the STK Object Model should be driven by the requirements of the application and your needs as a developer. However, this need not be an exclusive choice. You can use Connect commands in a COM application side by side with Object Model code.

Learning STK Object Model naming conventions

The STK Object Model contains naming conventions for classes, interfaces, and enumerations.

1. In the STK Programming Help, go to Library Reference > STK Object Model > Naming Conventions.
2. Note the various naming conventions described in the Naming Conventions Used in the Various Core Libraries topic.
3. Click the certain abbreviations link.
4. Note the abbreviations most commonly used in the names of classes, interfaces, and other types in the STK Object Model.

Using model diagrams

As a serious programmer, you can use diagrams to see how the STK traffic flow is connected. That way, you know what's to the right, left, above, and below, all while deep linking to the individual IAg interface topic pages.

1. In the STK Programming Help, go to Using Core Libraries > STK Object Model > STK Objects > Diagrams.
2. Click STK Object Model Diagram (PDF) in the STK Objects Object Model Diagrams topic.
3. Zoom in closer to the page for clarification.
4. Take some time to look at the STK 10.0 Object Model Diagram PDF.
5. Close the STK Object Model Diagram (PDF) when finished.
6. Keep the STK Programming Help open.

Connecting the STK software and MATLAB

You can integrate the STK software and MATLAB to automate, extend or visualize your analysis. There are two interface methods to connect the STK software with MATLAB: by using connectors that can be downloaded from AGI's website or by accessing the STK software's COM interface.

For each supported version of MATLAB, AGI produces a connector, which communicates over TCP/IP, which you can download from the (**https://support.agi.com/downloads**). An STK MATLAB Connector will allow you to use MATLAB plugin scripts, the AeroToolbox, and the MexConnect interface with the STK software.

> **Note:** If you do not have an agi.com account, you will need to create one. The user approval process can take up to three (3) business days. Please contact **support@agi.com** if you need access sooner.

You can also connect any 64- and 32-bit version of MATLAB to the STK software using the COM interface. Only the COM interface can be used to access the Object Model API.

For this tutorial, you will connect the MATLAB and STK applications via the COM interface.

Launching MATLAB

MATLAB is a programming and numeric computing platform used by millions of engineers and scientists to analyze data, develop algorithms, and create models.

1. Launch MATLAB.
2. Select the Home tab when MATLAB opens.

3. Click Layout in the MATLAB Toolstrip.

4. Select Default in the Select Layout menu. This makes it easier to follow the tutorial. If you're familiar with MATLAB and have a setup you prefer, keep it.

Creating a new MATLAB script

MATLAB script files are program files with the .m file extension. In these files, you can write groups of commands that you want to execute together. Scripts are the simplest kind of program file because they have no input or output arguments; they operate only on the available data in their workspace.

1. Select the Home tab in the MATLAB Toolstrip.
2. Click **New Script** .
3. Select the Editor tab in the MATLAB Toolstrip.
4. Select Save As.... in the Save menu.
5. Browse to the folder where you will save the .m file (e.g., C:\Users\<username>\Documents\MATLAB).
6. Enter STK_Matlab in the File name field.
7. Ensure Save as type is set to MATLAB Code files (UTF-8) (*.m).
8. Click **Save** .

> **Note:** After creating the script file for this tutorial, you can practice with it later. A completed version of the script is included with your STK install at **<Install Dir>\Data\Resources\stktraining\scripts\STK_MATLAB_Script.m**.

Locating the STK Object Model and MATLAB code snippets

Code snippets demonstrate tasks that are commonly encountered when working with the STK Object Model.

1. Return to the STK Programming Help.
2. Go to Using Core Libraries > STK Object Model > MATLAB Code Snippets.

Task 1: Creating an STK instance

The AgStkObjectRoot Object is the top-level object in the STK Object Model hierarchy. The AgStkObjectRoot object provides methods and properties to load scenarios, create new ones, and access the Object Model Unit preferences.

Writing comments in your script

Write comments by preceding the line with the % symbol.

1. In the MATLAB Editor, enter or copy and paste the following comment onto line 1 of your MATLAB script:

```
%Task1
```

2. Select the Enter key to move to the next line.

Referencing the IAgStkObjectRoot interface

From the IAgStkObjectRoot interface, you can access all other objects and collections exposed by the Object Model.

1. Locate the Initialization section under the subheading "How do I ..." on the MATLAB Code Snippets topic page.
2. Click the Start STK and get a reference to IAgStkObjectRoot link.
3. Enter or copy and paste the following comment and code onto lines 3 through 5 of your MATLAB script:

```
% Create an instance of STK
uiApplication = actxserver('STK13.Application');
uiApplication.Visible = 1;
```

4. Select the Enter key.
5. Place your cursor before `%Task1`.
6. Click **Section Break** on the MATLAB Toolstrip.
7. Place your cursor before `%Task1`.
8. Click **Run and Advance** in the MATLAB Toolstrip.
9. Look at the MATLAB Workspace.

You created a variable called uiApplication and started an instance of the STK application. You also made the STK application splash screen visible.

> **Note:** Save your MATLAB script after completing each section's task.

Selecting a root of the Application Model

The second part of the Start STK and get a reference to IAgStkObjectRoot snippet shows an example of how to expose the STK object root. Manually enter this information in the MATLAB Command Window for familiarization.

1. Click into the MATLAB Command Window.
2. Copy and paste or enter the following code in the Command Window:

   ```
   root = uiApplication.
   ```

   Make sure to include the period (.).
3. MATLAB can work with the STK software using IntelliSense, which will show you possible selections. Use IntelliSense to select an application root of the STK Object Model.
4. Return to the STK Programming Help.
5. Enter or paste IAgUiApplication in the search field located in the upper-right corner. The case doesn't matter.
6. Select the Enter key.
7. Select AgUiApplicationLib~IAgUiApplication in the search results. The IAgUiApplication interface represents a root of the Application Model.

Exposing the STK object root

Looking at the IAgUiApplication Interface topic page, most of what you see in IntelliSense matches the Public Methods and Properties.

1. Scroll down the IAgUiApplication Interface help topic page to the Public Properties section.
2. Select Personality2, which returns a new instance of the root object of the STK Object Model.

   You could have searched for any of the IntelliSense selections. Searching for Personality2 would have taken you directly to the help page and its description.
3. Return to the Command Window.
4. Select the Tab key while the cursor is just after the period (.) of the `root = uiApplication.` command.

5. Select `Personality2` in IntelliSense.
6. Select the Enter key.

   The Command Window will display the result, which should be:

   ```
   root =

           Interface.AGI_STK_Objects_13__IAgStkObjectRoot
   ```

7. Enter a semicolon (;) after `Personality2`.
8. Select the Enter key.

   Note that with the with the addition of the semicolon in the Command Window, the display of the output of the root = uiApplication.Personality2 command is suppressed.

9. Copy or enter the following whole line of code on line 6 of your MATLAB script:

   ```
   root = uiApplication.Personality2;
   ```

You've exposed the STK object root, so now you can start creating a new scenario.

Task 2: Scripting for a new scenario

You will now use your MATLAB script to interact with the STK application.

Creating a new scenario

Now that you've launched the STK application via the MATLAB interface, you can create a new scenario, set the scenario time period, and reset the animation time.

1. Enter or copy and paste the following section break and comment on lines 7 and 8 of the STK_Matlab.m script:

   ```
   %%
   %Task2
   ```

2. Select the Enter key.

   This adds a section break and the task identifier at the same time. You will do this for all subsequent tasks.

3. Return to MATLAB Code Snippets under Using Core Libraries > STK Object Model in the STK Programming Help.
4. Locate the Scenario Management section (under the Scenario heading) of the MATLAB Code Snippets topic page.

5. Click the Create a new Scenario link.

6. In MATLAB, enter or copy and paste the following comment and code onto lines 9 and 10 of your MATLAB script:

```
% IAgStkObjectRoot root: STK Object Model Root
root.NewScenario('Example_Scenario');
```

7. Select the Enter key.

Using the IAgStkObjectRoot interface

The IAgStkObjectRoot interface represents the automation interface supported by the root object of the Automation Object Model.

1. Return to the STK Programming Help.
2. Enter or copy and paste "IAgStkObjectRoot" in the Search field.
3. Select the Enter key.
4. Select STKObjects~IAgStkObjectRoot in the search results.
5. Select NewScenario in the Public Methods list.

Creating a new scenario

The NewScenario method creates a new scenario.

1. There is no syntax specifically for MATLAB, so look at the syntax for [Visual Basic .NET].
2. Click the ScenarioName link in the Visual Basic .NET syntax entry.

The NewScenario method expects a string that is the scenario name. In the line you added to Task 2, you set the scenario name to Example_Scenario.

Setting the analysis start and stop times

You want to return a scenario object and set the analysis period.

1. Return to MATLAB Code Snippets.
2. Locate the SCENARIO MANAGEMENT section of the MATLAB Code Snippets topic page.
3. Click the Change scenario time period link.
4. Enter or copy and paste the following comment and code onto lines 11 and 12 of your MATLAB script:

```
% IAgStkObjectRoot root: STK Object Model Root
root.CurrentScenario.SetTimePeriod('Today', 'Tomorrow');
```

5. Select the Enter key.

   - The CurrentScenario property (IAgStkObjectRoot) returns a scenario object or a null value if you have not loaded a scenario yet.
   - The SetTimePeriod method (IAgScenario) sets the scenario time period. It expects a start time and stop time of the scenario period.

Resetting the scenario animation time

You want to reset the animation time to the beginning of the analysis period.

1. Return to MATLAB Code Snippets.
2. Locate the Scenario Management section of the MATLAB Code Snippets topic page.
3. Click the Reset the scenario time link.
4. Enter or copy and paste the following comment and code onto lines 13 and 14 of your MATLAB script

```
% IAgStkObjectRoot root: STK Object Model Root
root.Rewind;
```

5. Select the Enter key. You are using the Rewind method (IAgAnimation) to stop and reset the animation.
6. Place your cursor in front of `%Task2`.
7. Click **Run and Advance** in the MATLAB Toolstrip.
8. Go to the STK application window.

A new scenario named Example_Scenario () is open and the time has been reset in the Current Scenario Time field of the Animation toolbar.

Task 3: Inserting a Target object

To create a new object such as a Target object, use the AgESTKObjectType enumeration. Not every object in the STK software has an associated snippet, so you may need to edit the available code snippets to meet your needs.

Creating a new Target object

Target, Place, and Facility objects contain the same properties, but they're seen as different object classes. In this instance, since there isn't a snippet to insert a new Target object into the scenario, you can use the Create a facility (on the current scenario central body) snippet and make some simple changes to the code.

1. Return to your MATLAB script.
2. Enter or copy and paste the following section break and comment onto lines 15 and 16 of your MATLAB script:

```
%%
%Task3
```

3. Select the Enter key.
4. Return to MATLAB Code Snippets.
5. Locate the Facility section of the MATLAB Code Snippets topic page.
6. Click the Create a facility (on the current scenario central body) link.
7. Enter or copy and paste the following comment and code onto lines 17 and 18 of your MATLAB script:

```
% IAgStkObjectRoot root: STK Object Model Root
target = root.CurrentScenario.Children.New('eTarget', 'MyTarget');
```

8. Select the Enter key.

   The snippet code is: `facility = root.CurrentScenario.Children.New('eFacility', 'MyFacility');`. You are swapping out the variable eFacility for eTarget and changing the name to 'My Target'.

Retrieving the current scenario

The CurrentScenario property returns a Scenario object or a null value if no scenario has been loaded yet.

1. Return to the STK Programming Help.
2. Enter "CurrentScenario" in the Search field.
3. Select the Enter key.
4. Select STKObjects~IAgStkObjectRoot~CurrentScenario in the search results.
5. Go to the Visual Basic.NET syntax entry on the CurrentScenario Property (IAgStkObjectRoot) topic page.
6. Click the IAgStkObject link.

Retrieving children of an STK object

The IAgStkObject interface represents a specific instance of an STK object. The Children property (IAgStkObject) returns a collection of direct descendants of the current object.

1. Go to the Public Properties section on the IAgStkObject Interface page.
2. Select Children.
3. Go to Visual Basic.NET syntax in the Children Property (IAgStkObject) topic page.
4. Select IAgStkObjectCollection.

Retrieving a collection of STK objects

The IAgStkObjectCollection interface represents a collection of STK objects.

1. Go to Public Methods in the IAgStkObjectCollection Interface topic page.
2. Select New.

Creating a new STK object

The New method (IAgStkObjectCollection) creates an STK object using a specified class and instance name.

1. Go to the Visual Basic.NET syntax on the New Method (IAgStkObjectCollection) topic page.
2. Select AgESTKObjectType.

Understanding the AgESTKObjectType enumeration

The AgESTKObjectType Enumeration topic page shows STK objects (members) with their values and descriptions. When using the AgESTKObjectType enumeration to insert a new Target object, you can use the member name eTarget or the value 23. In this case, you are using eTarget and setting the Target object's name to MyTarget. This is another way to use the STK Programming Help.

On your own, you might have to take some time working your way through the pages to find what you're looking for.

Assigning the location

Use a MATLAB code snippet to assign the location of the Target at a specific location.

1. Return to MATLAB Code Snippets.
2. Locate FACILITY in the MATLAB Code Snippets page.
3. Click the Set the geodetic position of the facility link.
4. Enter or copy and paste the following comments and code onto lines 19 through 21 of your MATLAB script:

```
% IAgTarget Interface provides access to the properties and methods used in defining a
target object.
% Latitude, Longitude, Altitude
target.Position.AssignGeodetic(50, 100, 0)
```

5. Select the Enter key.
   - The Position property (IAgTarget) gets the position of the target.
   - The IAgPosition interface provides access to the position of the object.
   - The AssignGeodetic method (IAgLLAPosition) assigns the position using geodetic representation.
6. In the script, place your cursor in front of `%Task3`.
7. Click **Run and Advance** in the MATLAB Toolstrip.

Checking the location in the STK application

Confirm the location of your new Target object by viewing its properties in the STK application.

1. Bring the STK application window to the front to view MyTarget () in the 2D and 3D Graphics windows.
2. Right-click on MyTarget () in the Object Browser.
3. Select Properties ().
4. Note the Latitude, Longitude, and Altitude on the Basic - Position page.

   If you have an Internet connection and are using Terrain Server, the Altitude will have a value higher than 0 km. That's because you placed the Target object on the surface of the terrain. If you are not using the Terrain Server, the Target object will be on the surface of the WGS84 ellipsoid.
5. Click **Cancel** to close MyTarget's Properties ().

Task 4: Inserting a Satellite object

Use Connect commands to propagate a Satellite object in your scenario.

1. Return to your MATLAB script.
2. Enter or copy and paste the following section break and comment onto lines 22 and 23 of your MATLAB script:

```
%%
%Task4
```

3. Select the Enter key.
4. Return to MATLAB Code Snippets.
5. Locate the SATELLITE section on the MATLAB Code Snippets page.
6. Click the Create a satellite (on the current scenario central body) link.
7. Return to your MATLAB script.
8. Enter or copy and paste the following comment and code onto lines 24 and 25 of your MATLAB script:

```
% IAgStkObjectRoot root: STK Object Model Root
satellite = root.CurrentScenario.Children.New('eSatellite', 'MySatellite');
```

9. Select the Enter key.

Learning about Connect

The Connect module provides you with an easy way to connect with the STK software and work in a client-server environment. You can use the library shipped with Connect to build applications that communicate with the STK application. This library contains functions, constants, and other messaging capabilities that you can use to connect third-party applications to the STK software.

1. Return to the STK Programming Help.
2. Go to Using Core Libraries > Connect.

This page is a great place to learn about Connect. Take some time to become familiar with the content linked from this page.

Propagating your satellite via Connect

Just like you did while using the STK Object Model and MATLAB Code Snippets, you can use the MATLAB Connect Code Snippets help topic to become familiar with Connect.

1. Go to Using Core Libraries > Connect > Connect Code Snippets.
2. Locate the Satellite section of the MATLAB Connect Code Snippets topic page.
3. Click the Set initial state of satellite and propagate link.

4. Return to your MATLAB script.
5. Enter or copy and paste the following code onto lines 26 and 27 of your MATLAB script:

```
root.ExecuteCommand('Units_Set * Connect Distance "Kilometers"'); %Default Connect Units
are meters
root.ExecuteCommand('SetState */Satellite/MySatellite Classical TwoBody
"UseAnalysisStartTime" "UseAnalysisStopTime" 60 ICRF "UseAnalysisStartTime" 6678.14 0 28
180 0 180');
```

   There are several propagators in the STK software (e.g., Two-Body, J2 Perturbation, J4 Perturbation, SGP4, etc.). This command is using the Two-Body propagator.
6. Select the Enter key.
   - The ExecuteCommand method (IAgStkObjectRoot) executes a custom Connect action.
   - The Units_Set command sets units of measure. You're using Units_Set because Connect's distance unit defaults to meters and you require kilometers.

Learning about the SetState Connect command

Look at the Alphabetical Listing of Connect Commands to get information on the SetState command.

1. Return to the STK Programming Help.
2. Go to Library Reference > Connect Command Library.
3. Note the links in the Connect Command Listings topic page.
4. Click the Alphabetical Listing of Connect Command link.

   The Alphabetical listing includes all Connect commands, regardless of their groupings.
5. Click "S" at the top of the Alphabetical Listing page.
6. Locate and select the SetState Classical command.
7. Scroll through the SetState Classical page and note the following:
   - The syntax will help you understand the input values of your SetState command.
   - The description table explains each input and its associated unit.
   - In the Examples section, there are several examples that you can copy and edit for your particular scenario.

Using MATLAB to set the Satellite state

Update the SetState syntax to definite the to define the satellite's orbital parameters.

1. Return to your MATLAB script.
2. Using SetState syntax, update the following values in the SetState Connect command that you entered on line 27 of your MATLAB script.

Option	Value
SemiMajorAxis	7200
Eccentricity	0
Inclination	90
ArgOfPerigee	0
RAAN	0
MeanAnom	0

The SetState command in MATLAB should match the following:

```
root.ExecuteCommand('SetState */Satellite/MySatellite Classical TwoBody
"UseAnalysisStartTime" "UseAnalysisStopTime" 60 ICRF "UseAnalysisStartTime" 7200 0 90 0
0 0');
```

3. Place your cursor in front of `%Task4`.
4. Click **Run and Advance** in the MATLAB Toolstrip.
5. Bring the STK application to the front to view MySatellite in the 2D and 3D Graphics windows.
6. Open MySatellite's ( ) Properties ( ).
7. Note and match the property settings and values to your SetState Connect command.
8. Click **Cancel** when finished.

Task 5: Creating access intervals between MySatellite and MyTarget

Use the STK Object Model to create access intervals from the Satellite object the to Target object.

Entering the access commands into MATLAB

Use your MATLAB script to enter access computation commands.

1. Return to your MATLAB script.
2. Enter or copy and paste the following section break and comment onto lines 28 and 29 of your MATLAB script:

```
%%
%Task5
```

3. Select the Enter key.
4. Return to the STK Programming Help.
5. Go to Using Core Libraries > STK Object Model > MATLAB Code Snippets.
6. Locate the ACCESS section on the MATLAB Code Snippets topic page.
7. Select Compute an access between two STK Objects (using IAgStkObject interface).
8. Return to your MATLAB script.
9. Enter or copy and paste the following comments and code onto lines 30 through 33 of your MATLAB script:

```
% Get access by STK Object
access = satellite.GetAccessToObject(target);
% Compute access
access.ComputeAccess();
```

   - The GetAccessToObject method (IAgStkObject) returns an IAgStkAccess object. In this instance, you're creating an access from the Satellite object to the Target object.
   - The ComputeAccess method (IAgStkAccess) recomputes the access between two objects.

10. Select the Enter key.

Computing and checking the access intervals

Run the latest MATLAB code section and check the access calculation results in the STK application.

1. In the script, place your cursor in front of `%Task5`.
2. Click **Run and Advance** in the MATLAB Toolstrip.
3. Bring the STK application window to the front.

   You can see the highlighted satellite ground tracks indicating periods of access in the 2D Graphics window.

Task 6: Retrieving the access data

You created access intervals between the Satellite object and the Target object. An easy way to extract and see those times is to create an interval collection of access start and stop times.

Scripting access interval retrieval in MATLAB

Write a MATLAB script task to get the access interval times from the STK software.

1. Return to your MATLAB script.
2. Enter or copy and paste the following section break and comment onto lines 34 and 35 of your MATLAB script:

```
%%
%Task6
```

3. Select the Enter key.
4. Return to MATLAB Code Snippets.
5. Locate the ACCESS section of the MATLAB Code Snippets page.
6. Click the Compute and extract access interval times link.
7. Return to your MATLAB script.
8. Enter or copy and paste the following comments and code onto lines 36 through 40 of your MATLAB script:

```
% IAgStkAccess access: Access calculation
% Get and display the Computed Access Intervals
intervalCollection = access.ComputedAccessIntervalTimes;
% Set the intervals to use to the Computed Access Intervals
computedIntervals = intervalCollection.ToArray(0, -1);
```

   - The ComputedAccessIntervalTimes property (IAgStkAccess) returns a list of the computed access interval times.
   - The IntervalCollections property (IAgCrdnIntervalsVectorResult) is a collection of interval collections.
   - The ToArray method (IAgIntervalCollection) returns a two-dimensional array of intervals beginning at a given position and having a specified number of rows.
   - The SpecifyAccessIntervals method (IAgStkAccess) specifies a list of intervals to use for the access calculation.

9. Select the Enter key.

Retrieving the access interval times

Run Task 6 of your MATLAB script to retrieve the access inteval times.

1. In the script, place your cursor in front of `%Task6`.
2. Click **Run and Advance** in the MATLAB Toolstrip.
3. In the MATLAB Workspace, double-click computedIntervals.
4. When the computedIntervals cell opens, expand columns 1 and 2 so you can read the access data. Column 1 shows the start times of the accesses and column 2 shows the stop times. Your data will have different values.
5. When finished, close the cell.

Task 7: Investigating data providers

The STK software generates reports and graphs using selected data providers for the specific report or graph style. You can view data providers, both groups and individual elements, when using the Report & Graph Manager inside of the STK application. The STK Programming Help contains details of data provider groups, individual elements, and report and graph styles.

1. Enter or copy and paste the following section break and comment onto lines 41 and 42 of your MATLAB script:

```
%%
%Task7
```

2. Select the Enter key.
3. Return to the STK Programming Help.
4. Go to Library Reference > Data Providers Reference.

You can spend a lot of time looking at an object's data providers, both groups and elements on the Data Providers by Object topic page.

Extracting data providers

Your scenario is 24 hours long. The Satellite object data provider group named "Passes" contains extensive data about the satellite's orbit for each pass. The data provider group "Precision Passes" contains, for specific times, the following:

- Precision pass number: This number has two parts: the integer pass number for the given time and the fraction of the orbit from the ascending node for that time.
- Precision path number: This is the satellite's orbit path number, which will remain at the value of 1 until the ground track repeats, at which time the number will increase to 2, and so on.

Use MATLAB to extract these data.

1. Go to Using Core Libraries > STK Object Model > MATLAB Code Snippets in the STK Programming Help.
2. Locate the DATA PROVIDERS section on the MATLAB Code Snippets topic page.
3. Click the Getting Data for a Single Point in Time link.
4. Return to your MATLAB script.
5. Enter or copy and paste the following comments and code onto lines 43 through 48 of your MATLAB script:

```
% IAgStkObjectRoot root: STK Object Model root
% IAgSatellite satellite: Satellite object
% Change DateFormat dimension to epoch seconds to make the data easier to handle in
MATLAB
root.UnitPreferences.Item('DateFormat').SetCurrentUnit('EpSec');
satPassDP = satellite.DataProviders.Item('Precision Passes').ExecSingle(2600);
pass = cell2mat(satPassDP.DataSets.GetDataSetByName('Precision Pass Number').GetValues);
```

6. Select the Enter key.

Setting the date format and elements

Break down your code to understand what time data will be returned.

1. Look at the code from line 46: `root.UnitPreferences.Item('DateFormat').SetCurrentUnit ('EpSec');`.
   - The UnitPreferences property (IAgStkObjectRoot) provides access to the Global Unit table.
   - The Item property (IAgUnitPrefsDimCollection) returns IAgUnitPrefsDim, for which you give a dimension name or an index. The data provider is DateFormat.
   - The SetCurrentUnit method (IAgUnitPrefsDimCollection) sets unit preferences. You are using epoch seconds, or EpSec.
2. Return to the STK Programming Help.
3. Go to Using Core Libraries > STK Object Model > STK Util > Dimensions and Units.
4. Scroll down to DateFormat in the STK Object Model Unit Preferences page.
5. Look at Available Units.

   The Epoch Seconds element is EpSec.

Your scenario's default date format is Gregorian UTC (UTCG). As stated in the SetCurrentUnit code snippet, changing the DateFormat dimension to epoch seconds makes the data easier to handle in MATLAB. Since your scenario is 24 hours long, the STK software expresses the scenario end time as 86400 EpSec. Thus, a scenario UTCG time of 30 Jun 2023 08:11:49.699 will now appear as 15109.699 EpSec. This is easier to read and use in MATLAB.

Selecting satellite data providers

Review your code to understand what data will be returned.

1. Return to your MATLAB script.
2. Look at the code from line 47: `satPassDP = satellite.DataProviders.Item('Precision Passes').ExecSingle(2600);`.
    - The DataProviders property returns the object representing a list of available data providers for the object. In this case, it is a Satellite object.
    - The ExecSingle method computes the data given a specific time. SingleTime uses the DateFormat Dimension, which you set to EpSec.
3. Return to the STK Programming Help.
4. Go to Library Reference > Data Providers Reference.
5. Select the Satellite in the Data Providers by Object topic page.
6. Select Precision Passes in the Available Data Providers table.

Retrieving the Precision Pass data

Examine the code to understand how the data Precision Pass data will be formatted.

1. Return to your MATLAB script.
2. Look at the code from line 48: `pass = cell2mat(satPassDP.DataSets.GetDataSetByName ('Precision Pass Number').GetValues);`.
    - The cell2mat function in MATLAB converts a cell array into a numeric array.
    - The DataSets property (IAgDrResult) returns a collection of datasets.
    - The GetDataSetByName method (IAgDrDataSetCollection) returns the element for a given name, which in this case is the Precision Passes data provider element Precision Pass Number.
    - The GetValues method (IAgDrDataSet) retrieves an array of values of the elements in the dataset.
3. Place your cursor in front of `%Task7`.

4. Click **Run and Advance** in the MATLAB Toolstrip.
5. Look at the pass value (e.g., 1.4276) in the MATLAB Workspace.

Summary

You began by viewing a small portion of the STK Programming Help pages. Next, using MATLAB code snippets, you performed the following:

- Created a connection between MATLAB and the STK application
- Exposed the STK object root
- Created a new scenario
- Inserted and positioned a Target object
- Inserted and propagated a Satellite object
- Calculated accesses between the Satellite object and the Target object
- Extracted access data from the STK software into MATLAB
- Used data providers to changes time units from UTCG to EpSec
- Extracted a satellite precision pass value from the STK into MATLAB

Part 14: Integrating STK with Python

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Problem statement

Developer, engineers, and operators want to analyzing the behavior of a satellite when it has contact with a ground site. The task will be repetitive, they want to consider methods of automating the process and extracting the data. Knowing that you can integrate the STK software with other tools, you decide to explore the integration process.

Solution

Analysis in the STK application can be automated with code. Using the STK software's *Integration* capability, you decide to run the process with Python. Using the resources available in the STK Programming Help and Github, you will explore how to model a mission with a script.

Prerequisites

Before starting this lesson, verify that you have a working environment for running the Python code and have installed the STK Python API.

1. Confirm you have a working Python environment and a Python interpreter.
    - See **https://www.python.org/about/gettingstarted/** for details.
    - Recommended interpreters are VS Code, Spyder, Jupyter Notebook / JupyterLab, etc.
2. Install the STK Python API.
    - See Getting Started with the STK Python API at **https://help.agi.com/stkdevkit/#python/pythonGettingStarted.htm** for details.

Note:

The STK Python API is only available with version 12.1 of the STK software or newer. The install wheel (.whl) file is included with your STK software installation. If it is not already installed, use pip to install it. An example Windows console command is below:

```
pip install "C:\Program Files\AGI\STK_ODTK 13\bin\AgPythonAPI\agi_
stk<..ver..>-py3-none-any.whl"
```

If you are using an older version of the STK software, then use the win32com or comtypes module.

3. Close any open or background STK scenarios, as this script will launch a new mission.

Using the STK Programming Help

Before attempting to code anything, take a moment to familiarize yourself with the STK Programming Interface Help system at **https://help.agi.com/stkdevkit/index.htm**.

You will find a wealth of information within the STK Programing Interface Help system, including integration tutorials, code snippets, decision trees, and library references.

Under the Using Core Libraries help topic, you will find additional information on using the STK Object Model and Connect.

The STK Object Model help topic listed under Using Core Libraries contains useful code snippets in various programming languages. It also contains information on the various COM libraries that make up the STK Object Model. This information includes useful diagrams that help visualize how the STK Object Model is structured.

In the Connect section of the Using Core Libraries help topic, you will find useful Getting Started information outlining the basics of Connect commands and response formats. The section also contains useful code snippets that demonstrate the syntax for various Connect commands.

Launching the code

Copy the starter script from the STK install and open it in your preferred interface.

1. Using File Explorer, browse to the location of the script file located at **<Install Dir>\Data\Resources\stktraining\scripts\Python_STK_Training.py**.
2. Copy the Python_STK_Training.py file.
3. Paste the Python_STK_Training.py file in your preferred accessible location.

4. Close File Explorer.

5. Use your preferred method to open the Python_STK_Training.py file.

> **Note:** If you don't want to use the Python script file to complete this tutorial, you can enter your commands right in a Python command window. Optionally, you can also use the Jupyter Notebook file at <Install Dir>\Data\Resources\stktraining\scripts\Python_Jupyter_STK_Training.ipynb. For using the .ipynb file, please open a new STK scenario so the script can connect to it.

Using the script

The Python script file contains the completed script needed for this tutorial. You can run the script in its entirety, however, it is recommended that you read through each block and run them one at a time to understand the structure and functionality of the various commands.

1. Click into the individual cell of code.

2. Run the cell (for example, Shift + Enter in VS Code) or a right-click context menu option (for example, "Run Selection/Line in Python Terminal").

3. Follow the steps as outlined.

Setting up your workspace

With the STK Python API, you can import packages.

1. The first cell of your script contains the import statements needed to run the STK scenario on the desktop and insert STK objects.

```
# Set up your Python workspace
# Note: The STK Python API used in this lesson is only available with STK 12.1 or later.
# If not already installed then use pip to install it:
# pip install agi.stk<..ver..>-py3-none-any.whl
# If you are using an older version of STK, then use win32api or comtypes


from agi.stk13.stkdesktop import STKDesktop
from agi.stk13.stkobjects import *

# If you are using the STK/Astrogator® capability, uncomment the below:
# from agi.stk13.stkobjects.astrogator

# If you are using the Aviator capability, uncomment the below:
# from agi.stk13.stkobjects.aviator

# Python helper library imports
# import os

# In the next section you will start the STK application.
```

2. Start the STK application instance. Use the appropriate script below depending on how you want to connect to the STK application:

```
# NOTE for the Ansys Systems Tool Kit® (STK®) Cloud(TM) web browser-based
# digital mission engineering software:
# you can change your script to connect to an active instance
# instead of creating a new instance of the STK application.

# Connect to an instance of the STK application:
# stk = STKDesktop.AttachToApplication()

# Create a new instance of the STK application:
# Optional arguments set the application visible state and the user-control
# (whether the application remains open after exiting python).

stk = STKDesktop.StartApplication(visible=True, userControl=True)

# Check your Task Manager to confirm that the STK application was called.
```

> **Note:**
> Use the STK_PID environment variable when you have multiple instances of the STK application running:
>
> ```
> STK_PID = os.getenv('STK_PID')
> stk = STKDesktop.AttachToApplication(pid=int(STK_PID))
> ```

3. Grab a handle on the STK application root:

```
root = stk.Root
```

Recall that the AgStkObjectRoot object is at the apex of the STK Object Model. The associated IAgStkObjectRoot interface provides the methods and properties to load or create new scenarios and access the Object Model Unit preferences. Through the Connect command, you have a pointer to the IAgUiApplication interface; however, the STK Python API provides a direct handle to the IAgStkObjectRoot via the Root property in STKDesktop or the NewObjectRoot method in STKEngine.

4. Check that the root object has been built correctly and check the type:

```
type(root)
```

The output should be `agi.stk13.stkobjects.AgStkObjectRoot`.

5. Once the above sections are run in the script, it will create a new STK application window.

Connecting to the STK application instance and designing a new scenario

Now that you have launched the STK application via the Python scripting interface, connect to the application instance to create a new scenario and analysis period and to reset the animation time.

1. The next task is to create a scenario via the NewScenario method of the IAgStkObjectRoot interface. According to the documentation, the NewScenario method expects to be passed a string representing the name of the scenario, but does not return anything.

```
# 1. Define a Scenario object.

root.NewScenario("Python_STK_Training")
scenario = root.CurrentScenario
```

2. Run the following code to set the analysis period:

```
# 2. Set the analytical time period.

scenario.SetTimePeriod('Today','+24hr')
```

3. Run he following code to reset the animation time:

```
# 3. Reset the animation time to the newly established start time.

root.Rewind()
```

Inserting and Configuring objects

With a new scenario created, it's time to populate the scenario with objects. Use the STK Python API and the STK Connect commands, via the ExecuteCommand method, to create a facility and a LEO satellite.

1. Run following code to add a target object to the scenario:

```
# 1. Add a Target object to the scenario.

target  = AgTarget(scenario.Children.New(AgESTKObjectType.eTarget,"GroundTarget"))
```

Casting the object returned from the New method allows for better IntelliSense in your integrated development environment (IDE), but is optional; the object returned will be AgTarget at runtime even without the case.

2. Run following code to move the target object to the desired location:

```
# 2. Move the Target object to a desired location.

target.Position.AssignGeodetic(50,-100,0)
```

3. Run the following code to add a satellite object to the scenario. Take a moment to create a LEO satellite using Python and STK Connect commands, via the ExecuteCommand method:

```
# 3. Add a Satellite object to the scenario.

satellite  = AgSatellite(root.CurrentScenario.Children.New
(AgESTKObjectType.eSatellite,"LeoSat"))
```

4. Examine the Connect command below before running. In it, you will be using the Set State Classical connect command. Rather than manually setting the times, you will use the defined scenario times. First, print them to confirm:

```
print(scenario.StartTime)
print(scenario.StopTime)
```

Then run the code to execute the command:

```
# 4. Propagate the Satellite object's orbit.

root.ExecuteCommand('SetState */Satellite/LeoSat Classical TwoBody "' + str
(scenario.StartTime) + '" "' + str(scenario.StopTime) + '" 60 ICRF  "' + str
(scenario.StartTime) + '" 7200000.0 0.0 90 0.0 0.0 0.0')
```

Computing access between objects

Now that the scenario is fully built, the final task is to extract data and perform a basic analysis. You have a scenario with a Target object and a Satellite object. Determine when the Satellite object can access the Target object. For practice, locate the code to compute an access between two STK Objects using the IAgStkObject interface **HINT:** Review the Python Code snippets at **http://help.agi.com/stkdevkit/index.htm#stkObjects/ObjModPythonCodeSamples.htm**.

1. Go to the STK Programming Interface Help files.
2. Locate the code needed to compute an access between two STK objects using the IAgStkObject interface. The access is between the Satellite object and the Target object.
3. If you cannot locate the code, see below:

   The location of the required code snippets is STK Programming Interface > Using Core Libraries > STK Object Model > Python Code Snippets. Locate STK Objects > Access. The required snippet is Compute an access between two STK Objects (using IAgStkObject interface).

   ```
   access = satellite.GetAccessToObject(target)
   access.ComputeAccess()
   ```

Retrieving access data and satellite altitude data

The final task is to extract data and perform a basic analysis. You have just computed access between the two objects, so you can use the STK data providers to pull data out of the scenario.

1. Retrieve and view the access data in Python:

   ```
   accessDP         = access.DataProviders.Item('Access Data')

   results          = accessDP.Exec(scenario.StartTime, scenario.StopTime)

   accessStartTimes = results.DataSets.GetDataSetByName('Start Time').GetValues()

   accessStopTimes  = results.DataSets.GetDataSetByName('Stop Time').GetValues()

   print(accessStartTimes,accessStopTimes)
   ```

   This will retrieve and view the access data in Python.

2. Retrieve and view the altitude of the satellite during an access interval. Run the following code to call to the satellite's LLA State data provider:

   ```
   satelliteDP        = satellite.DataProviders.Item('LLA State')

   satelliteDP2       = satelliteDP.Group.Item('Fixed')

   rptElements        = ['Time', 'Lat', 'Lon', 'Alt']

   satelliteDPTimeVar = satelliteDP2.ExecElements(accessStartTimes,accessStopTimes, 60,
   rptElements)

   satelliteAltitude  = satelliteDPTimeVar.DataSets.GetDataSetByName('Alt').GetValues()

   print(satelliteAltitude)
   ```

In the above lines, note how the data providers must follow the structure of data provider folder, subfolder, and selection.

Saving your work

You have just completed the Integrating STK with Python tutorial. Don't forget to save your work.

1. To save your scenario, run the following code to create a new directory and then tell Python to save all the scenario files there:

```
import os
os.mkdir('Python_STK_Training')
```

   This will create a new folder named Python_STK_Training at the same level as your .py script.

   > **Important:** If you are entering your Python commands directly through a command-line interface, this folder will be created in the directory out of which your Python interpreter is being run (for example, C:\Program Files\Python310). If however you are running your commands through a Python script, the new folder will be created in the same directory as your script. You should always pay close attention to where and how you are running things in your Python environment.

2. Run the next lines to use your custom path to save the scenario files:

```
imdirectory = os.getcwd() + "\\Python_STK_Training"
root.ExecuteCommand('Save / * \"' + directory + '\"');
```

With your scenario safely saved, you can close out of the STK application.

Summary

You can expand and automate workflows like those used in this tutorial to quickly build scenarios and analyze missions.

Part 15: Model Aircraft Missions with Aviator

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Aircrew mission planners require analytical tools that allow them to determine how atmospheric phenomena and terrain will affect the performance of an airborne mission. Furthermore, they need the ability to model real-world aircraft performance that accounts for variations in airframe performance characteristics and mission requirements. In this lesson, you want to fly a small commuter jet from the City of Colorado Springs Municipal Airport to Telluride Regional airport using Navigational Aids (NAVAIDs) as waypoints. You want to determine how much fuel is required and how much payload can be carried on board the aircraft in a fast, easy way.

Solution

Use the STK software's *Aviator* capability to:

- Design a cross-country flight route by defining elements of a mission
- Load airfield runway data into the scenario for takeoff and landing procedures using the Aviator Catalog Manager
- Add NAVAIDs using the Aviator catalog interface to simulate physical devices on the ground that aircraft can detect and fly to along its route
- Determine payload requirements and the amount of fuel consumed during the flight using selected Data Providers

Creating a new scenario

First, you must create a new scenario and then you will build from there.

1. Launch the STK application ( ).

2. Click **Create a Scenario** in the Welcome to STK dialog box.

3. Enter the following in the STK: New Scenario Wizard:

Option	Value
Name	STK_Aviator
Location	Default
Start	Default (recommend changing the time to 18:00:00.000 UTCG for daylight)
Stop	+ 1 hr

4. Click **OK** when you finish.

5. Click Save ( ) when the scenario loads. The STK application creates a folder with the same name as your scenario for you.

6. Verify the scenario name and location in the Save As dialog box.

7. Click **Save** .

Decluttering labels in the 3D Graphics window

Your analysis will take place in very mountainous terrain, which can obstruct object labels. Enable the Label Declutter option to separate the labels on objects that are in close proximity for better identification in small areas.

1. Bring the 3D Graphics window to the front.

2. Click Properties ( ) in the 3D Graphics window Default toolbar.

3. Select the Details page when the Properties Browser opens.

4. Select the Enable check box in the Label Declutter panel.

5. Click **OK** to accept the change and to close the Properties Browser.

Understanding the Aviator capability

The STK software's *Aviator* capability provides an enhanced method for modeling aircraft–more accurate and more flexible than the standard Great Arc propagator. An aircraft using *Aviator* is defined by the type of aircraft and by the mission it performs. This structure allows you to utilize an aircraft for much more than simple point-to-point travel.

An aircraft using *Aviator* as its propagator can carry out operations that are more complex than a just a transit between two points. The process of defining a mission in *Aviator*, therefore, encompasses much more than merely selecting route points. A mission includes the flight procedures and performance characteristics of the aircraft and describes not only where the aircraft goes, but how it goes there and what it does along the way.

Whether a mission is as simple as a transit between two points, or as complex as a patrol mission in which the aircraft has been retasked to respond to a threat, the method for designing a mission is the same in principle. To define a mission, you must:

- Select and configure the aircraft model that you wish to use
- Insert and define the phases of the mission and select the performance models you wish to employ in each
- Insert and define the procedures that the aircraft will execute in each phase

To aid in mission design and planning, *Aviator* provides a catalog structure for the loading and saving of aircraft, airports, NAVAIDs, runways, VTOL points, and waypoints. Each of these mission elements has an associated catalog in the STK application.

Using the Aviator Catalog Manager

The Aviator Catalog Manager is a utility that allows you to view the contents of catalogs, create new items, copy or edit existing items, and search for specific items. You can use the Aviation Catalog Manager to import catalogs of compatible data to define mission elements like runways and waypoints. To aid in your mission design, use the Aviation Catalog Manager to load an ARINC424 data file containing navigation information into your scenario.

Loading navigation data using the Aviator Catalog Manger

Open the Aviator Catalog Manager from the Utilities menu and load a ARINC424 data file containing navigation information. ARINC424 data files are the only valid data sources for NAVAID and airport sites.

1. Select Aviator Catalog Manager... ( ) in the Utilities menu.
2. Resize the Aviator Catalog Manger window by extending it out to the right so that you can see more space in the large blank area.
3. Expand (⊞) Runway ( ).
4. Select ARINC424 runways ( ).
5. Click the Use Master Data File ellipsis (...).
6. Navigate to <Install Dir>\Data\Resources\stktraining\samples when the Open dialog box appears.
7. Select the FAANFD18 data file.
8. Click **Open** to select the file and to close the Open dialog box.
9. Click **Save** when you return to the Aviator Catalog Manager.

Determining the length of a runway

The aircraft will fly to and land at Telluride Regional Airport that is located in Telluride, Colorado. Determine the length of the runway using the information displayed in the Aviation Catalog Manager.

1. Enter Telluride in the Filter field.
2. Select the Enter key.
3. Select TELLURIDE RGNL 09 27 ( ) in the list under ARINC424 runways ( ).

   Note that TELLURIDE RGNL 09 27 ( ) is marked with a red dot ( ). Items marked with a red dot in the Aviator Catalog Manager and the Aviator catalog interface are read only and cannot be modified.
4. Examine at the properties on the right.
5. Locate the Length field. The runway at Telluride Regional Airport is 7,111 feet long.

When looking at runway data in the Aviator Catalog Manager, the two numbers next to the runway are reciprocal headings of the runway. For instance, 09 is 90 degrees (meaning it points east) and 27 is 270 degrees (meaning it points west). If you're landing on runway 09, you are approaching it from the west. If there is more than one runway pointing in the same direction (that is, parallel runways), each runway is identified by appending left (L), center (C) and right (R) to the number to identify its position, when facing its direction; for example, runways one-five-left (15L), one-five-center (15C), and one-five-right (15R) are parallel runways on headings of 150 degrees.

Inserting analytical objects from the Aviator Catalog Manager

The small commuter jet will take off from the City of Colorado Springs Municipal Airport and land at Telluride Regional Airport. The Telluride Regional Airport runway sits on a plateau and dips slightly in the center, which can provide a challenging landing for the pilot. Weather conditions in the area often rapidly change, and pilots must be aware of issues impacting the airfield, such as high terrain exceeding 14,000 feet as well as the runway's location on a plateau with a 1,000-foot drop should the aircraft slide off of the runway.

Use the Aviator Catalog Manager to insert the Telluride Regional Airport and the City of Colorado Springs Municipal Airport as Place objects into the scenario.

Inserting the Telluride Regional Airport

Add the runway at Telluride Regional Airport as a Place object. The center point of the runway will form the coordinates of the Place object.

1. Right-click on TELLURIDE RGNL 09 27 ( ) under ARINC424 runways ( ).
2. Select Create STK Object from waypoint... in the shortcut menu.
3. Open the Type of Object drop-down list in the Create STK Objects window.
4. Select Place.
5. Click OK to close the Create STK Objects window.

Inserting the City of Colorado Springs Municipal Airport

Add a runway at the City of Colorado Springs Municipal Airport as a Place ( ) object.

1. Enter Colorado Springs in the Filter field.
2. Select the Enter key.
3. Right-click on CITY OF COLORADO SPRINGS MUNI 17L 35R ( ) in the list under ARINC424 runways ( ).
4. Select Create STK Object from waypoint... in the shortcut menu.
5. Open the Type of Object drop-down list in the Create STK Objects dialog box.
6. Select Place.

7. Click **OK** to close the Create STK Objects dialog box.
8. Close (✖) the Aviator Catalog Manager when finished.

Obtaining situational awareness

Now that you have the center points of both runways entered as Place objects, you can quickly zoom to them to view the runways and surrounding terrain features.

1. Bring the 3D Graphics window to the front.
2. Right-click on CITY_OF_COLORADO_SPRINGS_MUNI_17L_35R ( ) in the Object Browser.
3. Select Zoom To in the shortcut menu.
4. Use your mouse to change the view so that you can view the runway and its surroundings.
5. Zoom to TELLURIDE_RGNL_09_27.
6. Use your mouse to change the view so that you can view the runway and its surroundings.

Inserting an Aircraft object

Insert an Aircraft object, which you will use to create a flight plan.

1. Bring the Insert STK Objects tool to the front.
2. Select Aircraft ( ) in the Select An Object To Be Inserted list.
3. Select Insert Default ( ) in the Select A Method list.
4. Click **Insert...** .
5. Right-click on Aircraft1 ( ) in the Object Browser.
6. Select Rename in the shortcut menu.
7. Rename Aircraft1 ( ) Mission_Acft.

Selecting Aviator as the propagator

With *Aviator*, the aircraft's route is modeled by a sequence of curves parametrized by well-known performance characteristics of aircraft, including cruise airspeed, climb rate, roll rate, and bank angle. The precise state of an aircraft at any given time can be computed analytically – swiftly, and without excessive data storage needs. To use this capability, you must first set your aircraft to use *Aviator* as its propagator.

1. Right-click on Mission_Acft ( ) in the Object Browser.
2. Select Properties ( ).
3. Select the Basic - Route page when the Properties Browser opens.
4. Open the Propagator drop-down list.
5. Select Aviator.
6. Click **Apply** to accept the change and to keep the Properties Browser open.
7. Read the information in the Flight Path Warning dialog box.
8. Click **Optimize STK for Aviator** .
9. Click **OK** to close the Fight Path Warning dialog box.

Configuring an Aviator mission

The mission window is used to define the aircraft's route when Aviator has been selected as the propagator. The mission window contains three toolbars – Initial Aircraft Setup, Phases of Flight, and Procedures and Sites – that enable you to define the aircraft that you are modeling and to create, modify, and delete phases and procedures. The mission list provides an overview of the mission by listing each of the mission phases and the procedures within them, in the order in which they will be executed. The mission profile can display a variety of data describing the mission.

Selecting an aircraft model

The buttons on the Initial Aircraft Setup toolbar are used to define the aircraft model that will be used in the mission. The basic models found in Select Aircraft dialog box are representative of an aircraft type, but not a specific aircraft. It's up to you to customize the model you choose to match actual aircraft characteristics. Since this is an introduction to *Aviator*, you will only make some minor changes.

1. Click Select Aircraft ( ) In the Initial Aircraft Setup toolbar.
2. Right-click on Basic Business Jet ( ) in the User Aircraft Models ( ) list in the Select Aircraft dialog box.

   Note that Basic Business Jet ( ) is read only ( ). To make any modifications, you must first copy the model.
3. Select Duplicate in the shortcut menu.
4. Right-click on Basic Business Jet Copy ( ).
5. Select Rename in the shortcut menu.
6. Rename Basic Business Jet Copy ( ) to COS_to_TEX.

   COS is the IATA airport code for City of Colorado Springs Municipal Airport and TEX for Telluride Regional Airport.
7. Select COS_to_TEX ( ) in the User Aircraft Models ( ) list.
8. Click **OK** to close the Select Aircraft dialog box.
9. Click **Apply** to accept your changes and to keep the Properties Browser open.

Editing performance models

Aircraft Properties provide access to performance models. Performance models are used to define the behavior of the aircraft in flight. By specifying performance models to use with each phase of the mission, you can vary the manner in which the aircraft performs based on the priorities of the mission. You'll use default settings mostly.

The Basic tab is comprised of three sections - Level Turns, Climb and Descent Transitions, and Attitude Transitions. You want to determine how much fuel is consumed during the flight and the weight of the mission aircraft when it lands.

The Aerodynamics tab is used to define the methods used to compute lift, drag, angle of attack, sideslip and intermediate / derived values.

1. Click Aircraft Properties ( ) in the Initial Aircraft Setup toolbar.
2. Select the Acceleration ( ) Built-In Model ( ) in the COS_to_TEX dialog box.
3. Select the Aerodynamics tab.
4. Open the Strategy drop-down list.
5. Select HighFast.

The High Fast aerodynamics strategy uses thrust to generate a lift vector, which provides the ability to track fuel burn during lift. Additionally, it generates the forces perpendicular to the velocity vector to provide maneuvering.

6. Read the data in the AeroProp Warning dialog box.

   The High Fast aerodynamics strategy must be paired with its High Fast propulsion counterpart.

7. Click **OK** to accept your changes and to close the AeroProp Warning dialog box.

Changing the Cruise Performance Model parameters

The Basic Cruise performance model is comprised of a simple set of parameters that define the flight characteristics of the aircraft during level flight. Since this is a fairly short flight, the aircraft will climb to 25,000 feet and level off.

1. Select the Cruise ( ) Built-In Model ( ).
2. Enter 25000 ft in the Default Cruise Altitude field.
3. Click **Save** .
4. Click **Close** to close the COS_to_TEX dialog box.

Inserting the Aircraft Configuration settings

The Aircraft Configuration dialog box is used to define the aircraft's fuel and payload configuration. The Basic tab is used to define the empty parameters of the aircraft, and displays the total values, based on the stations and fuel tanks defined for it.

1. Click Configuration ( ) in the Initial Aircraft Setup toolbar.
2. Select the Basic tab when the Aircraft Configuration dialog box opens.
3. Enter 31000 lb in the Empty Weight field.

   This is where you add payload weight. For instance, this will account for the pilot, instructor, any passengers and baggage.

4. Note the Max Landing Weight value of 40000 lb.
5. Note the Total Weight value of 51000 lb.

   This is the empty weight plus the default fuel weight of Mission_Acft ( ), which is 20,000 pounds of fuel.

Understanding the Stations tab

The Stations tab is used to define internal fuel tanks, stations, and external fuel tanks that are attached to the stations.

1. Select the Stations tab.
2. Select Internal Fuel ( ).
3. Note the Capacity and Initial state values.

   After your initial analysis, you may need to adjust the initial state.
4. Click **OK** to accept your changes and to close the Aircraft configuration dialog box.
5. Click **Apply** to accept changes and to keep the Properties Browser open.

Adjusting the Mission Wind Model

Use the Wind and Atmosphere Model tool to simulate wind and atmospheric conditions for the scenario, a mission, a specific procedure, or a group of selected procedures. For the purposes of this scenario, you will use a Constant Bearing / Speed wind model for your analysis.

1. Click Mission Wind Model ( ) in the Initial Aircraft Setup toolbar.
2. Note that the Model Type defaults to Constant Bearing/Speed.
3. Enter 180 deg in the Wind Bearing field in the Wind panel when the Mission_Acft (UI) wind/atmosphere model dialog box opens.
4. Enter 20 nm/hr in the Wind Speed field.
5. Click **OK** to accept your changes and to close the Mission_Acft (UI) wind/atmosphere model dialog box.
6. Click **Apply** to accept the changes and to keep the Properties Browser open.

Building the phases of flight with sites and procedures

Every mission must have at least one phase, and can have as many phases as you desire. You can select a specific set of performance models to use with each phase, allowing you perform one mission with multiple performance characteristics.

Each phase is composed sites and procedures. A site defines the location (excluding altitude) of the procedure and the types of procedures that are available to select. The exact relationship between the site location and the procedure is dependent on the specific procedure. A procedure is an action the aircraft performs at, or relative to, a site.

Your aircraft's mission will have a single phase. It will take off from the City of Colorado Springs Municipal Airport and fly direct to the Blue Mesa VOR/DME. A VOR/DME is a radio beacon that combines a VHF omnidirectional range (VOR) with a distance measuring equipment (DME). Turning at the Blue Mesa VOR/DME, the aircraft will fly to the Cones VOR/DME, then begin its final approach to and land at Telluride Regional Airport.

Selecting the takeoff runway

If you have ARINC424 airport data available in the Aviator Catalog Manager, you can define a site using an airport in that data.

1. Click Insert Procedure After ( ) in the Procedures and Sites toolbar.
2. Select Runway from Catalog ( ) in the Select Site Type list in the Site Properties dialog box.
3. Enter Colorado Springs in the Filter field.
4. Select the Enter key.
5. Select CITY OF COLORADO SPRINGS MUNI 17L 35R ( ) Under ARINC424 runways ( ).
6. Click **Next >** .

Selecting the Takeoff procedure

A Takeoff procedure launches an aircraft from a runway site into the air.

1. Select Takeoff ( ) in the Select Procedure Type list in the Procedure Properties dialog box.
2. Set the following options:

Option	Value
Name	COS Runway
Runway Heading - Use headwind runway	Selected
Runway Altitude Offset	7 ft
Use Terrain for Runway Altitude	Selected

3. Click **Finish** .
4. Click **Apply** to apply the changes and to keep the Properties Browser open.

You set runway altitude offset to seven feet. Most aircraft models used in the STK application have their center point in the middle of the 3D Graphics model. You have to adjust this if you don't want your aircraft to appear half buried in the terrain.

Inserting an End of Previous Procedure site type

The end of the previous procedure can be used as a waypoint to define the site of the next procedure. In this instance, due to terrain, you want to gain altitude prior to flying to Blue Mesa VOR/DM, which is located in mountainous terrain.

1. Click Insert Procedure After ( ) in the Procedures and Sites toolbar.
2. Select End of Previous Procedure ( ) in the Select Site Type list in the Site Properties dialog box.
3. Enter Climb in the Name field.
4. Click **Next >** .

Selecting a Basic Maneuver procedure

A Basic Maneuver procedure is a single action undertaken by the aircraft. It is unlike most procedures in *Aviator*, which represent sets of actions that together comprise a common flying procedure.

1. Select Basic Maneuver ( ) in the Select Procedure Type list in the Procedure Properties dialog box.
2. Enter Straight 25 nm in the Name field.
3. Select the Horizontal / Navigation tab.
4. Note that Straight Ahead is selected for the Strategy.
5. Open the Strategy drop-down list to view other strategies.
6. Leave Strategy set to Straight Ahead when done.
7. Set the following options in the Basic Stop Conditions panel:

Option	Value
Time of Flight	Clear
Downrange	25 nm

This procedure will end if your aircraft reaches zero pounds of fuel or flies straight ahead for 25 nautical miles, whichever comes first.

Selecting the vertical / profile strategy

Vertical / Profile strategies can be specified for non-3D maneuvers.

1. Select the Vertical / Profile tab.
2. Open the Mode drop-down list in the Altitude panel.
3. Select Specify Altitude Change.
4. Enter 10000 ft in the Relative Altitude Change field.
5. Click **Finish** two times.
6. Click **Apply** to apply the changes and to keep the Properties Browser open.

In the Mission Profile, you can see that the aircraft climbs 10,000 feet in altitude from the end of its COS Runway procedure.

Selecting the Blue Mesa NAVAID

If you have ARINC424 NAVAID data available in the Aviator Catalog Manager, you can define a site using a NAVAID from that data.

1. Click Insert Procedure After ( ) in the Procedures and Sites toolbar.
2. Select Navaid from Catalog ( ) in the Select Site Type list in the Site Properties dialog box.
3. Enter HBU in the Filter field.
4. Select the Enter key.

   HBU is the FAA designator for the Blue Mesa VOR/DME near Gunnison, Colorado.
5. Right-click on HBU ( ) in the ARINC424 navaids( ) list.

   Note that HBU ( ) is read-only ( ).
6. Select Create STK Object from waypoint... in the shortcut menu.
7. Open the Type of object drop-down list in the Create STK Objects dialog box.
8. Select Place.
9. Click **OK** to close the Create STK Objects dialog box.
10. Click **Next >** .

Selecting a Basic Point to Point procedure

A Basic Point to Point procedure is a basic traverse between two waypoints.

1. Select Basic Point to Point ( ) in the Select Procedure Type list in the Procedure Properties dialog box.
2. Set the following options:

Option	Value
Name	Blue Mesa
Navigation Options - Nav Mode	Fly Direct
Enroute Options - Turn Factor	5.00

3. Click **Finish** .
4. Click **Apply** to apply the changes and to keep the Properties Browser open.

Setting the Nav Mode to Fly Direct tells the aircraft to fly direct to Blue Mesa. The Turn Factor is the maximum amount, expressed as a multiplier, that the turn radius will be increased to minimize the bank angle required to complete the turn.

Selecting the Cones NAVAID

Add the Cones NAVAID for the next procedure in the flight.

1. Click Insert Procedure After ( ) in the Procedures and Sites toolbar.
2. Select Navaid from Catalog ( ) in the Select Site Type list in the Site Properties dialog box.
3. Enter ETL in the Filter field.
4. Select the Enter key.

   ETL is the FAA designator for the Cones VOR/DME near Telluride, Colorado.
5. Right-click on ETL ( ) in the ARINC424 navaids ( ) list.
6. Select Create STK Object from waypoint... in the shortcut menu.
7. Open the Type of object drop-down list in the Create STK Objects dialog box.
8. Select Place.

9. Click **OK** to close the Create STK Objects dialog box.
10. Click **Next >** .

Selecting a Basic Point to Point procedure

The aircraft will start its turn prior to reaching Cones in order to line up with its decent into Telluride Regional Airport. It had climbed to approximately 25,000 feet at Blue Mesa. Now, it will start a slow descent to Cones while decreasing airspeed. It should be at approximately 15,000 feet when it reaches Cones.

1. Select Basic Point to Point ( ) in the Select Procedure Type list in the Procedure Properties dialog box.
2. Set the following options:

Option	Value
Name	Cones
Altitude - Use Aircraft Default Cruise Altitude	Clear
Altitude - Altitude	15000 ft
Enroute Options - Turn Factor	5.00
Enroute Cruise Airspeed - Airspeed Type (from drop-down list)	Other Airspeed
Enroute Cruise Airspeed - Airspeed	250 nm/hr

   The Nav Mode defaults to Arrive on Course for Next Procedure.

3. Click **Finish** .
4. Click **Apply** to apply changes and to keep the Properties Browser open.

Selecting the landing runway

Add the landing runway at Telluride Regional Airport for the final procedure of the flight.

1. Select Insert Procedure After ( ) in the Procedures and Sites toolbar.
2. Select Runway from Catalog ( ) in the Site Properties / Select Site Type section.
3. Enter Telluride in the Filter field.
4. Select the Enter key.

5. Select TELLURIDE RGNL 09 27 ( ) Under ARINC424 runways ( ).
6. Click **Next >** .

Inserting the Landing procedure

A Landing procedure brings an aircraft down from the air to a runway site. Using the Intercept Glideslope approach mode, the aircraft will perform a landing following VFR flight rules; it will use Basic Point to Point methodology to fly to the Initial Approach Fix Range and then descend to landing along the glideslope.

1. Select Landing ( ) in the Select Procedure Type list in the Procedure Properties dialog box.
2. Set the following options:

Option	Value
Name	Telluride Runway
Approach Mode	Intercept Glideslope
Runway Heading - Use headwind runway	Selected
Landing Options - Runway Altitude Offset	7 ft
Landing Options - Use Terrain for Runway Altitude	Selected

3. Click **Finish** .
4. Click **OK** to apply the changes and to close the Properties Browser.

Using the Message Viewer

The STK application uses the Message Viewer window to display error messages, warning messages, and informational messages. Currently, there is a warning in Message Viewer.

1. Open the View menu.
2. Select Message Viewer.
3. Expand as necessary.
4. Look at the latest messages.

   The maximum landing weight for the aircraft is 40,000 pounds. It is too heavy.

5. At the bottom of Message Viewer, you'll see a tab named All Messages.

6. Right-click on the All Messages tab.
7. Select Clear All Tabs in the shortcut menu.
8. Close (✖) the Message Viewer window.

Creating a custom report

You want to determine the payload requirements and the amount of fuel consumed during the flight. Start by creating a custom report to determine the amount of fuel consumed during the mission and the weight of the aircraft.

Building your report in the Report & Graph Manager

Open the Report & Graph Manager and create a new report style for your custom report.

1. Right-click on Mission_Acft ( ) in the Object Browser.
2. Select Report & Graph Manager... ( ) in the shortcut menu.
3. Select the My Styles ( ) folder in the Styles panel in the Report & Graph Manager.
4. Click Create new report style ( ) in the Styles toolbar.
5. Enter Fuel and Weight as the new report name.
6. Select the Enter key to set the new report name and to open the report's properties.

Selecting data providers and elements

The data provider Flight Profile By Time has the elements required for your analysis. You will use the Flight Profile By Time data provider and the following elements in your custom report:

- Time
- Fuel State: The total weight of fuel on board the aircraft in pounds.
- Fuel Consumed: The amount of fuel consumed during the mission in pounds. At mission start, the value is zero.
- Weight: The total weight of the aircraft = Empty Weight + FuelState, in pounds.

Flight data is sampled using a constant time step between grid points. This report style is only available for Aviator-propagated vehicles.

1. Select the Content page when the Properties Browser opens.
2. Expand (⊞) the Flight Profile By Time ( ) data provider in the Data Providers list.
3. Move ( ) the following data provider elements ( ) to the Report Contents list in the order shown:
   - Time
   - Fuel State
   - Fuel Consumed
   - Weight
4. Click **OK** to accept your changes and to close the Properties Browser.

Generating your custom report

Now that you have added the data providers to your report contents, generate and review the report.

1. Select Fuel and Weight ( ) in the My Styles ( ) folder.
2. Click **Generate...** .
3. Scroll through the report until you get to the last group of numbers.

   You can see the correlation between fuel state, fuel consumed and the weight of the aircraft. The initial fuel load was 20,000 pounds of fuel. You used approximately 2,800 pounds of fuel. Your landing weight is approximately 48,200 pounds. The maximum landing weight of the aircraft is 40,000 pounds. Using these numbers, you can determine how much fuel you need to fly from the City of Colorado Springs Municipal Airport to Telluride Regional Airport. When adjusting the initial fuel load, you want to land with a reserve fuel load of between 500 and 1,000 pounds in case of a missed landing, emergency, and so on.
4. Keep the report open.

Updating your custom report

Do some simple math and approximation. The most important thing is to get under the maximum landing weight while having at least 500 pounds of reserve fuel. You're using a constant wind speed and direction. Realistically, at altitude, winds will change, especially in the mountains. You need to remove at approximately 8,200 pounds of fuel to be under the maximum landing weight. If you remove 9,000 pounds of fuel, that would also get you under your maximum landing weight, but you'll still have over 8,000 pounds of fuel remaining. So you can adjust that amount by removing a total of 16,600 pounds of fuel.

Adjusting the initial fuel state

Remove the unneeded fuel by adjusting the initial fuel state of the aircraft.

1. Open Mission_Acft's ( ) Properties ( ).
2. Select the Basic - Route page when the Properties Browser opens.
3. Click Configuration ( ) in the Initial Aircraft Setup toolbar.
4. Select the Stations tab in the Aircraft Configuration dialog box.
5. Select Internal Fuel ( ).
6. Enter 3400 lb in the Initial state field.
7. Click **Apply** .
8. Click **OK** to close the Aircraft Configuration dialog box.
9. Click **OK** to accept the changes and to close the Properties Browser.

Refreshing the Fuel and Weight report

Refresh your report to account for the updated initial fuel state.

1. Return to the Fuel and Weight report.
2. Click Refresh (F5) ( ) in the report toolbar.
3. Scroll to the bottom of the report.

   Focusing on fuel consumed, fuel state and the weight of the aircraft, you can see that the plane is well below its maximum landing weight. You landed with plenty of reserve fuel.
4. Close the report and the Report & Graph Manager when you are finished.

Viewing the flight route in the 2D Graphics window

Use the 2D Graphics window to get a good view of the flight route.

1. Bring the 2D Graphics window to the front.
2. Zoom in until you can see the flight route.

Viewing the flight in the 3D Graphics window

Use the 3D Graphics window to observe the mission.

1. Bring the 3D Graphics window to the front.
2. Click Reset ( ) in the Animation toolbar.
3. Zoom To Mission_Acft ( ).
4. Adjust (,) the time step as desired.
5. Click Start ( ) to animate the scenario.
6. Click Reset ( ) when finished.
7. You can zoom to ETL ( ) and HBU ( ) to view the actual VOR/DME ground transmitter sites.

If you are flying in mountainous terrain, when using Aviator, it's a good idea to zoom out to view the entire flight route. Make sure none of the route enters and exits terrain. If that's the case, you will have to adjust your current procedures or add new ones to ensure a safe mission. This is a perfect example of why you need terrain data. If you are using the STK application where the Internet is not available, you should obtain local terrain data to be used in your scenario.

Summary

Using the *Aviator* capability, you planned a flight route for a small commuter jet taking off from the City of Colorado Springs Municipal Airport and landing at Telluride Regional Airport. You began by loading runway data using the Aviator Catalog Manager. During the initial aircraft configuration, you added 1,000 pounds to the aircraft to account for increased payload. You tweaked performance models to determine how much fuel was required for the flight. You added a constant wind bearing and speed to your analysis. You were introduced to multiple site properties and procedures that allowed you to mission plan an aircraft flight route from the City of Colorado Springs Municipal Airport to Telluride Regional Airport via Blue Mesa and Cones NAVAIDs. After the original analysis, you determined that you needed to remove excess fuel due to the aircraft being overweight during landing. After removing fuel, you determined how much fuel to use in order to fly between the airports and to land with an acceptable fuel reserve.

Part 16: Introduction to the Advanced CAT Tool

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Problem statement

Engineers and operators need to easily address situations in which the launch or operation of a satellite or system of satellites, or a related Earth-based operation, may be affected by the actual or apparent proximity of other orbiting objects. Such effects range from a temporary delay or loss of access to the system to physical damage to space-based assets. In this scenario, you want to analyze one days' worth of data to determine the probability of your satellite having a conjunction with any other satellites or orbital debris.

Solution

Use the Advanced CAT tool, which is a part of the STK software's *Conjunction Analysis* capability, to carry out a close-approach analysis between a primary object (your satellite) and secondary objects (satellites presenting a possible risk of collision). A conjunction occurs when two orbiting objects are closer than the specified minimum acceptable distance. You choose whether the distance is measured between the ellipsoidal threat volumes of the objects or by the range between the objects. In this scenario, you will complete a linear analysis using two-line element sets (TLEs).

Creating a new scenario

First, you must create a new scenario, then build from there.

1. Launch the STK application ( ).
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following options in the STK: New Scenario Wizard:

Option	Value

Name	STK_CAT
Location	Default
Start	Default
Stop	Default

4. Click **OK** when you finish.
5. Click Save (💾) when the scenario loads.

   The STK application automatically creates a folder with the same name as your scenario for you.
6. Verify the scenario name and location in the Save As dialog box.
7. Click **Save** .

Preparing your STK workspace

For this scenario, you will not use the Timeline View or the 2D Graphics window. Close them.

1. Close the Timeline View at the bottom of the STK workspace.
2. Close the 2D Graphics window.
3. Select the Window menu in the Menu Bar.
4. Select Tile Vertically to maximize the 3D Graphics window.

Updating the satellite databases

Satellite database files contain both identification information and orbital characteristics. While the identification information does not change often over time, the orbital characteristics do, and so the epoch for any TLE used for ephemeris generation for an object is always a concern. Thus, if you are searching for space objects with certain orbital characteristics, you should first obtain an appropriately dated satellite database file.

The Advanced CAT tool does not by itself add any objects to your scenario; instead, it propagates ephemerides for objects that you select for the specified time frame. With the Advanced CAT tool, you can use analytical objects defined within the scenario or by an external file, including TLE sets, GPS almanac files, and GP data files (that is, TLE files that use the SGP4 propagator).

- Satellite object files (*.sa) in your scenario
- Your STK scenario directory

- The TCE subdirectory of your user directory (this is the path of the SGP4 registry entry)
- The default satellite database directory, for example, C:\ProgramData\AGI\STK_ODTK 13\Databases\Satellite
- <GENDB>\Satellite, where GENDB is the path of the STKGENDB registry entry

Before using the Advanced CAT tool, update the TLE database appropriate for your analysis period.

Accessing the scenario database properties

You can download the current directory on the Scenario object's Database properties page by updating the database files.

1. Right-click on STK_CAT ( ) in the Object Browser.
2. Select Properties ( ) in the shortcut menu.
3. Select the Basic - Database page when the Properties Browser opens.

Saving the database

This will update the satellite database with the most current data for your analysis period.

1. Click **Update Database Files...** .
2. Ensure the Update Database option is selected when the Update Satellite Database dialog box opens.

   > **Important:** When performing an analysis for a past time period, the appropriate file would not be the most current file, but rather, an archived database(s) dated near the propagation time period. For such an analysis, use the Obtained Archived Database option. This option allows you to obtain an older version of the database from the date specified in the Archive Date field. If an archive is not available from the specified date, the archive for the next newest date is used instead. To avoid overwriting your default database file(s) with these archived file(s), click the Database ellipsis (...) to set the database directory, in which the appropriately dated database file(s) will be saved, to your scenario folder. The Advanced CAT tool will look in and use the database file(s) in your scenario folder first, before searching the default location.

3. Click **Update** .
4. Click **OK** to close the Information dialog box once the Database update is complete.
5. Click **Close** to confirm your changes and to close the Update Satellite Database dialog box.
6. Click **OK** to confirm your changes and to close the Properties Browser.

Inserting the Primary satellite

The STK software's *Conjunction Analysis (CAT)* capability is designed to address situations in which the launch or operation of a satellite or system of satellites, or a related Earth-based operation, may be affected by the actual or apparent proximity of other orbiting objects. Such effects range from a temporary delay or loss of access to the system to physical damage of space-based assets. The Advanced CAT tool provides a convenient way for you to carry out close approach analyses for multiple satellites. After applying selected filters, the Advanced CAT tool performs close approach analyses between primary objects (for example, satellites owned by or otherwise of interest to you) and secondary objects (those presenting a risk of collision), with reference to a threshold – a minimum acceptable distance – between the ellipsoidal threat volumes of the objects. Results are summarized in several available reports.

Inserting the satellite using the Orbit Wizard

For this scenario, you will use the Orbit Wizard to create a nominal primary satellite in order to focus on using the Advanced CAT tool.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Select Satellite ( ) in the Select An Object To Be Inserted list.
3. Select Orbit Wizard ( ) in the Select A Method list.
4. Click **Insert...** .

Using Orbit Wizard

The Orbit Wizard is a satellite-level tool designed to assist you in either creating any one of several standard orbits or designing your own satellite orbit. The configurable options available depends on the orbit type selected.

1. When the Orbit Wizard dialog box opens, set the following definitions for the satellite:

Option	Value
Type	Circular
Satellite Name	My_Sat
Inclination	45 deg
Altitude	700 km

2. Click **OK** to propagate My_Sat ( ) and to close the Orbit Wizard dialog box.

Inserting an AdvCAT object

The AdvCat object ( ) provides a convenient way for you to access the Advanced CAT tool to carry out close-approach analyses for multiple satellites and TLEs.

Adding the AdvCAT object to the Insert STK Objects tool

Add the AdvCAT object to the Insert STK Objects tool to be able to insert it into your scenario.

1. Bring the Insert STK Objects ( ) tool to the front.
2. Click **Edit Preferences...** .
3. Ensure the New Object page is selected when the Preferences dialog box opens.
4. Select the Show check box for the AdvCAT Object in the Define Default Creation Methods panel.
5. Click **OK** to confirm your selection and to close the Preferences dialog box.

Inserting the Advanced CAT object

Insert an AdvCAT object into your scenario.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert an AdvCAT ( ) object using the Insert Default ( ) method.

Setting up and defining the analysis objects

In analyzing close approaches between a primary and secondary object, the Advanced CAT tool assigns a threat volume, comprising an ellipsoidal shape enclosing the object, to each. Often, the volume represents the degree of uncertainty about an object's position. At any given instance, the Advanced CAT tool computes the range between the threat volumes of the two objects. A warning is given whenever the range falls below a user-selected threshold. A collision event occurs whenever the range between the two threat volumes becomes zero or negative.

Selecting the Primary object and defining its threat volume

Most of the Main page of the AdvCAT object's properties is comprised of two lists: a Primary List and a Secondary List. The Primary List contains satellites of interest to you, such as those that you own or wish to use. Select My_Sat as your Primary satellite.

1. Open AdvCAT1's ( ) Properties ( ).

   This also opens the Advanced CAT tool.

2. Select the Basic - Main page when the Properties Browser opens.
3. Select Satellite/My_Sat in the Available list in the Primary List panel.
4. Note that the Type is Scenario Object; this is the satellite object file in your scenario folder.
5. Move ( ) Satellite/My_Sat to the Chosen list.
6. Since you're evaluating potential linear conjunctions and not using covariance settings, keep the default Class, Tangential, CrossTrack, and Normal values.

   You can select the approach for the STK application to take to determine the dimensions of the threat volume. A Fixed dimension definition class specifies the dimensions of the threat volume ellipsoid based on the values you enter by three axes:

   - Tangential (in-track) – the dimension of the threat volume ellipsoid along the X axis
   - CrossTrack – the dimension of the threat volume ellipsoid along the Y axis
   - Normal (radial) – the dimension of the threat volume ellipsoid along the Z axis

   Actual threat volumes would be set based on established ellipsoid and covariance screenings.

7. Set the HardBodyRadius value to 3 m.
8. Click **Apply** to confirm your selection and to keep the Properties Browser open.

Selecting the Secondary object and defining its threat volume

The Secondary List contains satellites that present a potential risk of collision with, or unacceptably close approach to, satellites in the Primary List. The U.S. Strategic Command (USSTRATCOM) keeps track of thousands of space objects. These objects constitute the space object catalog. While most of the catalog is made available to the public, some information is restricted. AGI provides the publicly released information for use with the STK application in the form of satellite database files and TLE sets. AGI updates this data at least once and up to three times a day. Your analysis will check for potential conjunctions between My_Sat and the complete satellite database.

1. Select stkAllTLE.tce in the Available list in the Secondary List panel.
2. Note that the Type is GPElem File, and that the Date is the same as Satellite/My_Sat in the Primary List.
3. Move (➔) stkAllTLE.tce to the Chosen list.
4. Set the HardBodyRadius value to 2 m.
5. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Setting the Threshold value

Often, the threat volume represents the degree of uncertainty about an object's position at any given time. The Advanced CAT tool computes the range between the threat volumes of the two objects at specified time steps over a given interval. It gives a warning whenever the range falls below a user-selected threshold. A collision event occurs whenever the range between the two threat volumes becomes zero or negative. Increase the volume threshold.

1. Enter 15 km in the Threshold field.
2. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Reviewing the pre-filters

The basic idea behind close-approach processing is to start with all cataloged orbiting objects, then efficiently delete the members of the population that do not come within the specified distance of the reference object. The first step is eliminating as many members of the population as possible via geometric properties, which takes considerably less time than fully propagating these satellites and then computing close approaches. The final determination of the existence of close approaches is always done by sampling the ephemeris of the candidate and reference objects, but some pre-filters are usually applied to limit the amount of ephemeris generation that is required. Review the filters on the Advanced CAT - Advanced page.

1. Select the Basic - Advanced page.
2. Note the selected Pre-Filters and their values:
    - Apogee / Perigee — The Advanced CAT tool uses the apogee/perigee pre-filter to cut candidate close-approach objects having a range of altitude that does not overlap with that of the reference object.
    - Time — The goal of the time filter is to identify time intervals when each object in a pairing is close enough to the elliptical representation of the other object's trajectory to have a conjunction.

Computing possible conjunctions

After you have defined your close approach analysis parameters and selected and dimensioned your primary and secondary objects, you can perform your close approach analysis. When you compute possible conjunctions, any object threat volumes that fall within the threshold and pre-filter distances appear as ellipsoids in the 3D Graphics window after creating a report.

1. Select the Basic - Main page.
2. Select the Display Acknowledgment when done check box.

    Select this option to have a message box appear when close approach computation is completed.
3. Click **Compute** .

    During the computation process, the Advanced CAT tool will display information in the Status Bar at the bottom of the STK workspace to apprise you of the activity being carried out and the time remaining before computation is completed.
4. Click **OK** to close the Advanced CAT dialog box after reading the message.
5. Keep AdvCAT1's () Properties open.

Creating a Close Approach by Min Range report

The Advanced CAT tool makes available a variety of report styles for presenting and summarizing information about the primary and secondary objects and the results of the close approach analyses. Several styles are available from the Report & Graph Manager for an AdvCAT object.

Generating a Close Approach By Min Range report

Generate a Close Approach By Min Range report for your AdvCAT object.

1. Right-click on AdvCAT1 () in the Object Browser.

2. Select Report & Graph Manager... () in the shortcut menu.

3. Select Close Approach By Min Range () report style, located in the Installed Styles () folder, in the Styles panel when the Report & Graph Manager opens.

4. Click **Generate. . . .**

5. Look through the report and focus on Object Name, Time In (UTCG), Time Out (UTCG), Min Separation (km) and Min Range (km) when the report opens.

   - Object Name – This is the name of the secondary object involved in the close-approach event.
   - Time In – This is the start time of the event. For this scenario, it is when the threat volume is within 15 kilometers of the primary's threat volume.
   - Time Out – This is the stop time of the event. For this scenario, it is when the threat volume moves beyond 15 kilometers of the primary's threat volume.
   - Min Separation – This is the separation distance between the primary and secondary threat volumes at the time of closest approach. If the ellipsoids intersect, then the value will be "Intersect."
   - Min Range – This is the distance between the center points of the primary and secondary objects at the time of closest approach.

     Recall that you configured the Advanced CAT tool to only report objects with threat volumes that come within 15 kilometers of your primary satellite's threat volume and also pass its pre-filters.

6. When finished, close the report.

Customizing the report

Add three more Events by Min Range data provider elements to your report. This will add situational awareness.

1. Right-click on the Close Approach By Min Range () report in the Installed Styles () folder, located in the Styles panel.

2. Select Duplicate () in the shortcut menu.

3. Select the Content page when the Properties Browser opens.

4. Select the Events by Min Range-Min Range data provider element in the Report Contents list.

   This will automatically expand (⊞) the Events by Min Range () data provider.

5. Insert () the following data provider elements into the Report Contents list:

- Time of Close Appr (TCA) ( ) – This is the time when the minimum range occurs between the primary and secondary bodies.
- Relative Velocity ( ) – This is the magnitude of the difference in the inertial velocities of the primary and secondary objects at a given time.
- Collision Probability (Analytic) ( ) – This is the probability of collision computed using an analytic method derived from the book "Spacecraft Collision Probability" by F. Kenneth Chan.

6. Click **OK** to confirm your selections and to close the Properties Browser.

Renaming and running the report

Rename your custom report, then generate the report.

1. Right-click on the Close Approach By Min Range ( ) report style, located in the My Styles ( ) folder, in the Styles panel.
2. Select Rename in the shortcut menu.
3. Rename your custom report My Close Approach By Min Range.
4. Click **Generate. . .** .
5. Review the report.
6. Look at the first Object Name in the report.
7. Note the Time of Close Appr (TCA) (UTCG).

   This is when the two satellites, not the threat volumes, are closest to each other. Again, this is based on TLE data, which don't contain covariance information.

Visualizing a close approach

You can use the information in your custom report to help visualize a close approach.

Identifying and inserting a close approach

Focus on the first satellite in the report.

1. Copy the SSC number from the Object Name of the first satellite in the report.
2. Bring the Insert STK Objects tool ( ) to the front.
3. Insert a Satellite ( ) object using the From Standard Object Database ( ) method.
4. Clear the Online check box in the Data Sources section when the Search Standard Object Data dialog box opens.

   When you search for an SSC number, only local database files will be shown in the Results list.
5. Paste the SSC number that you copied into the Name or ID field.
6. Click **Search** .
7. Select the satellite in the Results list.
8. Click **Insert** .
9. When the selected Satellite object propagates, click **Close** to close the Search Standard Object Data dialog box.

Setting the animation time to the Time In

Use the report to set the animation time to the time when the threat volume first comes within 15 kilometers of the primary's threat volume.

1. Return to the My Close Approach By Min Range report.
2. Right-click on the Time In (UTCG) of your selected Object Name.
3. Select Time In in the shortcut menu.
4. Select Set Animation Time in the Time In submenu.

Viewing the event in the 3D Graphics window

Zoom to your primary satellite and view the animation in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on My_Sat ( ) in the Object Browser.
3. Select Zoom To in the shortcut menu.
4. Click Decrease Time Step ( ) in the Animation Tool Bar until your Time Step value is 0.01 sec.

5. Click Step in Reverse ( ) one time.
6. Using your mouse, zoom out until you can see the primary and secondary ellipsoids.

   The primary and secondary ellipsoids are green; when they come within the 15-kilometer threshold, the ellipsoids turn yellow. When they intersect, they turn red. Other satellites that enter the threshold are represented by their own ellipsoids. They aren't analytical objects.
7. Click Start ( ) to animate the scenario.
8. Run the animation through the conjunction.
9. After the conjunction, when the two satellites turn green again, click Pause ( ).

Setting the animation time to the Time of Close Approach

Set your animation time to the time when the minimum range occurs between the primary and secondary bodies.

1. Return to the My Close Approach By Min Range report.
2. Right-click on the Time of Close Appr (UTCG).
3. Select Time of Close Appr (TCA) in the shortcut menu.
4. Select Set Animation Time in the Time of Close Appr (TCA) submenu.
5. Return to the 3D Graphics window.
6. Using your mouse, zoom in so that you can get a good view of both satellites.
7. When you're finished, Reset ( ) the scenario.
8. In the 3D Graphics window, click Home View ( ) on the 3D Graphics toolbar.
9. Close the My Close Approach By Min Range report and the Report & Graph Manager.

Visualizing all the satellites

At this time, only the satellites having threat volumes that fell within the 15-kilometer threshold and passing the pre-filters are visible in the 3D Graphics window. You can display all secondary ellipsoids if desired.

1. Return to AdvCAT1's ( ) Properties ( ).
2. Select the 3D Graphics - Attributes page.

In the Visibility panel are the primary ellipsoid and secondary ellipsoids that have conjunctions.

3. Select the All option.
4. Click **OK** to confirm your selection and to close the Properties Browser.
5. Clear the check boxes for both Satellite ( ) objects in the Object Browser.
6. In the Animation Tool Bar, click Increase Time Step ( ) until your Time Step value is 10.00 sec.
7. Click Start ( ) to animate the scenario.

   Remember, these satellites are represented by ellipsoids that are 20 × 10 × 5 kilometers in size.
8. Use your mouse to zoom out until you can see the geostationary belt.
9. When you're finished, Reset ( ) the scenario.

Summary

You began by updating the local satellite database with satellites that matched your scenario time period. Then you inserted the primary satellite into the scenario. After inserting an AdvCAT object, you moved the primary satellite from the Available list to the Chosen list. Then you used the complete satellite database (stkAllTLE.tce) for the secondary objects by moving it from the Available list to the Chosen list. You updated the hard-body radii, set the threshold to 15 kilometers, and then launched the close approach computation process. Using the Events by Time In data provider and elements, you created a custom report that allowed you to view important information concerning possible conjunctions with your primary satellite. You ended by visualizing a close approach and by visualizing all the satellite ellipsoids from the stkAllTLE.tce database.

Part 17: Design Trajectories with Astrogator

Note: Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

Important: This lesson requires version 12.7 of the STK software or newer to complete in its entirety.

Note: The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators require a quick way to design high-fidelity spacecraft trajectories for mission planning and operations. In this scenario, you will design a launch from a launch pad and successfully inject a satellite into a geosynchronous equatorial orbit (GEO).

Solution

Use the *STK/Astrogator*® capability to design:

- A basic launch phase to place a satellite into a parking orbit (low Earth orbit, or LEO)
- A transfer orbit injection (TOI) maneuver to transfer from LEO to GEO
- A synchronized orbit injection (SOI) maneuver to circularize the orbit at GEO

Creating a new scenario

First, you must create a new scenario, then build from there.

1. Launch the STK application ( ).
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following in the New Scenario Wizard:

Option	Value
Name	STK_Astrogator
Location	Default
Start	Default
Stop	+ 5 days

4. Click **OK** when you finish.
5. Click Save () after the scenario loads. The STK software creates a folder with the same name as your scenario for you.
6. Verify the scenario name and location in the Save As dialog box.
7. Click **Save** .

Updating the Insert STK Objects tool

Ensure the Launch Vehicle object appears in the Insert STK Objects tool.

1. Click **Edit Preferences...** in the Insert STK Objects tool.
2. Select the Show check box for Launch Vehicle when the Preferences dialog box opens.
3. Click **OK** to confirm your change and to close the Preferences dialog box.

Creating a launch vehicle

Insert a Launch Vehicle object. A Launch Vehicle object models the properties and behavior of a vehicle that follows an ascent trajectory from a launch point to an orbit insertion point.

1. Select Launch Vehicle ( ) in the Select an Object to Be Inserted list in Insert STK Objects tool.
2. Select the Insert Default ( ) method in the Select a Method list.
3. Click **Insert. . .** .
4. Right-click on LaunchVehicle1 ( ) in the Object Browser.

5. Select Rename in the shortcut menu.
6. Rename LaunchVehicle1 ( ) LaunchToLEO.

Selecting the simple ascent propagator

Propagators define the path that the launch vehicle will follow. Use the simple ascent propagator to define the ascent trajectory from a launch point to an orbit insertion point. The simple ascent propagator creates an ascent trajectory based on launch and insertion parameters. The trajectory is a simple curve rising vertically from the launch pad that turns over smoothly to insert the launch vehicle into orbit with a zero flight path angle at the insertion point using the specified velocity.

1. Right-click on LaunchToLEO ( ).
2. Select Properties ( ) in the shortcut menu.
3. Select the Basic - Trajectory page when the Properties Browser opens.
4. Ensure the Propagator is set to SimpleAscent.
5. Enter 7.3 km/sec in the Burnout Velocity field.

   This is the velocity of the launch vehicle at the designated Stop Time (burnout). A velocity of 7.3 kilometers per second will keep the resulting orbit nearly circular.
6. Click **Apply** to confirm your change and to keep the Properties Browser open.
7. Select the 2D Graphics - Attributes page.
8. Change the Color to teal.
9. Click **OK** to confirm your changes and to close the Properties Browser.

Viewing the launch vehicle trajectory

View LaunchToLEO's launch vehicle trajectory and ground track in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom To LaunchToLEO ( ).
3. Use your mouse to zoom out so you can see the launch vehicle trajectory and ground track.

4. Notice that the Launch Vehicle's ( ) default location is a launch pad at Cape Canaveral, Florida.
5. Notice it's trajectory ends at burnout.

Inserting a satellite

Insert a Satellite object, which you will use to create the orbit.

1. Insert a Satellite ( ) object using the Insert Default ( ) method.
2. Rename Satellite1 ( ) GEO_Sat.

Using the Astrogator capability

You will use *Astrogator* to design your spacecraft trajectory. The *Astrogator* capability contains specialized analysis for interactive orbit maneuver and spacecraft trajectory design. The *Astrogator* capability acts as one of the several types of propagators available for a Satellite object. Propagation concerns the determination of the motion of a body over time. According to Newton’s laws, the motion of a body depends on its initial state, its position and orientation at some known time, and the forces that act upon it over time. High-fidelity numerical propagators like *Astrogator* attempt to include all significant force models acting on the body. The numerical approach to propagation is much more accurate than ones that used closed-form solutions and those that incorporate some approximations, because it selects the correct step size in time. Here, speed is sacrificed during computation for the sake of accuracy. The orbit modeling is derived using full algorithms and correct ephemerides and is typically the best solution to model realistic problems.

1. Open GEO_Sat's ( ) Properties ( ).
2. Select the Basic - Orbit page when the Properties Browser opens.
3. Open the Propagator drop-down list.
4. Select Astrogator.

Setting up the Mission Control Sequence

The Mission Control Sequence is the core of your space mission scenario. The MCS functions as a graphical programming language, in which mission segments dictate how *Astrogator* calculates the trajectory of the spacecraft based on the general settings that you specify for the MCS itself. You can configure the Mission Control Sequence by selecting and organizing MCS Segments in a manner that produces your desired trajectory. There are two general types of segments: those that generate ephemeris and those that affect the execution of the MCS. You can use these segments interactively, so that ephemeris generated by one segment can cause another segment to change the way in which the MCS continues to run.

The left side of the MCS is represented schematically by a tree structure, which lists the segments that make up the MCS and depicts their relationships to each other. Above the tree is the MCS toolbar, which contains buttons that perform various MCS and individual segment operations. By default, an *Astrogator* satellite's MCS contains two segments: an Initial State segment and a Propagate segment that produce a low Earth orbit. The right side of the window contains the parameters of the segment that is currently selected in the MCS Tree.

Deleting the Initial State segment

Since you are modeling your mission beginning with a Launch Vehicle, remove the Initial State segment.

1. Select Initial State ( ) in the MCS.
2. Click Delete Segment ( ) on the MCS toolbar.
3. Click **Yes** to confirm the deletion.

Adding a Follow segment

A Follow segment is used to set a spacecraft to follow another vehicle (such as a Satellite or Launch Vehicle) at a specified offset, and to separate from that vehicle upon meeting specified conditions. Insert a new Follow segment to set GEO_Sat to follow LaunchToLEO, and then separate from LaunchToLEO at the end of its ephemeris.

1. Right-click on Propagate ( ) in the MCS.
2. Select Insert Before... in the shortcut menu.
3. Select Follow ( ) when the Segment Selection dialog box opens.
4. Click **OK** to confirm your selection and to close the Segment Selection dialog box.

Defining the Follow segment

To define a Follow segment, you must set the general parameters to describe the epoch and nature of the following operation, then specify the joining conditions, separation conditions, and spacecraft physical values as required by the general parameters you choose

1. Select Follow ( ) in the MCS.
2. Select the General tab.

3. Click the Leader ellipsis (...).
4. Select LaunchToLEO ( ) when the Select Component dialog box opens.
5. Click **OK** to confirm your selection and to close the Select Component dialog box.
6. Open the Joining drop-down list in the Additional Options panel.
7. Select Join at End of Leader's Ephemeris.

The separation parameter is automatically set to Separate at End of Leader's Ephemeris. By selecting this joining parameter, GEO_Sat uses the LaunchToLEO's final ephemeris point as the initial and final state of the Follow Segment. The segment goes though the ephemeris points of LaunchToLEO, adding the specified offset to them and then adding these points to the ephemeris of GEO_Sat. Ephemeris points are not added until the joining condition is met and are added until the separation condition is met.

Setting the Fuel Tank configuration

In an Initial State, Follow, or Launch segment, you can configure the spacecraft to specify physical values such as the fuel mass, dry mass, and pressure coefficients using the Spacecraft Parameters and Fuel Tank tabs. The default parameters used by *Astrogator* are generalized to be consistent with modern spacecraft design. You can easily customize them to more accurately model your spacecraft's physical characteristics. For this lesson, you'll customize GEO_Sat's fuel tank volume and fuel mass.

1. Select the Fuel Tank tab.
2. Set the following in the order shown:

Option	Value
Tank Volume	1.5 m^3
Maximum Fuel Mass	6000 kg
Fuel Mass	5000 kg

3. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Running the Mission Control Sequence

To calculate the trajectory of the spacecraft you must run the Mission Control Sequence. *Astrogator* will proceed through the MCS and run each segment, generating an ephemeris for the spacecraft. As it runs the MCS, *Astrogator* carries the trajectory and state of the spacecraft determined so far from one segment to the next to calculate the trajectory of the spacecraft.

1. Select Propagate ( ) in the MCS.
2. Note the current stopping condition is Duration, with a Trip value of 43200 sec (equivalent to half a day).
3. Click Run Entire Mission Control Sequence ( ) on the MCS toolbar.
4. Bring the 3D Graphics window to the front.
5. Click Home View ( ) to view GEO_Sat's ( ) trajectory.

Specifying the satellite's engine model

An Engine Model component models the thrust and specific impulse (I_{sp}) of a rocket engine. You can create a custom engine model in order to produce accurate results.

Creating a new engine model component

You'll duplicate a pre-built engine model in the Component Browser, then build from there.

1. Return to GEO_Sat's ( ) Properties ( ).
2. Click Component Browser ( ) on the MCS toolbar.
3. Open the Show Component Type drop-down list when the Component Browser window opens.
4. Select Astrogator Components.
5. Select the Engine Models ( ) folder in the View All Astrogator Components list.
6. Select Constant Thrust and Isp ( ) in the Engine Models list on the right.
7. Click Duplicate component ( ) on the Engine Models toolbar.

   Constant Thrust and Isp ( ) is read only and cannot be modified, so you must duplicate it before you can modify it.
8. Enter Test Engine in the Name field when the Field Editor dialog box opens.
9. Click **OK** to confirm your changes and to close the Field Editor dialog box.

Customizing your engine model

Modify the Constant Thrust and Isp engine model to specify the thrust and I_{sp} for your engine.

1. Double-click on the Test Engine () entry in the Engine Models list.
2. Double-click on the Thrust entry when the Test Engine dialog box opens.
3. Enter 13500 N in the Real Number field when the Real Number Field dialog box opens.
4. Click **OK** to confirm your change and to close the Real Number Field dialog box.
5. Double-click on the Isp entry.
6. Enter 2000 s in the Real Number field when the Real Number Field dialog box opens.
7. Click **OK** to confirm your change and to close the Real Number Field dialog box.

Adding the engine model component to your user collection

You can save a custom-built component to your user collection to make it available in other scenarios.

1. Click **Add to Collection** .

   Note that Test Engine's icon has changed from to to indicate the component has been added to your user collection.

2. Click **Close** to confirm your changes and to close the Component Browser.

> **Further Reading:** For more information about engine models, refer to the *Astrogator* technical notes.

Designing the transfer orbit injection (TOI)

Use the Propagate segment to fly to the first maneuver time. The orbit is circular and therefore the burn can take place at any time and result in a similar Delta-V. You require an inclination as close as possible to zero once you enter GEO. To minimize the required Delta-V to both circularize and change inclination at GEO, you will combine those maneuvers into one. For that to be successful, the apogee of the transfer orbit will be the ascending or descending node of the orbit. This can be achieved by starting the TOI burn on either the ascending or descending node.

Updating the Propagate segment's properties

Rename the Propagate segment Prop to TOI to make its purpose in the MCS more clear.

1. Return to GEO_Sat's ( ) Properties ( ).
2. Right-click on Propagate ( ) in the MCS.
3. Select Rename in the shortcut menu.
4. Rename Propagate ( ) Prop to TOI.

Updating the Stopping Condition

Stopping conditions are *Astrogator* components that are used by Propagate, Follow, and Finite Maneuver segments to define the point at which propagation should stop. After each step, the segment checks to see if any stopping conditions were met during the step. If so, it then finds the exact point, within tolerance, where the stopping condition is satisfied. From that point, the segment either executes an Automatic Sequence (if that option has been selected) or stops the propagation and passes the state at that point to the next segment. *Astrogator* also adds an ephemeris point at the time that the stopping condition is triggered. Update Prop to TOI's Stopping Condition to stop at the ascending node after two full revolutions in the parking orbit.

1. Click New... ( ) in the Stopping Conditions panel toolbar.
2. Select AscendingNode ( ) when the New Stopping Condition dialog box opens.
3. Click **OK** to confirm your selection and to close the New Stopping Condition dialog box.
4. Select Duration in the Stopping Conditions panel.
5. Click Delete ( ) on the Stopping Conditions toolbar.
6. Enter 2 in the Repeat Count field.

   This will end the Propagate segment on the second ascending node.
7. Click **Apply** to confirm your changes and to keep the Properties Browser open.
8. Click Run Entire Mission Control Sequence ( ) on the MCS toolbar.

Defining a Target Sequence

You will use a Target Sequence to calculate the Delta-V required to move GEO_Sat into a transfer orbit. A Target Sequence acts as a structural element to define a maneuver and propagation in terms of the goals they are intended to achieve.

Inserting a new Target Sequence

Insert a new Target Sequence after the Propagate segment.

1. Return to GEO_Sat's ( ) Properties ( ).
2. Right-click on Prop To TOI ( ) in the MCS.
3. Select Insert After... in the shortcut menu.
4. Select Target Sequence ( ) when the Segment Selection dialog box opens.
5. Click **OK** to confirm your selection and to close the Segment Selection dialog box.
6. Rename Target Sequence ( ) Start Transfer.

Inserting a Maneuver segment

You can nest any MCS segment within a Target Sequence, including another Target Sequence. The segments within a Target Sequence are defined in the same manner as they are in the MCS itself. Add a Maneuver segment into your Target Sequence. For this first maneuver, Delta-V will be solved to achieve a particular altitude at the end of the transfer ellipse.

1. Right-click on Start Transfer ( ) in the MCS.
2. Select Insert After... in the shortcut menu.
3. Select Maneuver ( ) when the Segment Selection dialog box opens.
4. Click **OK** to confirm your selection and to close the Segment Selection dialog box.

   The Maneuver ( ) Segment appears below the Return ( ) Segment.
5. Click and drag the Maneuver ( ) Segment below and nested inside Start Transfer ( ).
6. Note that the default Maneuver type is Impulsive.

For Impulsive maneuvers, *Astrogator* calculates the new state of the spacecraft by adding a Delta-V vector to the final state velocity of the previous segment. This new state is then added to the ephemeris and passed to the next segment.

Updating the Maneuver segment's properties

An Impulsive Maneuver ( ) segment isn't visible in the 2D and 3D Graphics windows. Propagate segment colors are visible. Change the Maneuver segment's color to white to focus on the Propagate segments.

1. Right-click on Maneuver ( ) in the MCS.
2. Select Properties ... in the shortcut menu.
3. Enter TOI in the Name field when the Edit Segment dialog box opens.
4. Open the Color drop-down list.
5. Select white.
6. Click OK to confirm your changes and to close the Edit Segment dialog box.

Inserting a new Propagate Segment

Insert a Propagate segment to model the movement of the spacecraft along its current trajectory until meeting specified stopping conditions. The segment uses the defined propagator and integrator to propagate the state, adding each point to the ephemeris as it goes. After each step, the segment checks to see if any stopping conditions were met during the step. If so, it then finds the exact point, within tolerance, where the stopping condition is satisfied. From that point, the segment either executes an automatic sequence or stops the propagation and passes the state at that point to the next segment. *Astrogator* also adds an ephemeris point at the time that the stopping condition is triggered.

1. Right-click on TOI ( ) in the MCS.
2. Select Insert After... in the shortcut menu.
3. Select Propagate ( ) when the Segment Select dialog box opens.
4. Click OK to confirm your selection and to close the Segment Select dialog box.

Updating the Propagate segment's properties

Update the name and color of the Propagate segment.

1. Right-click on Propagate ( ) in the MCS.
2. Select Properties ... in the shortcut menu.
3. Enter Transfer in the Name field when the Edit Segment dialog box opens.

4. Open the Color drop-down list.
5. Select yellow.
6. Click **OK** to confirm your changes and to close the Edit Segment dialog box.
7. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Selecting the engine model

The Engine tab on a Maneuver segment defines the magnitude and the nature of the propulsion. The engine parameters specified on this tab are used primarily to define the maneuver direction when using a thruster set to seed a finite maneuver and to update the fuel mass. You can use Engine Model to quickly model the firing of a single engine. Use the Test Engine model you updated earlier.

1. Select TOI ( ) in the MCS.
2. Select the Engine tab.
3. Click the Engine Model ellipsis (...) in the Propulsion Type panel.
4. Select Test Engine ( ) when the Select Component dialog box opens.
5. Click **OK** to confirm your selection and to close the Select Component dialog box.

Selecting the control parameter

The actions that the Target Sequence takes are determined by the profiles that are defined for it. A Search profile defines goals and changes variables to achieve them. A differential corrector search profile uses a differential correction algorithm to achieve a goal value or set of values. The values that the profile targets are called independent variables. The values that define the goal of the profile are called dependent variables. When the Target Sequence runs, it will change the values of the independent variables to achieve the goal. You can use any element of a nested MCS segment or linked component as an independent variable. Select the Maneuver segment's Delta-V Magnitude as the independent variable for the Target Sequence.

1. Select the Attitude tab.
2. Note that the default Attitude Control is Along Velocity Vector.

   Attitude Control specifies the mode in which the maneuver pointing direction is prescribed. With the Along Velocity Vector selection, the satellite object’s attitude is such that the Delta-V vector aligns with the spacecraft's inertial velocity vector.

3. Select the Delta-V Magnitude target ( ).

Notice the target now has a check mark ( ). Selecting the target makes Delta-V Magnitude an independent variable. This tells *Astrogator* to determine the Delta-V magnitude based on user determined results. You will set up those results in an upcoming section.

4. Click **Apply** to confirm your changes and to keep the Properties Browser open.

> **Further Reading:** For more information about differential correctors, refer to the *Astrogator* technical notes.

Propagating to apoapsis

Update the Transfer Propagate segment's Stopping Condition to stop at apoapsis.

1. Select Transfer ( ) in the MCS.
2. Click New... ( ) in the Stopping Conditions panel toolbar.
3. Select Apoapsis ( ) when the New Stopping Condition dialog box opens.
4. Click **OK** to confirm your selection and to close the New Stopping Condition dialog box.
5. Select Duration in the Stopping Conditions panel.
6. Click Delete ( ) on the Stopping Conditions toolbar.
7. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Selecting the results variable

Dependent variables for a differential corrector profile are defined in terms of calculation objects. Use a calculation object for the radius of the orbit, R Mag, to be used as the dependent variable.

1. Select Transfer ( ) in the MCS.
2. Click **Results...** beneath the MCS Tree.
3. Expand (⊞) the Spherical Elems ( ) folder in the Available Components list when the User - Selected Results - Transfer dialog box opens.
4. Select R Mag ( ).
5. Click Insert Component ( ) to move R Mag ( ) to the selected components list.

This will enable you to set the radius of orbit at the end of the Propagate Segment.

6. Click **OK** to confirm your selection and to close the User - Selected Results - Transfer dialog box.

Setting up the differential corrector profile

You can configure a Target Sequence to execute in many different ways depending on the solution you are trying to achieve. Set up the differential corrector profile to change the Delta-V Magnitude to achieve a desired radius of orbit.

1. Select Start Transfer ( ) in the MCS.
2. Open the Action drop-down list.
3. Select Run active profiles.

Selecting run active profiles runs the Mission Control Sequence allowing the active profiles to operate.

Setting the control parameter

Use the Delta-V Magnitude as the control parameter (independent variable) in the Target Sequence.

1. Select Differential Corrector in the Profiles panel.
2. Click Properties... ( ) on the Profiles toolbar.
3. Ensure the Variables tab is selected when the Differential Corrector dialog box opens.
4. Select the Use check box for ImpulsiveMnvr.Pointing.Spherical.Magnitude in the Control Parameters panel.

Each independent variable that you selected is listed in the Control Parameters section of the Variables tab.

Setting the equality constraints (Results)

Set R Mag as the equality constraint (dependent variable) with a goal of 42,238 km. *Astrogator* will use R Mag to determine the required Delta-V magnitude.

1. Select the Use check box for R_Mag in the Equality Constraints (Results) panel.

   Each dependent variable that you selected is listed in the Equality Constraints (Results) section of the Variables tab.

2. Click on the Desired Value cell.
3. Enter 42238 km in the Desired Value cell.

The value that you want to achieve.

4. Click **OK** to confirm your changes and to close the Differential Corrector dialog box.
5. Click **Apply** to confirm your changes and to keep the Properties Browser open.
6. Save (![save icon]) your scenario.

Running the Entire Mission Control Sequence

With your Target Sequence configured, run the entire Mission Control Sequence.

1. Click Run Entire Mission Control Sequence (![run icon]) on the MCS toolbar.
2. When complete, look at the top of the STK application.

   During the execution of one or more Target Sequences, Target Status windows appear and provide valuable information on their progress, including a message informing you whether or not running the entire MCS converged or didn't converge. You can close these windows individually, or you can clear them all at once by left-clicking the target icon (![target icon]) in the upper-left corner of one of the windows and selecting "Close All Targeting Status Windows."
3. Look at the StartTransfer.Differential Corrector target status window.

   This shows you data based on running the active profile.
4. Close the StartTransfer.Differential Corrector target status window.
5. Bring the 3D Graphics window to the front.

You can see the iterations, the last one placing the satellite at the desired location and altitude.

Designing the synchronized orbit injection (SOI)

Now that the transfer has been analyzed, you will follow a similar process to create the SOI maneuver and circularize the orbit at GEO. At the same time, you will bring the inclination to 2 deg. You will target the inclination slightly above the desired inclination, so that it will drift down to the desired inclination (0 to 1 degree) over time.

Inserting another Target Sequence

Create another Target Sequence to create the SOI and circularize the orbit at GEO.

1. Return to GEO_Sat's ( ) Properties ( ).
2. Right-click on the bottom Return ( ) Segment in the MCS.
3. Select Insert Before... in the shortcut menu.
4. Select Target Sequence ( ) when the Segment Selection dialog box opens.
5. Click OK to confirm your selection and to close the Segment Selection dialog box.
6. Rename Target Sequence ( ) Finish Transfer.

Inserting a Maneuver segment

Insert a Maneuver ( ) segment. You will solve for the X (Velocity) & Y (Normal) components and the Delta-V vector in those references axes needed to achieve a circular orbit at GEO.

1. Right-click on Finish Transfer ( ) in the MCS.
2. Select Insert After... in the shortcut menu.
3. Select Maneuver ( ) when the Segment Selection dialog box opens.
4. Click OK to confirm your selection and to close the Segment Selection dialog box.

   The Maneuver ( ) Segment appears below the return ( ) Segment.
5. Click and drag the Maneuver ( ) Segment below and nested in Finish Transfer ( ).

Updating the Maneuver segment's properties

As before, change the Maneuver segment's name and its color to white.

1. Right-click on Maneuver ( ) in the MCS.
2. Select Properties ... in the shortcut menu.
3. Enter SOI in the Name field when the Edit Segment dialog box opens.
4. Open the Color drop-down list.
5. Select white.
6. Click OK to confirm your changes and to close the Edit Segment dialog box.

Inserting a Propagate segment

Insert a Propagate Segment, which will be used to determine the stopping condition.

1. Right-click on SOI ( ) in the MCS.
2. Select Insert After... in the shortcut menu.
3. Select Propagate ( ) when the Segment Select dialog box opens.
4. Click **OK** to confirm your selection and to close the Segment Select dialog box.

Updating the Propagate segment's properties

Edit the Propagate segment's name and color.

1. Right-click on Propagate ( ) in the MCS.
2. Select Properties ... in the shortcut menu.
3. Enter Prop 1 Rev in the Name field when the Edit Segment dialog box opens.
4. Open the Color drop-down list.
5. Select blue.
6. Click **OK** to confirm your changes and to close the Edit Segment dialog box.
7. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Setting the engine model

Use the Test Engine model you created earlier.

1. Select SOI ( ) in the MCS
2. Select the Engine tab.
3. Click the Engine Model ellipsis (...) in the Propulsion Type panel.
4. Select Test Engine ( ) when the Select Component dialog box opens.
5. Click **OK** to confirm your selection and to close the Select Component dialog box.

Selecting the control parameter

Use Thrust Vector as the attitude control setting. Then select the Delta-V vector's Cartesian X (Velocity) and Y (Normal) as the independent variables.

1. Select the Attitude tab.
2. Open Attitude Control drop-down list.
3. Select Thrust Vector.

   The thrust vector describes the direction of acceleration applied to a satellite. This direction is opposite to the exhaust of an engine. For example, for a single chemical rocket engine mounted to a satellite, the thrust vector is opposite to the direction of the center of the exhaust plume flames. If the satellite uses more than one engine together in a thruster set, the thrust vector is along the direction of the combined effective acceleration. This direction is determined by calculating the sum of the acceleration vectors of each individual thruster. With this attitude control setting, you specify the Delta-V vector in some reference frame using either Cartesian or spherical components. *Astrogator* then computes the attitude so that the total thrust vector in the body frame, as specified by the thruster set or engine model, aligns with this vector in the reference axes.

4. Select the X (Velocity) target ( ).
5. Select the Y (Normal) target ( ).
6. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Propagating for one day

Update the Prop 1 Rev Propagate segment's Duration Stopping Condition to stop after one day.

1. Select Prop 1 Rev ( ) in the MCS.
2. Enter 1 day in the Trip field in the Stopping Conditions panel.
3. Click **Apply** to confirm your change and to keep the Properties Browser open.

Selecting the results variable

Set Eccentricity and Inclination to be used as the equality constraints.

1. Ensure Prop 1 Rev ( ) is selected in the MCS.
2. Click **Results...** at the bottom of the MCS.

3. Expand (⊞) the Keplerian Elems ( ) folder in the Available Components list when the User - Selected Results - Prop 1 Rev dialog box opens.
4. Insert ( ) the Eccentricity ( ) component on the selected components list.
5. Insert ( ) the Inclination ( ) component on the selected components list.
6. Click **OK** to confirm your selections and close the User - Selected Results - Prop 1 Rev dialog box.
7. Click **Apply** to confirm your changes and to keep the Properties Browser open.

Setting up the differential corrector profile

Set up the differential corrector profile to change the Delta-V Magnitude to achieve a desired radius of orbit.

1. Select Finish Transfer ( ) in the MCS.
2. Open the Action drop-down list.
3. Select Run active profiles.

Setting the control parameters

Use the Delta-V vector's Cartesian X (Velocity) and Y (Normal) as the Control Parameters (independent variables).

1. Select Differential Corrector in the Profiles panel.
2. Click Properties... ( ) on the Profiles toolbar.
3. Select the Use check box for ImpulsiveMnvr.Pointing.Cartesian.X in the Control Parameters panel when the Differential Corrector dialog box opens.
4. Select the Use check box for ImpulsiveMnvr.Pointing.Cartesian.Y.

Setting the equality constraints (Results)

Set Eccentricity and Inclination as the Equality Constraints (dependent variables). Target an eccentricity of 0 within a 0.001 tolerance, and a 2 degree inclination within 0.01 degree of tolerance.

1. Select the Use check box for Eccentricity in the Equality Constraints (Results) panel.
2. Enter 0.001 in the Tolerance field.

   The profile will stop when it achieves a value within this range of the Desired Value.

3. Select the Use check box for Inclination.
4. Select the Desired Value cell.
5. Enter 2 deg in the Desired Value cell.
6. Enter 0.01 deg in the Tolerance field.
7. Open the Method drop-down list in the Scaling panel.

   The scaling method improves numerical behavior if the enabled parameters have diverse magnitudes.
8. Select By tolerance.
9. Click OK to confirm your changes and to close the Differential Corrector dialog box.
10. Click Apply to confirm your changes and to keep the Properties Browser open.

Running the entire Mission Control Sequence

With your SOI Target Sequence configured, run the entire MCS.

1. Click Run Entire Mission Control Sequence ( ) on the MCS toolbar.
2. Look at the Finish Transfer.Differential Corrector: Finished data window.
3. Close both the Start Transfer.Differential Corrector and Transfer.Differential Corrector Targeting Status windows.
4. Bring the 3D Graphics window to the front.

   You can see the iterations, the last one placing the satellite at the desired eccentricity and inclination.

Removing the iterations

Remove the visible iterations in the 3D Graphics window by adding a final Propagate segment.

1. Return to GEO_Sat's ( ) Properties ( ).
2. Right-click on the Bottom Return ( ) Segment.
3. Select Insert Before... in the shortcut menu.
4. Select Propagate ( ) when the Segment Selection dialog box opens.

5. Click OK to confirm your selection and to close the Segment Selection dialog box.
6. Click Run Entire Mission Control Sequence ( ) on the MCS toolbar.
7. Close both the Start Transfer.Differential Corrector and Transfer.Differential Corrector Targeting Status windows.
8. Bring the 3D Graphics window to the front.

Generating an MCS Segment Summary report

In addition to the reports that can be created using the Report & Graph Manager available for any STK object, *Astrogator* can generate three special reports that provide information about the execution of MCS segments – the MCS Segment Summary report, the Astrogator Log File report, and the Differential Corrector Log report. The MCS Segment Summary report provides run summary data for the currently selected segment in the MCS tree. The summary report gives information on maneuver times, expected Delta-V magnitude, estimated burn duration, estimated fuel usage, and other important data.

1. Return to GEO_Sat's ( ) Properties ( ).
2. Select Start Transfer ( ) in the MCS.
3. Click Summary ( ) on the MCS toolbar.
4. Look at the data in the report when the report opens.
5. When finished, close ( ) the summary report.
6. Select Finish Transfer ( ).
7. Click Summary ( ) on the MCS toolbar.
8. Look at the data in the report when the report opens.

Summary

This was a basic introduction to *Astrogator*. In this lesson, you did the following:

- Used a Launch Vehicle object to launch a vehicle that followed an ascent trajectory from a launch point to an orbit insertion point

- Inserted a Satellite object, switched the propagator to *Astrogator* and used a Follow Segment. This allowed the Satellite to follow the Launch Vehicle object, to separate from the Launch Vehicle at the end of its trajectory, and to place the Satellite object into a LEO.
- Used a Target sequence, a Maneuver segment, and a Propagate segment to place the Satellite orbit into a TOI
- Finalized its orbit by creating another Target Sequence that placed the Satellite into a GEO
- Used an MCS Segment Summary report to determine maneuver times, required Delta-V, estimated burn duration, and estimated fuel usage

Part 18: Ground-based SSA with EOIR

> **Note:** Visit **help.agi.com/stk/#training/Day2Overview.htm** for an extended version of this lesson. There, you can view reference images, access extra content, and follow a recording of an instructor completing this lesson.

> **Note:** The results of the tutorial may vary depending on the user settings and data enabled (online operations, terrain server, dynamic Earth data, etc.). It is acceptable to have different results.

Problem statement

Engineers and operators require a fast and easy way to model and simulate detection, tracking, and imaging performance of electro-optical and infrared sensors. You want to simulate tracking a polar satellite that is in a low Earth orbit (LEO) from an observatory located in Hawaii. You want to model the facility's telescope from specifications and take atmospheric effects, temperature, emissivity, and radiance into consideration for your analysis.

Solution

Use the STK application and the *Electro-Optical Infrared Sensor Performance (EOIR)* capability to model, simulate, and analyze a synthetic sensor scene from the 1.6-meter Advanced Electro-Optical System (AEOS) telescope at the Air Force Maui Optical Station (AMOS) observatory located at the Maui Space Surveillance Complex (MSSC) in Maui, Hawaii, that is tracking a polar satellite in LEO.

Creating a new scenario

First, you must create a new STK scenario, and then build from there.

1. Launch the STK application ().
2. Click Create a Scenario in the Welcome to STK dialog box.
3. Enter the following in the STK: New Scenario Wizard:

Option	Value

Name	STK_EOIR
Location	Default
Start	1 Aug 2024 15:00:00.000 UTCG
Stop	+ 10 min

4. Click **OK** when you finish.
5. Click Save ( ) when the scenario loads. The STK application creates a folder with the same name as your scenario for you.
6. Verify the scenario name and location in the Save As window.
7. Click **Save** .

Modeling the AEOS telescope

Add the AEOS 1.6-meter diameter telescope to your scenario.

Inserting a Facility object

Use the Insert STK Objects tool to insert a Facility from the Standard Object Database.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Select Facility ( ) in the Select An Object To Be Inserted list.
3. Select the From Standard Object Database ( ) method.
4. Click **Insert. . .** .

Selecting the MSSC 1.6-meter telescope for the Facility object

Search for select the MSSC 1.6-meter telescope to use for the Facility object.

1. If you have online operations enabled, clear the Online check box in the Data Sources.
2. This will ensure you only select the MSSC facility available in your local object database.

   Enter MSSC in the Name field in the Search Standard Object Data dialog box.

3. Click **Search** .
4. Select MSSC 1.6m in the Results list that uses the Local Database Data Source.
5. Click **Insert** .
6. Click **Close** to close the Search Standard Object Data dialog box.

Inserting a Sensor object

Add a sensor to MSSC_1_6m Facility object to model the 1.6-meter AEOS telescope.

1. Insert a Sensor ( ) object using the Insert Default ( ) method.
2. Select MSSC_1_6m ( ) in the Select Object dialog box.
3. Click **OK** .
4. Right-click on Sensor1 ( ) in the Object Browser.
5. Select Rename in the shortcut menu.
6. Rename Sensor1 ( ) Telescope.

Creating an EOIR sensor

The *Electro-Optical Infrared Sensor Performance (EOIR)* capability models the detection, tracking, and imaging performance of electro-optical and infrared sensors. You can use the *EOIR* capability to support concept development, design, field-testing, and operations. The STK software takes this modeling to the big picture by considering electro-optical and infrared performance in conjunction with sensor platform dynamics, communications, and other mission architecture elements to assess integrated performance. The *EOIR* capability has the ability generate performance metrics such as signal-to-noise ratio and sensor images with both Earth and space in the background.

Selecting the EOIR sensor type

Start by selecting the EOIR sensor type on the sensor's Definition page. The EOIR sensor type is used to model electro-optical and infrared sensors.

1. Right-click on Telescope ( ) in the Object Browser.
2. Select Properties ( ) in the shortcut menu.

3. Select the Basic - Definition page when the Properties Browser opens.
4. Select EOIR in the Sensor Type drop-down list.
5. Click **Apply** to accept your change and to keep the Properties Browser open.

Setting the sensor's spatial properties

When you choose EOIR as the sensor yype, you can select the Spatial tab to specify its spatial properties. The default input setting is Field-of-View and Number of Pixels. Update these spatial properties to define the total field-of-view angles to more accurately model the AEOS telescope.

1. Ensure the Spatial tab is selected on the Basic - Definition page.
2. Set the following parameters in the Field of View panel:

Option	Value
Horizontal Half Angle	7.5 deg
Vertical Half Angle	7.5 deg

   The *EOIR* capability uses these half angles to determine the full angular extent of the sensor field of view (FOV).

3. Review the settings in the Number of Pixels panel.

   These are the number of sensor pixels in the field of view in the horizontal and vertical directions. The default value is 128 for each direction.

4. Click **Apply** .

> **Note:** The Related Detector Parameters and Instantaneous Field of View values are based on the sensor spatial and optical properties. These are read-only fields and are automatically updated when you apply your changes.

Setting the sensor's spectral properties

The AEOS telescope observes the long infrared waveband. Specify the sensor's spectral properties to model this spectral band range. The sensor model samples your spectral band using the number of intervals you define. The more intervals you have, the higher the accuracy of the analysis. However, more intervals mean longer computation times.

1. Select the Spectral tab on the Basic - Definition page.
2. Set the following parameters (in micrometers) in the Spectral Band Edge Wavelengths panel:

> **Important:** You must set the High value first.

Option	Value
High	1.0
Low	0.7

3. Leave the Number Of Intervals set at 6.000000.
4. Leave the Spectral Shape set to the Use Optical and Radiometric Response option.

   This defaults the spectral shape to the individual optical transmission and quantum efficiency spectral characteristics.
5. Click Apply .

Setting the sensor's optical properties

Next, set the sensor's optical properties. You will specify the F Number (Effective Focal Length / Effective Pupil Diameter) and the diameter (in cm) of the "single lens equivalent" optical prescription.

1. Select the Optical tab on the Basic - Definition page.
2. Select F-Number and Entrance Pupil Diameter for the Input.
3. Set the following parameters:

Option	Value
F/#	200
Entrance Pupil Diameter	367.00

4. Select Negligible Aberrations in the Image Quality drop-down list.

   The *EOIR* capability models aberrations based on a root-mean-square wavefront error. The Negligible Aberrations setting introduces a 7% wave front error.
5. Click Apply .

Setting the sensor's radiometric properties

You can populate and edit sensor performance data with measurements from an actual sensor. The sensor's radiometric properties define the noise floor and the saturation ceiling. When preparing to take measurements with the sensor model, you specify an integration time. This is the time interval over which a radiant signal is collected before generating an image. The longer the time, the more photons that get collected. This field is equivalent to the "exposure time" setting on an analog film camera.

You can also define a set of points that relate Integration (Exposure) Time to NEI/SEI (noise equivalent irradiance / saturation equivalent irradiance). The STK software linearly interpolates between the points to get correct NEI/SEI for the integration time you set.

1. Select the Radiometric tab on the Basic - Definition page.
2. Set the following options in the Sensitivity panel:

Option	Value
Integration Time	100
Equivalent Value	1e-16

The Sensitivity defines the "noise floor" of the sensor. The sensor will not detect signals below this level.

3. Notice that Processing Level defaults to Sensor Output.

Processing levels enable you to visualize the geometric information in the sensor scene or the sensor output image. The Radiometric Input simulates the light entering the sensor lens before hitting the sensor detector when generating the EOIR sensor scene.

4. Enter 1 in the Line of Sight Jitter field in the Jitter panel.

This introduces a Gaussian vibration of one milliradian along the sensor boresight.

5. Click **OK** to apply your changes and to close the Properties Browser

---

Opening the EOIR toolbar

Before you can use the *EOIR* capability with your sensor, you must first display the EOIR toolbar. You can use the EOIR toolbar to access the EOIR configuration and EOIR sensor scene for a particular sensor.

1. Select View in the Menu Bar.
2. Select Toolbars in the View menu.
3. Select EOIR in the Toolbar submenu to show the EOIR toolbar.

Viewing the EOIR Configuration

View the sensor's EOIR Configuration. The EOIR Line of Sight and Field of View are synchronized to the STK Sensor object.

1. Click EOIR Configuration... ( ) in the EOIR toolbar.

   This displays the sensor's EOIR Configuration dialog box. All central bodies and objects, except for the source sensor, that are part of the EOIR Configuration are listed in the available target list.

2. Click **OK** to close the EOIR Configuration dialog box.

Generating an EOIR sensor scene

Now you are ready to generate an EOIR sensor scene. The EOIR Line of Sight and Field of View are synchronized to the STK Sensor object.

1. Select Telescope ( ) in the Object Browser.

2. Click EOIR Sensor Scene... ( ) in the EOIR toolbar.

   This generates an image that represents the radiometric input to the sensor. You will see some white dots and gray dots against a black background.

   The input scene represents the analog world by digitally sampling the "modeled universe" at 4 times the sensor's pixel's spatial frequency, 16 spatial samples per sensor pixel, and over the passband and at wavelengths defined by the sensor model. The input scene is thus a "box" of floating point numbers of dimension (Horizontal spatial resolution × Vertical spatial resolution × Spectral resolution). These scenes accurately portray sensor images for the processing level selected.

3. Right-click on the sensor scene.

4. Select Details... in the shortcut menu.

5. Move the EOIR Scene Visual Details dialog box so that it's not sitting on top of the sensor scene.

   You can use the EOIR Scene Visual Details dialog box to set the color map, determine the resolution of the Earth map using the scene detail box, adjust the brightness and contrast, change the file output settings and return back information on the pixel clicked inside of the sensor scene.

6. Select the BGRY option in the Color Map panel.

   EOIR uses false color to bring out details in the image data that are often lost when displayed on a monitor with less resolution than the *EOIR* sensor. For instance, a typical monitor can display only 256 levels of grayscale, whereas an *EOIR* sensor might have 4096 levels of grayscale resolution. Color mapping is only for visual effect and does not change any of the internal data values.

7. Click **Apply** .

   For the Sensor Output processing level, the raw sensor data and image can be saved out at every animation step. You can save the data in each sensor click to a file by selecting Pixel Spectral Data on the EOIR Scene Visual Details page. You can then compound these images to create a movie or run through external image processing software for further analysis.

8. Click around the scene to display information on the EOIR Scene Visual Details window for each pixel.
9. Click one of the stars to get more details on this object.
10. Close the EOIR Scene Visual Details dialog box and the EOIR Sensor Scene window when finished.

Inserting a Satellite object

Insert a Satellite object using the Orbit Wizard to model the polar satellite in LEO that you are tracking.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Insert a Satellite ( ) object using the Orbit Wizard ( ) method.
3. Set the following options in the Orbit Wizard:

Option	Value
Type	Circular
Satellite Name	LEO_Sat
Inclination	98 deg
Altitude	700 km
RAAN	24 deg

4. Click OK to accept your changes and to close the Orbit Wizard.

The orbit will take the satellite through the telescope's field of view.

Viewing LEO_Sat and the ground site in the 3D Graphics window

View the LEO satellite and the Facility in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on LEO_Sat ( ) in the Object Browser.
3. Select Zoom To.
4. Pan and zoom around so that you can view both LEO_Sat ( ) and MSSC_1_6m ( ).

5. Click Decrease Time Step ( ) in the Animation toolbar until the Time Step is set to 1 sec.
6. Click Start ( ) to animate the scenario.
7. Click Reset ( ) when finished.

> **Note:** Although this tutorial is a ground-to-space example, it is possible to host an EOIR sensor on both air and space vehicles. The work flow of setting up an *EOIR* sensor model is similar for all supported STK objects.

Viewing LEO_Sat in the EOIR sensor scene

An EOIR sensor will take images of objects that fall within the sensor's field of view that hey are sufficiently bright, either in reflected light or from self-radiance, at wavelengths that the sensor can detect. When LEO_Sat passes over the AMOS facility, AMOS is in darkness while the satellite is illuminated. This scenario gives good lighting conditions for imaging.

Viewing LEO_Sat's basic EOIR shape

With the *EOIR* capability, you can set image generation properties for selected STK objects. When you open an properties, you will see EOIR Shape as an option under its Basic properties. Examine the material and shape properties of LEO_Sat.

1. Open LEO_Sat's ( ) Properties ( ).
2. Select the Basic - EOIR Shape page when the Properties Browser opens.
3. Examine the following options:
    - Shape
    - Radius
    - Body Temperature
    - Temperature
    - Material
4. Keep the default settings.
5. Click Cancel to close the Properties Browser.

Adding LEO_Sat to the EOIR configuration

To see the LEO_Sat in the EOIR sensor scene, you must first add it as a target in the EOIR configuration.

1. Click EOIR Configuration... ( ) in the EOIR toolbar to open the EOIR Configuration dialog box.
2. Double-click on Satellite/LEO_Sat ( ) in the Available STK Objects list to move it to the Selected Targets list.
3. Click OK to close the EOIR Configuration dialog box.

Creating an Access between Telescope and LEO_Sat

Create an access report between Telescope and LEO_Sat. You will use the Access for further analysis.

1. Right-click on Telescope ( ) in the Object Browser.
2. Select Access... ( ).
3. Select LEO_Sat ( ) in the Associated Objects list.
4. Click Compute .

Setting animation time from an Access report

Generate an Access report. You will use the time of the first access to set a viewing time for your sensor scene.

1. Click Access... in the Reports panel.
2. Right-click on the first access start time in the Access report.
3. Select Start Time in the shortcut menu.
4. Select Set Animation Time in the Start Time submenu.

   This sets the Current Scenario Time in the Animation toolbar to the time when LEO_Sat first enters the telescope's field of view.
5. Close ( ) the access report.
6. Close ( ) the Access tool.

Creating the EOIR Sensor Scene

Generate the EOIR scenario scene.

1. Select Telescope ( ) in the Object Browser.
2. Click EOIR Sensor Scene... ( ) in the EOIR toolbar.
3. Right-click on the sensor scene.
4. Select Details... in the shortcut menu to open the EOIR Scene Visual Details dialog box.
5. Select the Gray Scale option in the Color Map panel.
6. Click **Apply** .

Note that the AGC check box is selected. AGC is Automatic Gain Control. When this option is selected, the *EOIR* capability automatically calculates brightness and contrast such that the brightest scene detail fits within the brightness resolution of the monitor.

Performing EOIR sensor scene analysis

View the EOIR sensor scene details.

1. Decrease ( ) the animation Time Step to 0.5 seconds.
2. Step Forward ( ) the scenario until you see the satellite come into the scene.

   The dot that represents the satellite moves across the scene while the stars stay relatively still.
3. Click one of the stars and the target satellite to get more details on those objects.
4. Click **Close** to close the EOIR Scene Visual Details dialog box when finished.
5. Close ( ) the EOIR sensor scene.

Creating custom graphs for EOIR sensors

The *EOIR* capability does more than simulate scenes created by an EOIR sensor. It can also calculate metrics a sensor would receive from a target's signal. The following will familiarize you with some of the available EOIR data providers.

Creating a new graph

First, create a new graph called Target Metrics.

1. Right-click on Telescope ( ) in the Object Browser.
2. Select Report & Graph Manager... in the shortcut menu ( ) to open the Report & Graph Manager.
3. Select the My Styles ( ) folder in the Styles panel.
4. Click Create new graph style ( ) in the Styles toolbar.
5. Enter Sensor to Target Metrics.
6. Select the Enter key to rename the graph and to open the graph's properties.

Setting the graph's data providers

You will use the EOIR Sensor To Target Metrics data provider and the In-band target irradiance and Signal to noise ratio elements.

- EOIR Sensor To Target Metrics: time dependent metrics for a unique EOIR Sensor-Band / Target pairing.
- In-band target irradiance: the irradiance at the sensor aperture from a target object whose angular extent is smaller than the effective instantaneous field of view, i.e. a point source target
- Signal to noise ratio: is the ratio of the difference in sensor response between target-containing pixel (s) and the local surrounding pixels to the total noise. For point source targets the background is assumed to be uniform (spatial clutter is neglected) and the target is assumed to be exactly centered on a pixel.

1. Expand (⊞) the EOIR Sensor To Target Metrics ( ) data provider in the Data Provider list.
2. Move ( ) the In-band target irradiance ( ) data provider element to the Y Axis list.
3. Move ( ) the Signal to noise ratio ( ) data provider element to the Y2 Axis list.
4. Select EOIR Sensor to Target Metrics-In-band target irradiance in the Y Axis list
5. Click **Units...** below the Y2 Axis box to open the Units dialog box.
6. Clear the Use Defaults check box.
7. Select Power in the Dimension column.

8. Select Watts (W) in the New Unit Value list.
9. Click **OK** to close the Units dialog box.

Setting the step size

Set the graph's step size to one second.

1. Enter 1.0 sec in the Step Size field.
2. Click **OK** to accept your changes and to close the Properties Browser.

Setting the Time properties

Set the time properties so the data is reported over the first Access interval.

1. Return to the Report & Graph Manager.
2. Select Specify Time Properties in the Time Properties panel.
3. Open (▼) the Start and Stop times drop-down menu.
4. Select Interval Component... to open the Select Time Interval dialog box.
5. Select Facility-MSSC_1_6m-Sensor-Telescope-To-Satellite-LEO_Sat ( ) in the Objects list.
6. Expand (⊞) AccessIntervals ( ) in the Intervals for list.
7. Select First ( ).
8. Click **OK** to close the Select Time Interval dialog box.

This limits the analysis period to the interval when LEO_Sat is visible in the EOIR sensor scene and decreases the computation time needed for the analysis.

Generating the custom graph

Now, generate the custom graph over the first Access interval.

1. Select the Sensor to Target Metrics ( ) in the Styles list.
2. Click **Generate. . . .**
3. Look at the graph.

This graph shows the signal is small relative to the noise, however using gray scale color mapping, you are able to see the target in the scene. Keep the graph open.

Viewing the effects of EOIR atmosphere modeling

By default, the *EOIR* capability neither applies an atmosphere model nor an atmosphere parameter setting when generating a sensor scene. You will apply a Simple Atmosphere model and adjust its parameters to view the effects the atmosphere has on your data.

Selecting the Simple Atmosphere model

Set the *EOIR* atmosphere model.

1. Click EOIR Configuration... ( ) in the EOIR toolbar to open the EOIR Configuration dialog box.
2. Click **Atmosphere and Textures. . .** to open the EOIR Atmosphere, Clouds, and Texture Maps dialog box.
3. Take a minute to view the different atmosphere models:
   - Simple Atmosphere: This model calculates the atmospheric properties at the wavelengths corresponding to the Spectral Band Edges, and at a spectral resolution specified by the Number of Intervals set on the Sensor's Spectral Properties page. The Simple Atmosphere model only uses variations of atmospheric properties with altitude. It does not calculate the horizontal variations – which constitute weather – nor clouds.
   - MODTRAN Derived Lookup Table: MODTRAN is a community standard, and the MODTRAN Derived Lookup Table atmosphere model is one of the highest-fidelity atmospheric models available in EOIR.
4. Select the Simple Atmosphere option in the Modes panel.
5. Set the following parameters in the Parameters panel:

Option	Value
Aerosol Models	Maritime
Visibility	40
Humidity	70

   The Atmosphere Parameters provide some control over the atmosphere's physical characteristics. These characteristics include:

   - Aerosol Models: This selects the type of aerosol model for the Simple Atmosphere model to use. Aerosols are tiny particles in the air that cause whitish haze visible to the human eye. Each aerosol model comprises a distribution of particles of different sizes and how light interacts with them. For instance, in a Maritime atmosphere, salt crystals from wind and wave action are a major contributor to aerosols.

- Visibility: This specifies the meteorological visibility in kilometers. Meteorological visibility is the greatest distance at which a black object of suitable dimensions, located near the ground, can be seen and recognized when observed against a bright background.
- Humidity: This specifies the relative humidity as a percentage from 0.0 to 100.0.

6. Click **OK** to close the EOIR Atmosphere, Clouds, and Texture Maps dialog box.
7. Click **OK** to close the EOIR Configuration dialog box.

Refreshing the custom graph

Refresh the open graph to see the changes.

1. Return to the custom graph.
2. Click Refresh (F5) ( ) in the graph toolbar.

The degradation is due to atmospheric effects. Keep the graph open.

Turning off the atmosphere model

Now that you have seen the effects the atmosphere has on your data, turn the atmosphere off.

1. Click EOIR Configuration... ( ) in the EOIR toolbar to open the EOIR Configuration dialog box.
2. Click **Atmosphere and Textures. . .** to open the EOIR Atmosphere, Clouds, and Texture Maps dialog box.
3. Select the Atmosphere Off option in the Modes panel.
4. Click **OK** to close the EOIR Atmosphere, Clouds, and Texture Maps dialog box.
5. Click **OK** to close the EOIR Configuration dialog box.

Viewing the effects of a custom EOIR shape

Earlier, you viewed LEO_Sat's basic EOIR Shape properties. Now, update some of those properties to more closely model LEO communications satellite.

Redefining LEO_Sat's EOIR shape

Define the material and shape properties of the satellite.

1. Open the LEO_Sat's ( ) Properties ( ).
2. Select the Basic - EOIR Shape page.
3. Set the following options:

Option	Value
Shape	LEOComm
Body Temperature	Static
Temperature	400 K
Material	Aluminum MLI

> **Note:**
> - LEOComm: the shape is based on the 3D model iridium.glb
> - Static: the STK software applies this temperature to the entire shape. This then applies for the entire EOIR scene time period
> - Aluminum MLI: multi-layer insulation. Each surface material has optical properties used in the sensor scene calculation. One of these is reflectance. Light bouncing off a surface is selectively reflected by wavelength; i.e., some wavelengths are absorbed instead of reflected. In the visible wavelengths, this is what gives objects color. Each surface naterial has a table of its reflectance versus wavelength.

4. Click OK .

Regenerating the EOIR sensor scene

Regenerate the EOIR sensor scene with LEO_Sat's updated EOIR Shape properties.

1. Right-click at the beginning of the custom graph.
2. Select Set Animation Time.
3. Select Telescope ( ) in the Object Browser.
4. Click EOIR Sensor Scene... ( ) in the EOIR toolbar.

5. Right-click on the sensor scene.
6. Select Details... in the shortcut menu to open the EOIR Scene Visual Details dialog box.

Viewing LEO_Sat in the EOIR sensor scene

You can view how LEO_Sat now appears in the EOIR Sensor Scene window.

1. Step Forward ( ) to see the satellite move across the scene.
2. Click on the target satellite to view information about it.

   Note that LEO_Sat's updated EOIR Shape properties, including the Temperature and Material, now display in the Scene Pick Information panel.
3. Click **Close** to close the EOIR Scene Visual Details dialog box when finished.

Refreshing the custom graph

Refresh the custom graph to see how the updated properties affect it.

1. Return to the custom graph.
2. Click Refresh (F5) ( ) in the graph toolbar.

The curve is showing a single minimum rate in in-band target irradiance and SNR that coincides with the satellite passing near the facility's zenith.

Analyzing the light signature of a tumbling satellite

In your previous analysis, the satellite was holding a nadir-pointing attitude profile. Because of this, the EOIR sensor sees a near-constant satellite cross-section during the overhead pass. Now, you will analyze the light signature of a satellite that is tumbling through space.

Updating the satellite's Attitude properties

Update LEO_Sat's attitude profile to be that of a precessing spin.

1. Open the LEO_Sat's ( ) Properties ( ).
2. Select the Basic - Attitude page.

3. Set the following options:

Option	Value
Type	Precessing Spin
Body Spin Axis	Type: Cartesian X: 0 Y: 1 Z: 0
Precession - Rate	30 revs/min
Spin - Rate	30 revs/min

4. Click **OK** to accept your changes and to close the Properties Browser.

Viewing LEO_Sat in the 3D Graphics window

View LEO_Sat spinning in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom To LEO_Sat ( ).
3. Decrease ( ) the animation Time Step to 0.1 seconds.
4. Step Forward ( ) to see the satellite tumble.

Updating the custom graph

Refresh your custom graph and update its step size to better show the tumbling satellite.

1. Return to your custom graph.
2. Click Refresh (F5) ( ) in the graph toolbar.
3. Enter 0.1 sec in the Step field.
4. Select the Enter key.

As the spacecraft rotates, various panels reflect varying amounts of light. The pot is jagged, thus confirming that the spacecraft is tumbling.

Summary

This was an introduction to the STK software's *EOIR* capability. You modeled, simulated, and analyzed the MSSC's 1.6-meter AEOS telescope at the AMOS observatory in Maui, Hawaii, that tracked a polar satellite in LEO. You set your Sensor object attached to the ground site to use the *EOIR* capability. You set its spatial, spectral, optical, and radiometric properties. Using the EOIR sensor scene, you obtained data on stars and the LEO satellite. You became familiar with satellite EOIR shape configurations and attitude issues. Using the EOIR configuration, you applied atmospheric changes to your analysis. You created a custom graph and graphed the various changes to your analysis.

Become Level 2 - STK Master Certified

Once you are Level 1 - STK Certified and have completed Comprehensive training (the Level 1 - Beginner and Level 2 - Advanced tutorials), you will be well prepared to complete the Level 2 - STK Master Certification test. The STK Master Certification is the second level of certification and validates your ability to perform more advanced modeling and analysis using the Ansys Systems Tool Kit® (STK®) digital mission engineering software through use of the STK software's *Analysis Workbench*,*STK/Astogator®*, *Aviator*, *Communications*, *Coverage*, and *Radar* capabilities.

What's in the test?

The STK Master Certification test consists of several scenario development exercises. There are multiple-choice questions for each exercise. You have 30 days from the registration date to complete the STK Master Certification test. The following objectives are tested:

- Modeling your systems – Advanced Aircraft (*Aviator*), Advanced Satellite (*Astrogator*), Missile, Sensor, Constellation, Chain, Advanced Constraints, Terrain, *Communications*, *Radar*, RF Environment models, STK External Propagator, Vehicle Attitude
- Analyzing your systems – Access tool, Report & Graph Manager, Custom Reports, *Coverage*, Figure of Merit, *Analysis Workbench* (Vector Geometry tool)

If you pass your STK Master Certification test, you will receive an STK Master Certification certificate.