STK Level 1 and Level 2 Training Manual

STK VERSION 12.10, MAY, 2025

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

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

Setting up a new STK scenario

Create a new STK Scenario ( ) object using the STK: New Scenario Wizard tool. The 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.
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 (e.g. C:\Users\username\Documents\STK 12).
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.

1. Click Save () when the scenario loads. STK creates a folder with the same name as your scenario for you.
2. Verify the scenario name and location in the Save As window.
3. Click **Save** .
4. Click **Close** in the Insert STK Objects tool.

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, practice workspace customization.

1. Extend the View menu.
2. Hover over Toolbars to extend the menu. This is one way to choose the toolbars you require.
3. Right-click in the empty space to the right of the currently available toolbars. This is another way to quickly enable and disable Menu selections and toolbars.

Understanding the 2D Graphics window

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.

Grab Globe

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

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

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 Graphics Defaults toolbar.
2. Place your cursor on the 2D Graphics window map.
3. Hold down the left mouse button.
4. Drag your mouse cursor across the area of the 2D map window you want to zoom. This draws a box around the area of interest.
5. Release the left mouse button.

Zoom Out

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

1. Click Zoom Out ( ) in the 2D Graphics Defaults toolbar to zoom out of the 2D window one zoom level.
2. You can continue to click Zoom Out ( ) until the 2D map fills the display window.

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, hitting 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 on the 2D Graphics window map.
3. Move the mouse scroll wheel down to zoom out on the 2D Graphics window map.

Microsoft Bing™ Maps

The Microsoft Bing™ Maps tool displays a background image (Microsoft Bing™ map or Basic.bmp) in 2D Graphics windows.

1. Click Microsoft Bing™ Map ().
2. Try different map styles.
3. When finished, reset the map style to Aerial.

Understanding the 3D Graphics window

STK 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. Like you did with the 2D Graphics window, you'll practice using selected tools.

Using the Mouse

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 the left mouse button 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.

3D Graphics window default tools

You can use the mouse to interact with the 3D Graphics window. Most of the tools have functions similar to the 2D Graphics 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.

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 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 Graphics window Defaults toolbar.
4. Hold down the left mouse button and drag your mouse 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.

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 Graphics 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 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.

Home View

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.

Animation toolbar

The animation tool bar 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.

Timeline View

Use 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.

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 rest the view.
2. Click Zoom Out ( ) in the 2D Graphics 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.
6. Click Save ( ).

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 STK.

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.

AREA TARGET LOCAL 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 .

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, sun light or umbra restrictions, gimbal speed, range, and more. Engineers also require the ability to create reports and graphs that summarize static data or show dynamic data during animation.

Solution

With STK, you can determine accesses between objects and generate reports to summarize your data. Building on your fundamental understanding of STK, use two important tools in STK to solve this problem:

- The Access Tool
- The Report & Graph Manager

Inserting the satellite tracking station

A teleport is located in Castle Rock, Colorado. Enter the teleport into the scenario as a Facility ( ) object.

1. Select Facility ( ) in the Insert STK Objects tool.
2. Select the From Standard Object Database ( ) method.
3. Click Insert...

4. Type castle rock in the Name: field when the Search Standard Object Data dialog box opens.
5. Click Search .
6. Select Castle Rock Teleport using the INTELSAT Network selection.
7. Click Insert .
8. Click Close to close the Search Standard Object Data dialog box.

Viewing the tracking station in 3D

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

Streaming terrain from a Terrain Server

For the purposes of this scenario, disable Terrain Server.

1. Right-click on AccessReportsGraphs ( ) 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 in the Terrain Server frame.
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 ( ) appears to be floating. STK is still referencing the altitude of Castle_Rock_Teleport ( ) based on terrain data.

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.
4. Click Apply to accept your changes and to keep the Properties Browser open.
5. Return to the 3D Graphics window.

Restricting the field-of-view of Castle Rock Teleport

You can restrict Castle_Rock_Teleport's () field-of-view by setting constraints, or you can restrict and visualize its field-of-view by using a Sensor () object.

1. Return to Castle_Rock_Teleport's () properties ().
2. Select the Constraints - Active page.
3. Click OK to close the Properties Browser.

Inserting a Sensor object onto Castle Rock Teleport

Use a Sensor () object that mimics Castle_Rock_Teleport's () antenna field-of-view.

1. Insert a Sensor () object using the Insert Default () method.
2. Select Castle_Rock_Teleport () in the Select Object dialog box.
3. Click OK .
4. Rename the Sensor1 () to CR_FOV. It's 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 constraints

1. Select the Constraints - Active page.
2. Click Add new constraints () in the Active Constraints toolbar.
3. Select Range 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. Select the Max: check box in the Range frame.
7. Enter 1000 km in the Max: field.
8. Click **OK** to accept your changes and to close the Properties Browser.

View the sensor in 3D

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 three Satellites objects

Propagate three operational CUBESAT satellites using the Standard Object Database tool.

1. Insert a Satellite () object using the From Standard Object Database () method.
2. Type cubesat in the Name or ID: field when the Search Standard Object Data dialog box opens.
3. Open the Operational Status: shortcut menu.
4. Select Operational.
5. Click **Search** .
6. Select all the operational satellites in the Results: list.
7. Click **Insert** .

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

Using the Access Tool

An access is defined by two or more objects - a primary object and an associated object or objects- for which the access is computed. You want to analyze when the CUBESAT satellites pass through CR_FOV's ( ) field-of-view.

1. Click Access ( ) in the STK Tools toolbar.
2. Within the Access Tool, click Select Object... to the right of Access for: field.
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 you're looking from.
5. Select all the Satellite ( ) objects in the Associated Objects list.
6. Click Compute

Generating an Access Report

In the Reports frame, clicking Access... generates a 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.

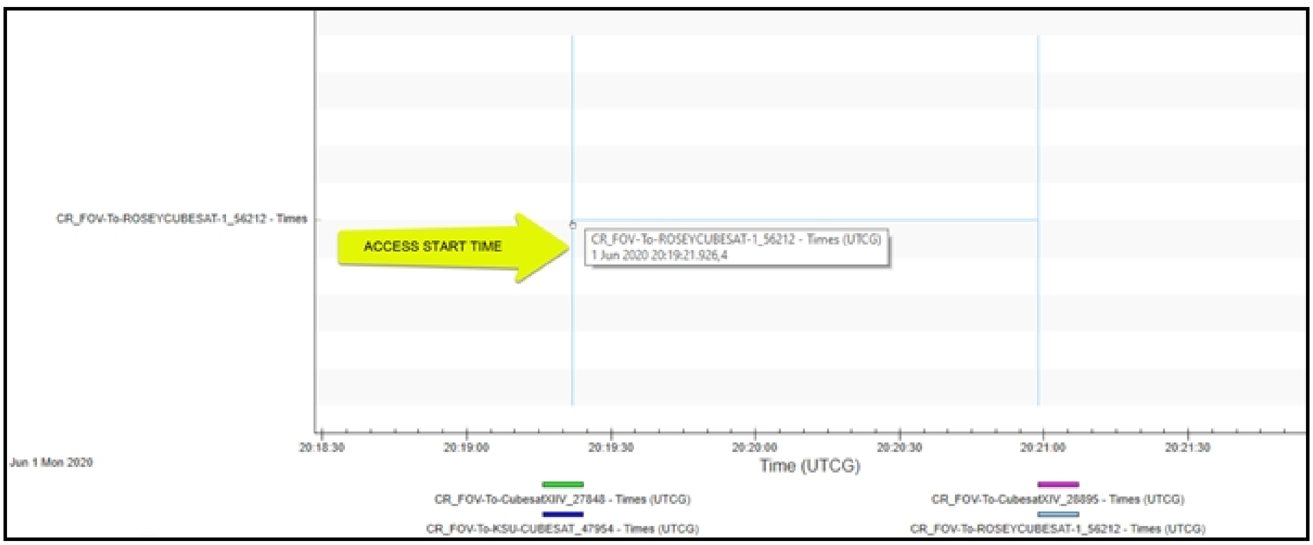
1. Ensure all the satellites are selected in the Associated Objects list.
2. Click Access... in the Reports frame.
3. Scroll through the report to become familiar with the layout.
4. Close the access report once you are done.

Generating an Access Graph

In the Graphs frame, clicking **Access...** generates a 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 frame.

   When you generate a graph, the zoom in function is automatically on.
3. Locate the first access in the graph and using your mouse, hold down the left mouse button, draw a box around the access. This can be done multiple times until the graph is filled with the one access.
4. Place the cursor at the beginning of the access. A text box will appear with information about the access start time.



5. Click Zoom Out ( ) until you see the whole graph.
6. Close the access graph when finished.

Generating an Azimuth Elevation Range Report (AER)

In the Reports and Graphs frames, clicking **AER...** generates an access report or graph with azimuth, elevation, and range data. In order to interpret the data correctly, you should understand the following:

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

   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 (WGS84). 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.

Extending CR_FOV's ( ) range

Extend CR_FOV'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 1500 km in the Range frame's Max: field.
5. Click OK to accept your change and to close the Properties Browser.

Refreshing the access report

The access report is showing the old data.

1. Click Refresh (F5) ( ) in the access report toolbar. You also have the option of clicking F5 on your keyboard to refresh a report.
2. Compare your new data to your old data. You have the same number of accesses but your durations are longer.

Changing the reports units

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. Duration is now reported in minutes instead of seconds.

Quick Reports

Unlike .txt and .csv files, a Quick Report is saved inside of STK.

1. Click Save as quick report ( ) in the access report toolbar.
2. Click Quick Report Manager... ( ) at the top of STK in the Data Providers Toolbar.
3. Change the quick report's name to Sensor to CUBESAT.
4. Click the Enter key on your keyboard.
5. Disable the Show on Load option.
6. Click OK .
7. Close the access report.
8. Close the Access Tool.

Viewing the quick report

1. Return to the Quick Report Manager... ( ).
2. Extend the shortcut menu.
3. Select Sensor to CUBESAT.

Using the Report & Graph Manager

You can generate the following types of output for most STK objects using the Report & Graph Manager which is available from the Analysis menu or the Data Providers toolbar:

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 CubesatXIV_28895 ( ) in the Object List.

Multiple objects can be selected, but for this scenario, focus on CubesatXIV_28895 ( ).

Managing report and graph styles

You can manage the report and graph styles.

1. Ensure (⊞) Installed Styles ( ) in the Installed Styles list is expanded.
2. Select the Classical Orbit Elements ( ) report.
3. Click **Generate** .

Using Data Providers, Groups and Elements

Data Providers, Groups and Elements are the organizing principles of the Data Provider Functionality. STK provides hundreds of prebuilt reports and graphs.

1. Close the report.
2. Return to the Report & Graph Manager.
3. Look at the very top of the styles field. The Classical Orbit Elements ( ) report is now available in the My Favorites ( ) folder.
4. Take a close look at Classical Orbit Elements located in the Installed Styles list. One is a graph style and one a report style. Notice the locks.

The reports and graphs located in the Installed Styles list cannot be customized. However, they can be duplicated and the duplicate can be customized.

Data Providers by Object

1. Right-click on the Classical Orbit Elements ( ) report in the Installed Styles list.
2. Select Properties ( ).
3. Select the Content page when the Properties Browser opens.
4. In the Report Contents list, select Classical Elements-J2000-Semi-major Axis.
5. Return to the Data Providers list. You can see the hierarchy of Classical Elements (Data Provider)- J2000 (Group)- Semi-major Axis (Element).

Report Contents

As previously stated, you will replace argument of perigee, true anomaly and mean anomaly data provider elements with J2000 X Y Z Cartesian Position elements.

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** .
3. Remove the asterisk (*) at the top of the Data Providers list, in the Filter field.
4. Type Cartesian.
5. Click **Filter** . This narrows down your choices to only those data providers containing Cartesian elements.
6. Expand (⊞) Cartesian Position.
7. Expand (⊞) J2000.
8. Move ( ) X, Y, and Z to the Report Contents list.
9. If desired, use the up ( ) and down ( ) arrows to place elements where desired.
10. Click **OK** to accept your changes and to close the Properties Browser.
11. Click **OK** after reading the warning.

Generate the report

1. Expand (⊞) My Styles ( ) in the Styles list.
2. Right-click on Classical Orbit Elements ( ).
3. Select Rename.
4. Rename Classical Orbit Elements ( ) to 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 Sensor ( ) object and the satellites.

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 frame.
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 frame.
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 frame.
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** frame.
3. Browse to your scenario folder (e.g., C:\Users\username\Documents\STK 12\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** frame.
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 frame:
    - 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 STK. Having a spreadsheet containing data, they need to transfer the data into and out of STK 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 STK and extract data from STK or create automation.

Solution

You will learn what an STK Connect command is and how to find and use the Connect Command Library. Using the STK 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.

Create 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 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 STK opens.

The Web Browser

In STK, 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 STK Automation.
5. Click Open .
6. Select API Demo.
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.

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 entered 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 library to easily build applications that communicate with STK.

1. Extend the Help menu.
2. Select STK Help.
3. Select Integrating with STK on the left.
4. Select Connect Command Library.

Alphabetical listing

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

1. Select Alphabetical Listing of Connect Commands.
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.

SetPosition Command

Use the SetPosition command to set the position of a facility, place, target or area target.

1. Return to the alphabetical listing.
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 {CoordType} <Parameters> chart.
6. Note the Geodetic <Parameters> <Lat> <Lon> {<Altitude> | Terrain} [MSL].
7. Return to STK.

Understanding the SetPosition Connect Command

Before you run the code, look at the SetPosition command.

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 code

1. Remove 0.0 (altitude) at the end of the SetPosition Connect command.
2. Type 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

We won't keep jumping back and forth between STK and the Connect Command Library. However, feel free to go there on your own as we proceed through the training. 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 it needs to be propagated.

Propagating MySat ( )

1. Select Modify satellite in the API Demo Utility's Examples list.

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

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

2. Compare the text in the Code Sandbox to the syntax and change the following:

Option	Value
Object name	MySat
Inclination	45 (deg)
RAAN	180 (deg)

3. Click **Run Code** .

Inserting an 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. Then you set the propagator to Great Arc, reference mean sea level (MSL) and create waypoints.

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

3. Click **Run Code** .

Modifying 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	Clear all waypoints from a great arc vehicle.

3. Click Run Code .

Computing an 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	Calculate 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	Define the display of geometric components in the 3D Graphics window.
VO View	Set 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. Right-click on MyFacility_Con ( ) in the Object Browser.
8. Select Zoom To.

   To 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 ( ).

9. Click Start ( ) to animate the scenario.
10. Click Reset ( ) when finished.
11. 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 STK that you can use to send Connect commands.

- STK Button Tool: The Button Tool 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 Example HTML Utilities. This is a utility that allows you to send a Connect command to STK 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 one scenario development exercise; you have 14 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 a personalized STK Certification certificate, an Ansys lanyard, and a custom STK Certification pin. Register now to take the Level 1 - STK Certification test at **https://www.agi.com/training/#cert**.

Upon registration for the STK Certification test, you will receive an email confirmation with an attachment for a 14-day demo license. This license provides you access to all the capabilities needed to complete the certification.

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.

Several of the tutorials use MATLAB, Python, and/or Microsoft Excel. While not required to complete the tutorials, these programs are very useful to have when using the STK software. If you do not have access to these programs, you can open example files in a text editor to view the syntax.

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.

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 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 AzEl Mask Tool and Volumetrics Learn how to use the AzEl Mask Tool and build a volumetric object.	STK Pro, *Analysis Workbench*, *Coverage*	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro

Perform Trade Studies with Analyzer Learn how to use the STK software's Analyzer capability to automate STK 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
Integrating STK with Excel Learn how to control the STK application through an external application like Excel.	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 Advanced CAT Tool Learn how to use the STK software's Conjunction Analysis capability to avoid a satellite collision.	STK Pro, *CAT*	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 could be used to schedule times to transmit location information if someone has access to GPS and a simple FM transmitter and wants to schedule prearranged communications with the base team using a low-Earth-orbiting satellite.

Solution

Use the STK application to load a USGS Digital Elevation Model (DEM) file and analyze the impact of local terrain on accesses between the GPS constellation, the test team, a CubeSat satellite, and the base team. First, change the DEM file into a STK terrain file (.pdtt), 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, factor in the impact of local terrain using an azimuth-elevation (AzEl) mask, and utilize a Chain object to create connections between all the nodes.

Creating a new scenario

Create a new scenario with a run time of 24 hours.

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 File Explorer, browse to the location of the DEM file inside the compressed CodeSamples.zip file at <STK install folder>\CodeSamples\CodeSamples.zip\SharedResources\Scenarios\Events.
2. Copy the file named hoquiam-e.dem.
3. Paste the hoquiam-e.dem file on your desktop.
4. Close File Explorer.

Using the DEM file for analysis and visualization

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 CodeSamples file for both analytical and visual terrain in your scenario. The file contains digital elevation data for the Mount Saint Helens 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 checkbox in the Terrain Server panel.
5. Click **Apply** to accept your changes and 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 visualization and 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 desktop.
5. Select hoquiam-e.dem.
6. Click **Open** to select the file and to close the Open dialog box.
7. Click **Apply** to accept your changes and keep the Properties Browser open.

Updating the local satellite database

Database properties enable you to set the defaults for the city, facility, satellite, and star databases. You can specify a stock STK database or one of your own that meets the STK software's format requirements.

If you do not have an internet connection, you can disregard this task. You'll still be able to insert GPS satellites using the default satellite database that is installed with your version of the STK application. The two-line element (TLE) data will be older, but it will suffice for this training. Skip to **Creating a terrain inlay using the Imagery and Terrain Converter**.

If you're using an internet connection, use the following steps:

1. Select the Basic - Database page.
2. Make sure Satellite is showing in the Database Type drop-down list.
3. Click **Update Database Files...** .
4. Click **Update** when the Update Satellite Database dialog box opens.
5. Click **OK** to close the Information dialog box when the database update is complete.
6. Click **Close** to close the Update Satellite Database dialog box.
7. Click **OK** to accept your changes 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 file (.pdtt) for a specific region.

Selecting the DEM file for conversion

Select your input data file for the source of the terrain data.

1. Open the Utilities menu.
2. Select Imagery and Terrain Converter....
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 to be created.

1. Click the Directory ellipsis (...).
2. Navigate to the location of your scenario (e.g. C:\Users\username\Documents\STK 12\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 terrain and imagery files in Globe Manager

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 3D Graphics window toolbar.
3. 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.
4. Select Add Terrain/Imagery... ( ) in the shortcut menu.

Selecting the files to display in the 3D Graphics window

You can use the Globe Manager: Open Terrain and Imagery Data dialog box to select imagery and terrain data to display in the 3D Graphics window.

1. Open the Path drop-down list when the Globe Manager: Open Terrain and Imagery Data dialog box opens.
2. Select the path to your scenario (e.g. C:\Users\username\Documents\STK 12\TerrainChainsConstellations).
3. Select the checkbox for StHelensTerrain.pdtt.
4. Click **Add** .
5. 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 overlay in the 3D Graphics window.

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

Modeling the GPS satellite constellation

The Global Positioning System (GPS) is deployed and operated as a constellation of navigational satellites. A Constellation ( ) object in the STK application allows you to group objects together for use with other analysis tools available in the STK software, such as *Chains* and *Coverage* capabilities. Constellation objects can be used to group together not just satellite constellations, but also ground station networks, groups of targets, and multiple sensors. A Constellation object allows you to apply constraints and a routing file that determine the criteria as to how the Constellation is used in Chain computations.

Inserting a Satellite object

Start by inserting a Satellite ( ) object using the From Standard Object Database ( ) method.

1. Bring the Insert STK Objects tool ( ) to the front.
2. Select Satellite ( ) in the Insert STK Objects tool ( ).
3. Select the From Standard Object Database ( ) method.
4. Click Insert. . .

Inserting active GPS satellites

If you were analyzing navigational accuracy, dilution of precision, and so on, you would want to use the From GPS Almanac method to propagate your GPS satellites using the GPS propagator. Since you are analyzing accesses between the test team and GPS satellites, you can use your local database of satellites and TLE sets.

1. If you have an Internet connection, clear the Data Sources - Online checkbox when the Search Standard Object Data dialog box opens. This will only show satellite selections from your local database.

   If you do not have an internet connection, the Online data source option will be unavailable and the Local option will be selected by default.

2. Enter GPS in the Name or ID field.
3. Select the Operational Status - Active checkbox. This selection will only show GPS satellites that are active or operational during your analysis time period.
4. Click **Search** .
5. Use CTRL+ Click to select only those GPS satellites whose Mission is listed as Navigation in the Results list.
6. Select the Create Constellation from Selected checkbox in the Insert Options panel. This will automatically group all of the selected GPS satellites into a Constellation ( ) object.
7. Enter GPS_Satellites in the Name field in the Insert Options panel.
8. Click **Insert** .
9. Click **Close** to close the Search Standard Object Data dialog box after your satellites have been propagated.

Viewing the objects assigned to the Constellation

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

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

   You can see that the GPS Satellite ( ) objects were placed in the Assigned Objects list when you created the Constellation ( ) object in the Search Standard Object Data dialog box.

Updating the Constellation object'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 accept your changes and to close the Properties Browser.

Cleaning up the 2D and 3D Graphics windows

Although it's not required, you can adjust the properties of the 2D and 3D Graphics windows to have better situation awareness later in your scenario.

Clearing the satellite orbit tracks

Remove the visible orbit tracks from the 2D and 3D Graphics windows.

1. Open TerrainChainsConstellations' ( ) Properties ( ).
2. Select the 2D Graphics - Global Attributes page when the Properties Browser opens.
3. Clear the Show Orbits/ Trajectories checkbox in the Vehicles panel.
4. Click **OK** to accept your change and to close the Properties Browser.

Decluttering 3D Graphics window labels

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 Graphics window Default toolbar.
3. Select the Details page when the Properties Browser opens.

4. Select the Enable checkbox in the Label Declutter panel.
5. Click OK to accept your change 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 ( ) to TestTeam.

Updating the test team's position

The test team is located in very mountainous terrain. Update the position of TestTeam ( ) 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 accept your changes and to keep the Properties Browser open.

Using the 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 accept your changes 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 Morton'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 accept your changes and to keep the Properties Browser open.

Defining an azimuth-elevation mask for 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 AzEk 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 accept 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 accept your changes 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 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 ( ) to CubeSat.

Propagating the CubeSat satellite

You can set the orbital parameters of the CubeSat satellite using CubeSat'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 software 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 ( ) to 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_Satellites ( ) 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_Satellites ( ) 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, it will upload its 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 download 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 accept your changes and to close the Properties Browser.

Computing the Chain object 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 report to analyze whether or not you have accesses during your scenario analysis period. A complete chain access reports the time intervals for which the chain is completed. These intervals are computed by overlapping all the strand accesses.

1. Right-click 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 schedule times during the next 24 hours, during which time you will have windows of opportunity to transmit test team's location to the base team.
5. Right-click on the Start Time of the longest access duration.
6. Select Start Time in the shortcut menu.
7. Select Set Animation Time in the Start Time submenu.

Viewing the accesses in the 3D Graphics window

You can view the complete chain access in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Zoom to Morton ( ).
3. 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 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 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

Problem statement

Engineers and technicians require additional capabilities when using STK to create custom functions and calculations relative to times, positions, and reference frames. You work at a ground station that tracks a recently launched satellite. The ground station's sensor must be turned off when its boresight is within 10 degrees of the Sun. You need a report that tells you when it's safe to track the satellite.

Solution

Use a combination of STK Pro and STK'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

Create a new scenario.

1. Launch STK ( ).
2. Click Create a Scenario ( ).
3. Enter the following in the New Scenario Wizard:

Option	Value
Name	STK_AnalysisWorkbench

Location	Default
Start	1 Mar 2024 16:00:00.000 UTCG
Stop	+ 2 days

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.
7. Click **Save** .

Disabling Terrain Server

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

1. Right- click on STK_AnalysisWorkbench () in the Object Browser.
2. Select Properties ().
3. Select the Basic - Terrain page.
4. Clear the Use terrain server for analysis check box.
5. Click **OK** to accept the changes and close the Properties Browser.

Inserting a satellite tracking station

Insert a Facility object which will act as a satellite tracking station.

1. Select Facility () in the Insert STK Objects tool.
2. Select the Insert Default () method.
3. Click **Insert...** .
4. Right-click on Facility1 () in the Object Browser.

5. Select Rename.
6. Rename Facility1 ( ) Tracking_Station.

Inserting a recently launched satellite

Insert a Satellite object which will be tracked by the tracking station.

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

Option	Value
Type	Circular
Satellite Name	New_Sat
Inclination	50 deg
Altitude	800 km
RAAN	-85 deg

3. Click OK .

Calculating Access

Calculate the times the tracking station can access or see the satellite using the Access tool. This access will be used as a component when using the *Analysis Workbench*.

1. Right-click on Tracking_Station ( ) in the Object Browser.
2. Select Access... ( ).
3. Select New_Sat ( ) Associated Objects list in the Access Tool.
4. Click (Compute).
5. Look at the Timeline View. By default the Timeline View is docked at the bottom of the STK GUI.

6. Note the multiple accesses between Tracking_Station ( ) and New_Sat ( ).
7. Click **Close** to close the Access tool.

Using the Vector Geometry Tool

The Vector Geometry Tool (VGT) offers the ability to build custom geometric models from any combination of out-of-the box or user-created Vector ( ), Axes ( ), Point ( ), System ( ), Angle ( ) and Plane ( ) components. We will display predefined vectors in the 3D Graphics window.

Opening the Tracking Station's Properties

Open Tracking_Station's ( ) 3D Graphics - Vector Properties page. We will select vectors to display in the 3D Graphics window.

1. Open Tracking_Station's ( ) properties ( ).
2. Select the 3D Graphics - Vector page.
3. Select the Vectors tab.

Displaying Sun Vector

You will display the Sun Vector. In this scenario, the Sun Vector is anchored to tracking station's center point and targets the Sun.

1. Select the Sun Vector Show check box.
2. Click **Apply** to accept the changes 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. Right-click on Tracking_Station ( ) in the Object Browser.
3. Select Zoom To.

4. Zoom out far enough so that you can see the Sun vector.
5. Click Start ( ) in the Animation toolbar to animate the scenario. The Sun Vector follows the Sun.
6. Click the Reset ( ) when you are finished.

Displaying the 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. View the New_Sat's ( ) To Vector.

1. Return to Tracking_Station's ( ) 3D Graphics - Vector properties ( ) page.
2. Select the Vectors tab.
3. Click **Add...** .
4. Notice Tracking_Station ( ) is selected in the Object list on the left of the Add Components window.
5. Click the Expand (⊞) button beside To Vectors ( ) in the Components for: Tracking_Station list on the right.
6. Select New_Sat ( ).
7. Click **OK** to add the New_Sat vector to the Vectors list and close the Add Components dialog box.
8. Notice that Show is selected for the New_Sat Vector.

Showing Magnitude

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

1. Return to the 3D Graphics - Vector page.
2. Ensure New_Sat Vector is selected in the Vectors list.
3. Select the Show Magnitude check box.
4. Click **Apply** .

Viewing the To Vector

View the To Vector and the Magnitude in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Start ( ) to animate the scenario.

   The New_Sat Vector may be below the surface of the central body because the satellite is below the horizon, but will eventually surface. The vector follows the satellite. Show Magnitude provides a constant distance between the tracking station and the satellite, dynamically, in the 3D Graphics window.

3. Click the Reset () when you are finished.

Creating a Custom Angle

Create an angle ( ) between the Sun Vector and the New_Sat To Vector. For the purposes of this scenario, whenever the angle is 10 degrees or less, the satellite tracking system needs to be shut down.

Opening Analysis Workbench Tools

Open the Analysis Workbench Tools, and select the Vector Geometry tab.

1. Right-click on Tracking_Station () in the Object Browser.
2. Select Analysis Workbench... ().
3. Select the Vector Geometry tab in Analysis Workbench window.

Creating a Between Vectors Angle

Create a Between Vectors angle type. Then, create an angle between the Sun Vector and the New_Sat Vector.

1. Select Tracking_Station () in the Object list.
2. Click Create new Angle ( ) to open the Add Geometry Component dialog box.
3. Ensure the Type: is set to Between Vectors. This is the default Type.
4. Type TrackingAngle in the Name: field.

Selecting the From Vector

Select New_Sat's ( ) To Vector as the From Vector. The custom angle, TrackingAngle, will be measured starting from New_Sat's To Vector.

1. Click the From Vector: ellipsis ( ).
2. Select Tracking_Station ( ) in the Objects list in the Select Reference Vector dialog box.
3. Expand (⊞) To Vectors ( ) in the Vectors for: Tracking_Station list.
4. Select New_Sat ( ).
5. Click **OK** to close the Select Reference Vector window.

Selecting the To Vector

Select the Sun Vector as the To Vector. The custom angle, TrackingAngle, will be measured between the New_Sat's To Vector and the Sun vector.

1. Note that To Vector: already has the Tracking_Station Sun vector selected in the Add Geometry Component dialog box.
2. Click **OK** to close the Add Geometry Component dialog box.
3. Keep Analysis Workbench ( ) open.

Displaying the Custom Angle

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

1. Return to Tracking_Station's ( ) properties ( ).
2. Select the 3D Graphics - Vector page.
3. Select the Angles tab.
4. Click **Add...** .
5. Select Tracking_Station ( ) in the Objects list in the Add Components dialog box.

6. Select TrackingAngle ( ) in My Components ( ) in the Components for: Tracking_Station list.
7. Click OK to add the TrackingAngle angle to the Angles list and close the Add Components dialog box.
8. Click OK .

Viewing Tracking Angle

View the tracking angle in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Start ( ) to animate the scenario.
3. Note as the scenario animates, the angle increases and decreases, updating dynamically.
4. Click the Reset ( ) when you are finished.

Creating a Scalar

You will create a Scalar 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. You will start by creating a new Scalar Calculation ( ).

Opening the Calculation Tool

Open the Calculation Tool.

1. Return to Analysis Workbench.
2. Select the Calculation tab to open the Calculation Tool.

Defining a Scalar Calculation

The default scalar calculation type is Angle. The Angle type is an angular displacement specified by an Angle component from the Vector Geometry Tool.

1. Select Tracking_Station () in the Object list.
2. Click Create new Scalar Calculation ().
3. Ensure Type: is set to Angle in the Add Calculation Component dialog box. This is the default Type.
4. Type Scalar_TrackingAngle in the Name: field.
5. Click the Input Angle: ellipsis ().
6. Select Tracking_Station () in the Objects list in the Select Reference Angle dialog box.
7. Select TrackingAngle ( ) in the Angles for: Tracking_Station list.
8. Click OK to close the Select Reference Vector dialog box.
9. Click OK to close the Add Calculation Component dialog box.

Defining a Condition

Create a new Condition ( ).

1. Return to the Calculation Tool.
2. Select Tracking_Station () in the Object list.
3. Click Create new Condition ( ).

Defining Calculation Condition Type

The default Calculation type is Scalar Bounds. Scalar Bounds defines a condition by combining a specified Scalar component with scalar bounds.

1. Ensure the Type: is set to Scalar Bounds in the Add Calculation Component dialog box. This is the default Scalar Bounds.
2. Type Below_10_Degrees in the Name: field.

Defining the Scalar

Set the Scalar Calculation, Scalar_TrackingAngle, as the Reference Scalar Calculation.

1. Click the Scalar ellipsis (...).
2. Select Tracking_Station () in the Object list.
3. Select Scalar_TrackingAngle ( ) in the Scalar_TrackingAngle for: Tracking_Station list.
4. Click **OK** to close the Select Reference Scalar Calculation dialog box.

Defining the Calculation Conditions

Define the calculation conditions to be 10 deg or less.

1. Open the Operation: shortcut menu.
2. Select Below Maximum.
3. Type 10 deg in the Maximum: field.
4. Click **OK** to close the Add Calculation Component dialog box.

Creating a Report

You can create reports and graphs directly inside the Calculation Tool. Create a report showing when the tracking angle is below 10 degrees.

1. Return to the Calculation Tool.
2. Select Tracking_Station () in the Object list.
3. Expand (⊞) Below_10_Degrees ( ) in the Components for: Tracking_Station list.
4. Right-click on SatisfactionIntervals ( ).
5. Select Report...

   The report shows when the tracking angle is 10 degrees or less during the entire scenario interval. However, it doesn't restrict the report to those times when there are accesses between the tracking station and the satellite.
6. Close () the report.
7. Keep Analysis Workbench open.

Viewing Accesses and Satisfaction Intervals in the Timeline View

The Timeline View can be used to visualize a variety of time intervals within your scenario. Use the Timeline View to visualize the times Tracking_Station ( ) has access to New_Sat ( ) and the times the tracking angle is less than 10 degrees.

1. Return to the Calculation Tool.
2. Drag and drop SatisfactionIntervals ( ) into the Timeline View's Time Display.
3. Observe the Timeline View.

Using the Time Tool

Using the Time Tool, you will create an Interval List ( ) by merging access times and the below 10 degree satisfaction intervals. Subtract the below 10 degree satisfaction intervals from the accesses to define tracking opportunities.

1. Return to Analysis Workbench.
2. Select the Time tab.
3. Select Tracking_Station ( ) in the Object list.
4. Click Create new Interval List ( ) to open the Add Time Component dialog box.
5. Type Optimal_Tracking_Times in the Name: field.

Defining the Interval List Type

Create a Merged Interval List. Merged Interval Lists are intervals merged from multiple Interval or Interval List Time Components using a merge operation.

1. Click Select... beside the Type: field.
2. Expand (⊞) Interval List.
3. Select Merged ( ) in the Select Component Type list in the Select Component Type dialog box.
4. Click OK to close the Select Component Type dialog box.

Defining the Operation

Define the merge operation as Minus. Subtract the below 10 degree satisfaction intervals from the accesses to define tracking opportunities. The Minus operation is available when there are only two Time Components in the merge list.

1. Return to the Add Time Component dialog box.
2. Select MINUS for the Operation.

Removing Time Components

Remove default time components.

1. Select the first time component in the Time Components: list.
2. Click **Remove** .
3. Repeat the above steps until all time components are removed.

Defining the Time Components

In this instance, there are two time components: the accesses between the tracking station and the satellite and the below 10 degree satisfaction intervals.

1. Click **Add...** .
2. Select Facility-Tracking_Station-To-Satellite-New_Sat ( ) in the Object list in the Select Time Intervals dialog box.
3. Select AccessIntervals ( ) in the Components for: Facility-Tracking_Station-To-Satellite-New_Sat list.
4. Click **OK** to close the Select Time Intervals dialog box and add the access intervals to the Time Components list.
5. Click **Add...** .
6. Select Tracking_Station ( ) in the Object list in the Select Time Intervals dialog box.
7. Expand (⊞) Below_10_Degrees ( ) in the Components for: Tracking_Station list.
8. Select SatisfactionIntervals ( ).

9. Click **OK** to close the Select Time Intervals dialog box and add the below 10 degree satisfaction intervals to the Components list.
10. Click **OK** to close the Add Time Component dialog box.

Viewing Intervals in the Timeline View

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

1. Drag and drop Optimal_Tracking_Times ( ) into the Timeline View's Time Display.
2. Observe the Timeline View to see the merged intervals.
3. Return to Analysis Workbench.
4. Click **Close** .

Modeling a Tracking Station Sensor

Attach a Sensor object () to the tracking station to track the satellite. Then, using the custom components created in this lesson, model an outage on the sensor when the tracking angle goes below 10 degrees.

Inserting a Sensor on Tracking Station

Insert a sensor object on the tracking station.

1. Insert a Sensor () object using the Define Properties () method.
2. Select Tracking_Station () in the Select Object dialog box.
3. Click **OK** .
4. Rename Sensor1 () to Tracking_Sensor.

Modeling Limited Field-of-View

Set a limited field of view for the sensor to provide situational awareness.

1. Select the Basic - Definition page.
2. Keep the default Simple Conic Sensor Type.
3. Enter 2 deg in the Cone Half Angle: field.
4. Click Apply .

Targeting the Sensor

Use the Targeted pointing type to point the sensor at the satellite.

1. Select the Basic - Pointing page.
2. Select Targeted for the Pointing Type.
3. Select New_Sat ( ) in the Available Targets list.
4. Move ( ) New_Sat ( ) to the Assigned Objects list.
5. Click OK .

Calculating Access

Compute an access between the sensor ( ) and the satellite ( ).

1. Right-click on Tracking_Sensor ( ) in the Object Browser.
2. Select Access... ( ).
3. Select New_Sat ( ) in the Associated Objects list when the Access Tool opens.
4. Click Compute (Compute).
5. Click Access... in the Reports frame.
6. Note the multiple accesses between Tracking_Station ( ) and New_Sat ( ).
7. Keep the Access tool and the report open.

Adding Temporal Constraints

You want to know when the tracking station's sensor accesses the satellite, but restrict access when the sensor's boresight is within 10 degrees of the Sun. You can do this by adding a Temporal Constraint to your sensor object using the custom components created earlier in this lesson.

Adding a Temporal Intervals Constraint

First you need to add the Temporal Intervals constraint to the active constraints list.

1. Open Tracking_Sensor's ( ) properties ( ).
2. Select the Constraints - Active page.
3. Click Add new constraints ( ).
4. Clear the All Categories check box in the Filter by Categories list in the Select Constraints to Add dialog box.
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 Constraint Properties

Set the constraint to merged interval list.

1. Choose Select time component as the Source in the Constraint Properties section.
2. Click **Select..** .
3. Select Tracking_Station ( ) in the Object list in the Select Interval, Interval List or Interval Collection dialog box.
4. Select Optimal_Tracking_Times ( ) in the Components for: Tracking_Station list.
5. Click **OK** to close the Select Interval, Interval List or Interval Collection dialog box.

6. Note that the Exclude Time Intervals option is clear. This means you are only including the merged interval times. The Access tool will only report accesses during the optimal tracking times.

7. Click OK .

Creating a New Access Report

Create a new access report to compare the original access report with the constrained report.

1. Return to the Access tool ( ).
2. Click Access... in the Reports section.
3. Compare your original access report's Total Duration with the constrained access report's Total Duration.
4. Note the temporal constraint shortens your access time.
5. Note the times in the access report are the times your sensor can safely access the satellite.

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 frame.
4. Select Global.
5. Enter 4 deg in the Lat/Lon field located in the Point Granularity frame.
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 frame.
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 frame.
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 frame 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 frame.
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 frame.

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 frame.
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 frame.
3. Select the Show Contours option in the Display Metric frame.
4. Click Remove All in the Level Attributes frame.

5. Set the following in the Level Adding frame:

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 frame.
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) frame.

    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 frame.
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.

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. Engineers want to determine how much impact the Earth, terrain and the radome will have on the radar's field of view.

Solution

Using STK, insert Facility objects which will simulate the radar and communication sites. Use a local terrain file for analysis and determine access times between the radar field of view and five earth observation satellites. Use the AzEl Mask Tool to determine if the communication 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. Apply the Analysis Workbench's Spatial Analysis Tool to build a constrained grid to determine how much of the radar's field of view is blocked by the Earth, terrain and the radome at a selected distance and altitudes.

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 STK: New Scenario Wizard:

Option	Value
Name:	AzElMask_Volumetrics
Location:	Default
Start:	15 Mar 2024 12:00:00.000 UTCG
Stop:	16 Mar 2024 12:00:00.000 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.
7. Click Save .

Turning off Terrain Server

A local analytical terrain file will be used in this analysis. Disable the 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.
5. Click OK to accept the change and close the Properties Browser.

Adding analytical and visual terrain

An STK Terrain File (pdtt) located in the STK install area will be used for analysis and situational awareness in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Click Globe Manager () in the 3D Graphic window toolbar.
3. Click Add Terrain/Imagery () on the Globe Manager toolbar when Globe Manager opens.

4. Select Add Terrain/Imagery... ( ) in the shortcut menu.
5. Click the Path: ellipsis ( ) in the Globe Manager: Open Terrain and Imagery Data dialog box.
6. Navigate to <STK Install Folder>\Data\Resources\stktraining\imagery (e.g. This PC then C:\Program Files\AGI\STK 12\Data\Resources\stktraining\imagery) in the Browse For Folder dialog box.
7. Click OK .
8. Select RaistingStation.pdtt.
9. Click Add .
10. Click Yes to enable terrain for analysis in the Use Terrain for Analysis prompt.

Decluttering 3D Graphics window labels

Label Declutter is used 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 Windows toolbar.
3. Select the Details page.
4. Select the Enable check box in the Label Declutter frame.
5. Click OK .

Inserting the radar site

Use a Facility object as the radar site location.

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 the Facility1 ( ) in the Object Browser.

6. Select Rename in the shortcut menu.

7. Rename the Facility1 () to Radar_Site.

Moving the radar site to its location

The radar site is located in Germany.

1. Open Radar_Site's () properties ().

2. Select the Basic - Position page.

3. Set the following:

Option	Value
Latitude:	47.8996 deg
Longitude:	11.1142 deg

4. Click Apply .

Defining an Azimuth-Elevation Mask

Define an Azimuth -Elevation Mask (AzEl Mask) to use local terrain analytically. The AzEl Mask constraint leverages a provided or computed AzEl Mask to determine visibility.

1. Select the Basic - AzElMask page.

2. Set the following:

Option	Value
Use:	Terrain Data
Use Mask for Access Constraint	on

3. Click OK .

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).

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 the Sensor1 ( ) to Radar_FOV.

Creating the radar's field of view

Use a Complex Conic sensor pattern. 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: shortcut menu.
4. Select Complex Conic.
5. Enter 180 deg in the Outer: field in the Half Angles frame.
6. Click Apply .

   By setting the Half Angles - Outer: value to 180 deg (vertical angle) and leaving the default Clock Angles values (horizontal angle), you've created a 360 degree FOV.

Raising the antenna's field of view

The radar's antenna is positioned twenty (20) feet above the ground's surface. Sensor Location properties enable you to position a sensor with respect to its parent object. A 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: shortcut menu.
3. Select Fixed.
4. Enter -20 ft in the Z: field in the Fixed Location frame.
5. Click **Apply** .

Using the AzEl Mask constraint

A Sensor object can 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** .

Visualizing the terrain constraint

2D Projection Graphics for sensors 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 field of view of the sensor.

1. Select the 2D Graphics - Projection page.
2. Select Use Constraints in the Field of View frame.
3. Select AzElMask in the list.
4. Click **Apply** .

Defining the 3D Graphics Projection properties

3D Graphics Properties for Sensors - Projection is used to control the display of a sensor's cone into space as well as the sensor's extension into 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. 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 frame.
3. Click OK .

Viewing the radar antenna's field of view

You are using a Sensor object to visualize the projected field of view of a radar antenna.

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.

   Your image might look different from the image in this tutorial. You can orient the 3D Graphics window to obtain the same view but it's not required.

   If you were only using the Line of Sight constraint, the sensor field of view would be round. 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 Satellites

The radar site's primary purpose is to track three (3) Technology Development satellites.

1. Insert a Satellite ( ) object using the From Standard Object Database ( ) method.
2. Set the following in the Standard Object Database tool:

Option	Value
Owner:	Germany
Mission:	Technology Development
Operational Status:	Operational

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

Determining Access

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

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 Associated Objects list when the Access tool opens.
4. Click Compute .
5. Click **Access...** in the Reports frame.
6. Scroll to the bottom of the report.
7. Note the Total Duration value in the Global Statistics section (e.g. ~ 15137 seconds).

Saving access report as an external text file

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

1. Return to the access report.
2. Click Save as text ( ) in the Access report toolbar.
3. Ensure your scenario folder displays in the in Address bar in the Save Report dialog box.

4. Type Sensor to Satellites Terrain Only in the File name: field.
5. Click **Save** .
6. Close the access report.
7. Close the Access tool.

Inserting the communication site radome

Construction crews will build a large communication site radome less than one half a kilometer from the proposed radar site. Use a Facility object as the radar site location.

> **Note:** In this scenario, you're using a Facility object to simulate the communication 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.

1. Insert a Facility ( ) object using the Insert Default ( ) method.
2. Rename Facility2 ( ) to Comm_Radome.
3. Open Comm_Radome's ( ) properties ( ).
4. Select the Basic - Position page.
5. Set the following:

Option	Value
Latitude:	47.899 deg
Longitude:	11.1113 deg

6. Click **OK** .

Viewing the communication site radome

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.
4. Change your view so that you can see the communication 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 communication site's radome. The Facility object's 3D Graphics model is not being used as an obstruction during analysis. In order to use the Facility object's 3D graphics model as an obstruction, you use the AzEl Mask tool.

Opening the AzEl Mask tool

Use the AzEl Mask tool to create a body masking file (.bmsk) that can be used in access computations and visualization. The static body masking files (.bmsk) that are created are used to restrict visibility to a sensor.

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

   The 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 AzEl Mask dialog box enables you to identify obscuring objects and define the instant in time at which obscuration contours are computed.

Preparing the AzEl Mask tool

Start by setting up the AzEl Mask tool prior to creating a .bmsk file. 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 frame.

4. Click Apply .

5. Click Compute...

6. Ensure your .bmsk file (use the default file name) is being saved in your scenario folder in the Select Body Mask File dialog box.

7. Click Save .

8. Close the AzEl Mask window and the Az/El Mask View window when the computation is complete.

Constraining the sensor with the AzEl Mask

You can use the body mask file (.bmsk) as a Sensor object's access constraint.

1. Open Radar_FOV's ( ) properties ( ).
2. Select the Basic - Sensor AzEl Mask page.
3. Open the Use: shortcut menu.
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 Use Mask for Access Constraint.
10. Click Apply .

Visualizing the sensor AzEl Mask constraint

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 frame.
3. Scroll down the Field of View - Use Constraints list until you locate SensorAzElMask.

4. Press Ctrl on your keyboard and select SensorAzElMask.
5. Click OK .

Viewing the sensor AzEl Mask constraint in the 3D Graphics window

You can view the constraint 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 Access

Determine the total time each satellite appears within the sensor's field of view.

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 frame.
5. Scroll to the bottom of the report.
6. Note the Total Duration value in the Global Statistics section (e.g. ~ 15027 seconds).

Comparing data

Compare access data between the access using terrain and the access using terrain and a .bmsk file. The best result would be that you don't lose any access duration.

1. Open Windows File Explorer ().
2. Browse to your scenario folder (e.g. C:\Users\username\Documents\STK 12\AzElMask_Volumetrics).

3. Open the Sensor to Satellites Terrain Only.txt file.
4. Return to STK.
5. Compare the Global Statistics Total Duration time in the text file to the Total Duration time in the Access Report.
    - Did the communication site radome affect your total duration access time?
6. Close both reports and the Access tool when finished.
7. Open the Analysis menu.
8. Select Remove All Accesses.

Analyzing the radar's field of view

Based on the curvature of the Earth and that both 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, terrain and the radome in a 360 degree circle using a radius of 3000 kilometers from the radar site. A 3D volume of space will be analyzed inside the sphere between 10 to 700 kilometers in altitude. In order to determine how much the radar field of view is affected, you will use an Area Target object, a Volumetric object, and the Analysis Workbench Spatial Analysis tool.

Inserting an Area Target

Insert an Area Target ( ) to model the 300 km circle on the ground.

1. Insert an Area Target ( ) object using the Area Target Wizard ( ) method.
2. Set the following in the Area Target Wizard ( ):

Option	Value
Name:	OpsArea
Area Type:	Ellipse
Semi-Major Axis:	3000 km
Semi-Minor Axis:	3000 km

Centroid:	47.8996 deg (Latitude)
Centroid:	11.1142 deg (Longitude)

3. Click OK .

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 ( ).
3. Move your view so that you can see OpsArea ( ).

Opening the Analysis Workbench Time tool

There is no need to calculate the radar field of view for the entire analysis period of 24 hours. You can use the Analysis Workbench Time tool to create a one (1) second 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 Time Interval

You want a time interval of one (1) second.

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 close the Select Component Dialog box.
5. Type One_Second in the Name: field when you return to the Add Time Component dialog box.
6. Enter 15 Mar 2024 12:00:01.000 UTCG in the Stop Time: field.
7. Click **OK** .

Creating the Spatial Analysis Reference Grid component

The Anslysis Workbench's 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. You need to create two (2) grids: a Cartographic 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.

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. Type Ref_Grid in the Name: field.
6. Look at the following check boxes:
7. Click **Set Grid Values...**
8. Set the following in the Altitude frame 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.

9. Click OK to close the Grid Values dialog box.
10. Click OK to close the Add Spatial Analysis Component dialog box.

Creating the Spatial Analysis Constrained grid component

A constrained volume grid is one in which the grid points from the reference grid are available only when the spatial condition is satisfied.

1. Select OpsArea ( ) in the Object list.
2. Click Create New Volume Grid ( ).
3. Click Select... for Type: when the Add Spatial Analysis Component dialog box opens.
4. Select Constrained ( ) in the Select Component Type list when the Select Component Type dialog box opens.
5. Click OK to close the Select Component Type dialog box.
6. Type OpsArea_Constrained in the Name: field when you return to the Add Spatial Analysis Component dialog box.

Selecting the Reference Grid

Select the Ref-Grid ( ) 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.

Setting the Spatial Condition

Select the Radar_FOV's ( ) Visibility as the Spatial Condition.

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

The Volumetric object defines a 3-dimensional grid of points using various coordinate definitions, with respect to various reference coordinate systems from the Vector Geometry tool. It also defines the conditions and calculations that depend on locations in 3D space and evaluates these conditions and calculations across grid points.

1. 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 1000 kilometers. Change it to the 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 .

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** .

Selecting the Volumetric Basic Interval

Apply the one second time interval created in the Analysis Workbench 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** .

Computing visibility inside the grid

The 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. Click Save ().
2. Select Volumetric1 () in the Object Browser.
3. Open the Volumetric shortcut menu.
4. Select Compute.

Generating a report

Generate a report that shows how much of the radar's field of view is visible.

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 list when the Report & Graph Manager opens.
4. Click **Generate...** .

   You can see in the report that the radar field of view's percent satisfied is approximately 44 percent. 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 3D Graphics Grid page is 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** .

Viewing Volumetric 3D Graphics Volume

The Volumetric 3D Graphics Volume page is used to show active grid points or spatial calculation levels. Focus on 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 in the Insert Evenly Spaced Values dialog box:

Option	Value
Units	km
Start Value:	10 (km)
Stop Value:	700 (km)
Step Size:	100 (km)

5. Click **Create Values** .

Adjusting 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 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 Volumetric 3D Graphics Legends

The Volumetric 3D Graphics Legends page allows you to place a legend in the 3D Graphics window which explains what the colors mean.

1. Select the 3D Graphics - Legends page.
2. Select the Fill Legend tab.
3. Set the following:

Option	Value
Show Legend	on
Title:	Altitude (km)
Number Of Decimal Digits:	0
Color Square Width (pixels):	40

4. Click OK .

Viewing the contours

1. Bring the 3D Graphics window to the front.
2. Use your mouse to change your view and get an idea of how obscurations affect the different levels of altitude.

Summary

A new radar site is being proposed that will track LEO satellites. 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: Perform Trade Studies with Analyzer

> **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 how various orbital parameters such as semi-major axis, eccentricity, inclination, argument of perigee, right ascension of the ascending node or true anomaly will effect the ability of a sensor or camera to view the surface of the Earth. In this lesson, you want to quickly run multiple trade studies to determine how inclination, eccentricity or a combination of both will effect the percentage of coverage over the entire Earth during a 24 hour period.

Solution

Use STK Pro and STK's *Coverage* and *Analyzer* capabilities to run Parametric and Carpet Plot analyses. The studies will determine the best combination of inclination and eccentricity that provides the highest percentage of global coverage.

Create a New Scenario

Create a new scenario.

1. Launch STK ( ).
2. In the Welcome to STK window, click Create a Scenario ( ).
3. Enter the following in the New Scenario Wizard:

Option	Value

Name:	STK_Analyzer
Location:	Default
Start:	1 Jul 2020 16:00:00.000 UTCG
Stop:	2 Jul 2020 16:00:00.000 UTCG

4. When you finish, 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 and click Save.

View 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 Time Line at the bottom of STK.
2. Click Window in the STK menu bar.
3. Select Tile Vertically.

Turn Off Terrain Server

Analytical and visual terrain is not required in this analysis.

1. Open STK_Analyzer's () properties ().
2. Select the Basic - Terrain page.
3. Clear the Use terrain server for analysis option.
4. Click OK to accept the changes, and close the Properties Browser.

Design a Satellite Orbit

Insert a Satellite ( ) object, and place it in a retrograde orbit. Use the Orbit Wizard Orbit Designer which allows you to create a custom orbit.

1. Using the Insert STK Objects tool, insert a Satellite ( ) object using the Orbit Wizard ( )method.
2. Set the following:

Option	Value
Type:	Orbit Designer
Satellite Name:	MySat
Semimajor Axis:	10600 km
Eccentricity:	0.363
Inclination:	116 deg
Argument of Perigee:	270 deg
RAAN:	104 deg
True Anomaly:	90 deg

3. Click OK .

Sensor Object

Use a Sensor ( ) object that provides a simple conic, 20 degree, field-of-view. The field of view will simulate a camera's field-of-view.

1. Using the Insert STK Objects tool, insert a Sensor ( ) object using the Define Properties ( ) method.
2. When the Select Object window opens, select MySat ( ).
3. Click OK.
4. On the Basic - Definition page, change the Simple Conic - Cone Half Angle: value to 10 deg.

5. Click OK to accept the changes, and close the Properties Browser.

6. Rename the Sensor ( ) object Camera_View.

Coverage Definition Object

The Coverage Definition object defines a coverage area for analysis. In this instance, you require global coverage.

1. Using the Insert STK Objects tool, insert a Coverage Definition ( ) object using the Insert Default method.

2. Rename the Coverage Definition ( ) object Global_Grid.

3. Open Global_Grid's ( ) properties ( ).

Coverage Grid

Global Coverage grid type creates a grid that covers the entire globe.

1. On the Basic - Grid page, change Grid Area of Interest - Type: to Global.

2. Click Apply to accept the changes, and keep the Properties Browser open.

Coverage Assets

Assets properties allow you to specify the STK objects used to provide coverage.

1. Select the Basic - Assets page.

2. In the Assets list, select Camera_View ( ).

3. Click Assign.

4. Click OK to accept the changes, and close the Properties Browser.

Figures of Merit

A Figure of Merit ( ) object allows you to specify the method by which the quality of coverage is measured. The default Coverage type for a Figure Of Merit (

) object is simple coverage.

1. Using the Insert STK Objects tool, insert a Figure Of Merit () object using the Insert Default method.
2. When the Select Object window opens, select Global_Grid ().
3. Click OK.
4. Rename the Figure Of Merit () object Simple_Cov.

Compute Accesses Tool

The ultimate goal of Coverage is to analyze accesses to an area using assigned assets.

1. Right-click Global_Grid () in the Object Browser.
2. Select CoverageDefinition.
3. Select Compute Accesses.

Percent Satisfied

Percent Satisfied is the percentage of 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. In this scenario, you're interested in the Percent Coverage static value of Simple Coverage, meaning the percent of the coverage grid area covered by at least one asset at some point during the coverage interval.

1. Right-click on Simple_Cov () in the Object Browser.
2. Select Report & Graph Manager ().
3. When the Report & Graph Manager opens, 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. ~49 percent). The value you see is the scenario benchmark.
6. When finished, close the report and the Report & Graph Manager.

Analyzer

Analyzer provides a set of analysis tools that:

- Enable you to understand the design space of your systems.
- Enable you to perform analyses in STK easily, without involving programming or scripting.
- Introduce trade study and post-processing capabilities.
- Can be used with all STK scenarios, including those with STK Astrogator satellites.

Determine the Impact of Satellite Inclination on Percent Satisfied

The first study you will perform varies inclination and its effect on global coverage. You need to select input and output variables from the main *Analyzer* window to pass to the Parametric Study tool.

1. Click View on the STK menu bar.
2. Select Toolbars.
3. Select Analyzer.
4. Click the *Analyzer* ( ) button on the *Analyzer* Tool Bar.

Analyzer Layout

Use the *Analyzer* Main Form to configure input and output variables available for further analysis. You will first select an object in the STK Variables tree on the left. When an object is selected, all possible input variable candidates are listed under the STK Property Variables - General tab and the Active Constraints tab. All output variable candidates are listed under the Data Provider Variable - Data Providers tab and Object Coverage tab.

Input Inclination

Start by selecting Inclination as the Input variable.

1. Select MySat ( ) in the STK Variables tree.
2. Expand (⊞) Propagator (TwoBody) in the STK Property Variables - General tab.
3. Double-click on Inclination to move it to the *Analyzer* Variables field as an Input.

Output Percent Satisfied

The same data providers that are in the Report & Graph Manager are available in the Data Provider Variables list. Select Percent Satisfied as the Output variable.

1. In the STK Variables tree, expand (⊞) Global_Grid ( ).
2. Select Simple_Cov ( ).
3. Expand (⊞) Static Satisfaction in the Data Provider Variables - Data Providers tab.
4. Double-click on Percent Satisfied to move it to the Analyzer Variables field as an Output.

Parametric Study Tool

1. Click Parametric Study... ( ) on the *Analyzer* toolbar to open the Parametric Study Tool.
2. Drag and drop the Design Variable, Inclination, from the Component Tree on the left to the Parametric Study Tool on the right.
3. Set the following:

Option	Value
starting value:	112
ending value:	120
step size:	1

4. Notice the number of samples is automatically calculated as nine based on the values we set.
5. Drag and drop the Percent Satisfied data provider element into the Responses field.
6. Click Run...
7. Notice nine runs are performed since the number of samples is set to nine.

Running the data explorer

The Data Explorer is a Trade Study tool used to display data collected from STK. While data is being collected in the Table, the Data Explorer window displays a progress meter, a halt button, and the data. Once the Parametric study is complete, the Table page and 2D Scatter Plot display the collected data.

1. When the Parametric study is finished, close the 2D Scatter Plot.
2. Select the Table Page - Trade Study 1 - Data Explorer window to bring it to the front.
3. Notice nine runs were performed at 1 deg increments from 112 deg to 120 deg inclination.
4. Notice the second row shows the global coverage percentage for each change in inclination.
5. Note that an inclination of 120 deg provides the highest percentage of global coverage.
6. On the Table Page tool bar, click Add View.
7. Select 2D Line plot.
8. Click Axes.
9. Select the Ticks tab.
10. Change the Max # value to 20.
11. Click anywhere on the plot to close the Axes menu.
12. Looking at the 2D Line Plot, you see that 120 deg inclination (basing the analysis on one degree increments) gives you the best choice for global coverage during the 24 hour period.
13. Close the 2D Line Plot and the Table Page.
14. Select No when the Save window appears.
15. Close the Parametric Study ( ) window.
16. Return to the *Analyzer* ( ) window.

Input Eccentricity

Eccentricity could have an impact on the sensor's footprint. However, you have to take into consideration the possibility of the satellite impacting the Earth's surface when changing the eccentricity.

1. Select MySat ( ) in the STK Variables tree.
2. In the STK Property Variables - General tab, expand (⊞) Propagator (TwoBody).
3. Double-click on Eccentricity to move it to the *Analyzer* Variables field as an Input.
4. Click the Parametric Study Tool ( ) button on the *Analyzer* toolbar.

Eccentricity Parametric Study

Let's set up a Parametric Study with Eccentricity as the Design Variable and Percent Satisfied as the Response.

1. Drag and drop the Design Variable, Eccentricity, from the Component Tree on the left to the Parametric Study Tool on the right.
2. Set the following:

Option	Value
starting value:	.362
ending value:	.364
number of samples:	10

3. Notice the step size is automatically calculated based on the values we set.
4. Drag and drop the Percent Satisfied data provider element into the Responses field.
5. Click Run...

Running the data explorer

1. When the Parametric study is finished, close the 2D Scatter Plot.
2. Select the Table Page - Trade Study 2 - Data Explorer window to bring it to the front.
3. On the Table Page tool bar, click Add View.
4. Select 2D Line plot.
5. Click Axes.
6. Select the Ticks tab.

7. Change the Max # value to 40.

8. Click anywhere on the plot to close the Axes menu.

9. Place your cursor over one of the points. A small window will appear with analytical information concerning the point.

10. Close the 2D Line Plot and the Table Page.

11. Select No when the Save window appears.

12. Close the Parametric Study () window.

13. Return to the *Analyzer* ( ) window.

Carpet Plot Tool

1. To access the Carpet Plot Tool (), click Carpet Plot... on the *Analyzer* toolbar.

2. Drag and drop Inclination from the Component Tree on the left to the first Design Variable field on the right.

3. Drag and drop Eccentricity from the Component Tree on the left to the second Design Variable field on the right.

4. Set the following Inclination Design Variable:

Option	Value
From	119
To	121
Step Size	1

5. Set the following Eccentricity Design Variable:

Option	Value
From	.361
To	.365
Step Size	0.001

6. Drag and drop the Percent Satisfied data provider element into the Responses field.

7. Click Run.

The Best Combination

Using the Carpet Plot Tool, look for the best combination of Inclination and Eccentricity. First, you will make the Carpet Plot easier to read, then look at the data.

1. Click Axes.
2. Select the Lines tab.
3. Change the Grid Lines value to 10.
4. Click anywhere on the plot to close the Axes menu.

   The original benchmark of global coverage was approximately 49 percent. In this study, an Inclination of 120 degrees and an Eccentricity of 0.361 provided the best percentage of global coverage of approximately 51.6 percent.

5. When finished, close the Carpet Plot and the Table Page.
6. Select No when the Save window appears.
7. Close the Carpet Plot Tool ( ) window.
8. Close *Analyzer* ( ).

Summary

You began the scenario by placing a Satellite ( ) object in a retrograde orbit. You attached a Sensor ( ) object to the Satellite ( ) with a 20 degree field of view and orientated to point straight down below the Satellite ( ) to the Earth's surface. Using a Coverage Definition ( ) object, you created a global grid and assigned the Sensor ( ) object as the asset. Using a Figure Of Merit ( ) object and Simple Coverage, you determined that approximately 49 percent of the Earth's surface was accessed by the Sensor ( ) object during a 24 hour analysis period. The Satellite's ( ) original Inclination was 116 degrees and its Eccentricity was 0.363. Using *Analyzer*, you ran two Parametric studies changing Inclination and Eccentricity and studied their effects on global coverage. You ended the analysis by running 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 Inclination was 120 degrees and Eccentricity was 0.361. This combination raised global coverage percentage from the benchmark of approximately 49 percent to 51.6 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 STK ( ).
2. Click **Open a Scenario** ( ) in the Welcome to STK dialog box.
3. Go to <STK install folder>\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 STK 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 STK to others. However, you want to use a scenario file while working with STK on your machine.

If you open a VDF file, STK keeps it as a VDF and does not automatically convert it to a scenario file. This means, STK 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. Open the File menu.
2. Select Save As.
3. Select the STK User folder on the left side of the Save As window.
4. Click **New Folder** .
5. Rename New Folder (e.g. CommsPart1, CommsPart2, RadarIntro, etc). You will use the name of the folder as the name of your Scenario object.

   > When naming objects, keep in mind that the name:
   >
   > - Cannot be more than 64 characters in length.
   > - Can contain only alphanumeric characters, underscores and hyphens.
   > - Cannot be "_Default" or "end" (regardless of case), as these are reserved words in STK.

6. Open the folder you just created.
7. Enter the name of the folder into **File name** field. This is 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 checkbox 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.
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 dropdown 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.
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 (![STK icon]).

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 frame.
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 frame.
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 frame:

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 frame.
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:

- 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 frame.
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 frame.
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 frame.
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 frame.
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.

Problem statement

If you are a developer, engineer, or operator, you need to do the following tasks quickly and easily:

- Automate repetitive tasks from outside of the STK application
- Integrate STK 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 an STK scenario using MATLAB. To do so, you need to understand access to the following Application Programming Interfaces (API): the STK Object Model and Connect. You also need a basic understanding of the structure and contents of the STK Programming Help.

Solution

Using STK, including the STK *Integration* capability, and MATLAB, create a new instance of STK 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 STK 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.

Accessing 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.

1. Open your preferred internet browser.
2. Go to help.agi.com.

3. Click STK Programming Help.

Integrating with other applications

STK's *Integration* capability enables you to automate repetitive tasks from outside the STK application, integrate STK 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. Click **Select the Right Technology** in the table of contents of the STK Programming Help.
2. Select **Integrate Technologies**.
3. Select **clickable decision tree** on the Integrate With Other Applications topic 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 STK.

Automating repetitive tasks

To automate repetitive tasks in STK, you can use HTML, Connect, and STK Objects to build tools that are accessible from within STK. You can find more information on this in the STK Programming Help by following these steps:

1. Under **Select the Right Technology**, click **Automate Tasks**.
2. Select **clickable decision tree** 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 **clickable decision tree** 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 in STK

STK X enables developers to add advanced STK 2D and 3D visualization 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 API consists of two major subsystems: Connect and the STK Object Model.

1. In the STK Programming Help, click **Using Core Libraries** in the table of contents.
2. Note the advantages and disadvantages of using the Connect and Object Model libraries.

   Choosing either Connect or 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 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 Diagrams PDF.
5. Close the STK Object Model Diagram (PDF) when finished.
6. Keep the STK Programming Help open.

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 script

MATLAB script files are program files with the .m extension. In these files, you can write groups of commands that you want to execute together. Scripts do not accept inputs and do not return any outputs. They operate on data in the workspace. Scripts are the simplest kind of program file because they have no input or output arguments. After creating a script file for this tutorial, you can use it later to practice with.

1. Select the Home tab in the MATLAB Toolstrip.
2. Click **New Script**.
3. Select the Editor tab in the MATLAB Toolstrip.
4. Open the Save drop-down menu.
5. Select **Save As...**.
6. Browse to the folder where you will save the .m file (e.g., C:\Users\username\Documents\MATLAB).
7. Type STK_Matlab in the **File name** field.
8. Ensure **Save as type** is set to MATLAB Code files (UTF-8) (*.m).
9. Click Save .

Writing comments in your script

Write comments by preceding the line with the % symbol.

1. In the MATLAB Editor, type or copy and paste the following comment onto line 1:

```
%Task1
```

2. Press Enter on your keyboard.

Locating 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.

Creating an STK instance

The AgStkObjectRoot Object is the top-level object in the Object Model hierarchy. The AgStkObjectRoot object provides methods and properties to load scenarios, create new ones, and access the Object Model Unit preferences. From the AgStkObjectRoot object, you can access all other objects and collections exposed by the Object Model.

1. Locate Initialization under the subheading "How do I ..." on the MATLAB Code Snippets topic page.
2. Select **Start STK and get a reference to IAgStkObjectRoot**.
3. Type or copy and paste the following code to the STK_Matlab.m script onto lines 2 through 4.

```
% Create an instance of STK
uiApplication = actxserver('STK12.Application');
uiApplication.Visible = 1;
```

4. Press Enter.
5. Place your cursor in front of %Task1.
6. Click **Section Break** in 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 STK. You also made the STK splash screen visible.

10. AGI recommends that you save your MATLAB script after completing each task section.

Using MATLAB IntelliSense and STK Programming Help

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. MATLAB can work with STK using IntelliSense.

Selecting a root of the Application Model

Use IntelliSense to select an application root of the STK Object Model.

1. Put your cursor in the Command Window.
2. Type the following: **root = uiApplication.**; make sure to include the period (.).
3. Press the Tab key on your keyboard after the period (.). IntelliSense will show you possible selections.
4. Return to the STK Programming Help.

5. Type or paste IAgUiApplication in the search field located in the upper-right corner. Case doesn't matter.
6. Press Enter.
7. Select AgUiApplicationLib~IAgUiApplication in the search results. IAgUiApplication represents a root of the Application Model.

Exposing the STK object root

Looking at the IAgUiApplication page, most of what you see in IntelliSense matches the Public Methods and Properties.

1. Scroll down the IAgUiApplication Interface help topic page to Public Properties.
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. Press the Tab while the cursor is just after the period (.) of the **root = uiApplication.** command.
5. Select Personality2 in IntelliSense.
6. Press Enter.
7. Place a semicolon after Personality2.
8. Press Enter.
9. Add this whole line (**root=uiApplication.Personality2;**) to your MATLAB script on line 6.
10. Look at the MATLAB Workspace. You've exposed the STK object root, so you can start creating a scenario.

Scripting for a new scenario

You will use your MATLAB script to interact with STK. You've launched STK via the MATLAB interface. Now create a new scenario, set the scenario time period, and reset the animation time.

1. In the STK_Matlab.m script, type or copy and paste the following comment on lines 7 and 8:

```
%%
%Task2
```

2. Press Enter. 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 SCENARIO MANAGEMENT (under Scenario) on the MATLAB Code Snippets topic page.
5. Select **Create a new Scenario**.
6. In MATLAB, type or copy and paste the following to the STK_Matlab.m script onto lines 9 and 10.

```
% IAgStkObjectRoot root: STK Object Model Root
root.NewScenario('Example_Scenario');
```

7. Press Enter.

Using the IAgStkObjectRoot Interface

IAgStkObjectRoot Interface represents the automation interface supported by the root object of the Automation Object Model.

1. Return to the STK Programming Help.
2. Type or copy and paste "IAgStkObjectRoot" in the Search field.
3. Press Enter.
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. Select ScenarioName in the Visual Basic .NET syntax.

The NewScenario method expects a string that is the scenario name. In the line you added to Task 2, you set the scenario name as Example_Scenario.

Set the analysis start and stop times

1. Return to MATLAB Code Snippets.
2. Locate SCENARIO MANAGEMENT on the MATLAB Code Snippets topic page.
3. Select **Change scenario time period**.
4. Type or copy and paste the following to the STK_Matlab.m script onto lines 11 and 12.

```
% IAgStkObjectRoot root: STK Object Model Root
root.CurrentScenario.SetTimePeriod('Today', 'Tomorrow');
```

5. Press Enter.
    - CurrentScenario Property (IAgStkObjectRoot) returns a scenario object or null if you have not loaded a scenario yet.
    - SetTimePeriod Method (IAgScenario)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

1. Return to MATLAB Code Snippets.
2. Locate SCENARIO MANAGEMENT on the MATLAB Code Snippets topic page.
3. Select **Reset the scenario time**.
4. Type or copy and paste the following to the STK_Matlab.m script onto lines 13 and 14:

```
% IAgStkObjectRoot root: STK Object Model Root
root.Rewind;
```

5. Press Enter. You are using 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.

Looking at STK, a new scenario named Example_Scenario () is open and the time has been reset in the Current Scenario Time field of the Animation toolbar.

Inserting a Target object

To create a new object such as a Target () object, use AgESTKObjectType Enumeration. Not every object in STK has an associated snippet, so you might need to do some editing. Target (), Place ( ), and Facility ( ) objects contain the same properties. 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, use the Facility object code and make some simple changes to the code.

1. Return to your MATLAB script.
2. Type or copy and paste the following comments onto lines 15 and 16:

```
%%
%Task3
```

3. Press Enter.
4. Return to MATLAB Code Snippets.
5. Locate FACILITY on the MATLAB Code Snippets topic page.
6. Select **Create a facility (on the current scenario central body)**.
7. Type or copy and paste the following to the STK_Matlab.m script onto lines 17 and 18.

```
% IAgStkObjectRoot root: STK Object Model Root
target = root.CurrentScenario.Children.New('eTarget', 'MyTarget');
```

8. Press Enter.

   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 null if no scenario has been loaded yet.

1. Return to the STK Programming Help.
2. Enter "CurrentScenario" in the Search field.
3. Press Enter.
4. Select STKObjects~IAgStkObjectRoot~CurrentScenario in the search results.
5. Go to Visual Basic.NET syntax in the CurrentScenario Property (IAgStkObjectRoot) topic page.
6. Select IAgStkObject.

Retrieving children of an object

IAgStkObject Interface represents the instance of an STK object. Children Property (IAgStkObject) returns a collection of direct descendants of the current object.

1. Go to Public Properties on the IAgStkObject Interface topic page.
2. Select Children.
3. Go to Visual Basic.NET syntax in the Children Property (IAgStkObject) topic page.
4. Click IAgStkObjectCollection.

Retrieving a collection of STK objects

IAgStkObjectCollection Interface represents a collection of STK objects.

1. Go to Public Methods in the IAgStkObjectCollection Interface topic page.
2. Select New.

Creating an STK object

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. Click AgESTKObjectType.

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.

Placing the Target object

Place the Target object at a specific location.

Assigning the location

Use a MATLAB code snippet to assign the location of the Target.

1. Return to MATLAB Code Snippets.
2. Locate FACILITY in the MATLAB Code Snippets page.
3. Click **Set the geodetic position of the facility**.
4. Type or copy and paste the following to the STK_Matlab.m script onto lines 19 through 21.

```
% 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. Press Enter.
   - Position Property (IAgTarget) gets the position of the target.
   - IAgPosition Interface provides access to the position of the object.
   - 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 STK

1. Bring STK to the front to view MyTarget ( ) in the 2D and 3D Graphics windows.
2. Right-click 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, 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 ( ).

Inserting a Satellite object

1. Return to your MATLAB script.
2. Type or copy and paste the following comments onto lines 22 and 23:

```
%%
%Task4
```

3. Press Enter.
4. Return to MATLAB Code Snippets.
5. Locate SATELLITE in the MATLAB Code Snippets page.
6. Select **Create a satellite (on the current scenario central body)**.
7. Return to your MATLAB script.
8. Type or copy and paste the following to the STK_Matlab.m script onto lines 24 and 25.

```
% IAgStkObjectRoot root: STK Object Model Root
satellite = root.CurrentScenario.Children.New('eSatellite', 'MySatellite');
```

9. Press Enter.

Learning about Connect

The Connect module provides you with an easy way to connect with STK and work in a client-server environment. You can use the library shipped with Connect to build applications that communicate with STK. This library contains functions, constants, and other messaging capabilities that you can use to connect third-party applications to STK.

1. Return to the STK Programming Help.
2. Go to Using Core Libraries > Connect.

   This is a great place to learn about Connect. Take some time to become familiar with the links on 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. Use Connect to propagate your Satellite ( ) object.There are several propagators in STK (e.g., Two-Body, J2 Perturbation, J4 Perturbation, SGP4, etc.).

1. Go to Using Core Libraries > Connect > Connect Code Snippets.
2. Locate Satellite in the MATLAB Connect Code Snippets topic page.
3. Select **Set initial state of satellite and propagate**.
4. Return to your MATLAB script.
5. Type or copy and paste the following to the STK_Matlab.m script onto lines 26 and 27.

```
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');
```

6. Press Enter.
   - ExecuteCommand Method executes a custom Connect action.
   - Units_Set sets units of measure. You're using Units_Set because Connect's distance unit defaults to meters and you require kilometers.

Setting the Satellite state

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 page.
4. Select Alphabetical Listing of Connect Commands. 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:
    - Syntax will help you understand the input values of your SetState command.
    - The description chart 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

1. Return to your MATLAB script.
2. Using SetState syntax, set the following values in the SetState Connect command that you entered into 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 STK 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.

Creating access intervals between MySatellite and MyTarget

Use the STK Object Model to create access intervals from MySatellite ( ) to MyTarget ( ).

Entering the access commands into MATLAB

Use your MATLAB script to enter access computation commands.

1. Return to your MATLAB script.
2. Type or copy and paste the following comments onto lines 28 and 29:

```
%%
%Task5
```

3. Press Enter.
4. Return to STK Programming Help.
5. Go to Using Core Libraries > STK Object Model > MATLAB Code Snippets.

6. Locate ACCESS 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. Type or copy and paste the following to the STK_Matlab.m file onto lines 30 through 33.

```
% Get access by STK Object
access = satellite.GetAccessToObject(target);
% Compute access
access.ComputeAccess();
```

10. Press Enter.

Computing and checking the access intervals

Run the latest MATLAB commands and check the access calculation results in STK.

- GetAccessToObject Method (IAgStkObject) returns an IAgStkAccess object. In this instance, you're creating an access from the Satellite object to the Target object.
- ComputeAccess Method (IAgStkAccess) recomputes the access between two objects.

1. In the script, place your cursor in front of %Task5.
2. Click **Run and Advance** in the MATLAB Toolstrip.
3. Bring STK to the front. You can see access lines over Asia in the 2D Graphics window.

Retrieve 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 STK.

1. Return to your MATLAB script.
2. Type or copy and paste the following comments onto lines 34 and 35:

```
%%
%Task6
```

3. Press Enter.

4. Return to MATLAB Code Snippets.

5. Locate ACCESS in the MATLAB Code Snippets page.

6. Select **Compute and extract access interval times**.

7. Return to your MATLAB script.

8. Type or copy and paste the following to the STK_Matlab.m script onto lines 36 through 40.

```
% 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);
```

9. Press Enter.

   - ComputedAccessIntervalTimes Property (IAgStkAccess) returns a list of the computed access interval times.
   - IntervalCollections Property (IAgCrdnIntervalsVectorResult) is a collection of interval collections.
   - ToArray Method (IAgIntervalCollection) returns a two-dimensional array of intervals beginning at a given position and having a specified number of rows.
   - SpecifyAccessIntervals Method (IAgStkAccess) specifies a list of intervals to use for the access calculation.

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 column 1 and column 2 so you can read the access data; your data will have different values. Column 1 shows the start times of the accesses and column 2 shows the stop times.

5. When finished, close the cell.

Investigating data providers

STK 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 STK. The STK Programming Help contains details of data provider groups, individual elements, and report and graph styles.

1. Type or copy and paste the following to the STK_Matlab.m script onto lines 41 and 42:

```
%%
%Task7
```

2. Press Enter.
3. Return to STK Programming Help.
4. Go to Library Reference > Data Providers Reference.

   In the Data Providers by Object page, you can spend a lot of time looking at an object's data providers, both groups and elements.

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 DATA PROVIDERS on the MATLAB Code Snippets topic page.
3. Select **Getting Data for a Single Point in Time**.
4. Return to your MATLAB script.

5. Type or copy and paste the following to the STK_Matlab.m file onto lines 43 through 48

```
        % 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. Press Enter.

Setting date format and elements

1. Look at the code from line 46: **root.UnitPreferences.Item('DateFormat').SetCurrentUnit('EpSec');**
    - UnitPreferences Property (IAgStkObjectRoot) provides access to the Global Unit table.
    - Item Property (IAgUnitPrefsDimCollection) returns IAgUnitPrefsDim, for which you give a dimension name or an index. The data provider is DateFormat.
    - 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.

   Your scenario's default date format is Gregorian UTC (UTCG). As stated in the code snippet, changing the DateFormat dimension to epoch seconds to makes the data easier to handle in MATLAB. Since your scenario is 24 hours long, STK 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.
5. Look at Available Units. The Epoch Seconds element is EpSec.

Selecting satellite data providers

1. Return to your MATLAB script.
2. Look at the code from line 47: **satPassDP = satellite.DataProviders.Item('Precision Passes').ExecSingle (2600);**
    - DataProviders Property (IAgStkObject) returns the object representing a list of available data providers for the object. In this case, it is a Satellite ( ) object.
    - ExecSingle Method (IAgDataPrvTimeVar) computes the data given a specific time. SingleTime uses the DateFormat Dimension, which you set to EpSec.
3. Return to STK Programming Help.
4. Go to Library Reference > Data Providers Reference.
5. Select Satellite in the Data Providers by Object page.
6. Select Precision Passes in the Available Data Providers chart.

Retrieving the Precision Pass data

1. Return to your MATLAB script.
2. Look at the code from line 48: **pass = cell2mat(satPassDP.DataSets.GetDataSetByName('Precision Pass Number').GetValues);**
    - cell2mat converts a cell array into an ordinary array in MATLAB.
    - DataSets Property (IAgDrResult) returns a collection of datasets.
    - GetDataSetByName Method (IAgDrDataSetCollection) returns the element for a given name, which in this case is the Precision Passes data provider element Precision Pass Number.
    - 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 STK into MATLAB
- Used data providers to changes time units from UTCG to EpSec
- Extracted a satellite precision pass value from STK into MATLAB

Part 13: 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

You will be analyzing the behavior of a satellite when it has contact with a ground site. The task will be repetitive and you 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 integrated and automated with code. You decide to run the process with Python. Using the resources on the STK Help and Github, you will explore how to model a mission with a script.

- Connect the STK application to a programming interface
- Build a mission through a script
- Extract data from the STK software

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.

Within the Using Core Libraries directory, you will find additional information on using the STK Object Model and Connect.

The STK Object Model section 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 Using Core Libraries, you will find useful Getting Started information outlining the basics of Connect command and response formats. The section also contains useful code snippets that demonstrate the syntax for various Connect commands.

Creating an STK scenario

First, 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 New Scenario Wizard:

Option	Value
Name	STK_Python_Training
Start Time	Default
Stop Time	Default

4. Click OK .
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 .

Integrating STK and Python

The Integrated Jupyter Notebooks for Python Plugin provides an interface to launch a Jupyter server and open an integrated user interface to use Python.

Launching the Python Scripting Interface

There are several ways to open a Python notebook. You will try opening it within the integrated web browser in the STK application. To do so, you need to launch the Python Scripting Interface.

> **Note:** If you don't want to use a Python notebook file, you can enter the commands right in the Python command window; you can download a zipped Python script file at **https://support.agi.com/download/?type=training&dir=sdf/help&file=Python_STK_Training.zip**.

> **Note:** If you are not already logged in, you will be prompted to log in to agi.com to download the file. 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.

1. Select View on the STK menu bar.
2. Select Toolbars.
3. Select Integrated Jupyter Notebooks for Python.
4. Examine the three buttons. The first button from the left opens the interface, the second opens the Github, and the final one opens the Programming Help.
5. Click Launch Python Scripting Interface ( ) to open the Integrated Jupyter Notebooks for Python window.
6. If not already specified, click the ellipsis button (...) in the Python Interpreter Path field to set the Python install path.
7. If not already specified, click the ellipsis button (...) in the Path to script or folder field to set the folder location where you would like to save your script. By default, it will point to your STK User folder at C:\Users\username\Documents\STK 12\PythonScripts.
8. Clear the Create new script on open? check box.

Copying the Python notebook file

To open a Python notebook in the Python Scripting Interface, you must place the Python notebook file in the directory you selected on start-up.

1. Browse to the installed Python_Jupyter_STK_Training.ipynb Python notebook file located at <STK Install Folder>\Data\Resources\stktraining\scripts.

   > **Note:** If you are using the zipped Python notebook file:
   >
   > a. Right-click on the downloaded zipped folder.
   >
   > b. Select Extract All... in the shortcut menu.
   >
   > c. Set the Files will be extracted to this folder: path to a directory of your choice.
   >
   > d. Click **Extract** .

e. Navigate to the directory you specified.

f. The Python_Jupyter_STK_Training.ipynb notebook file will be in the Python_Jupyter_STK_Training folder.

2. Copy the Python notebook file to the Path to script or folder you specified in the Integrated Jupyter Notebooks for Python window.

Launching the Jupyter Notebooks interface

Launch the Jupyter Notebooks interface in the STK application's integrated web browser.

1. Ensure the Launch Inside STK? check box is selected.
2. Click **Launch** to close the Integrated Jupyter Notebooks for Python window and to launch the Jupyter Notebooks interface.
3. Select the check box next to the Python_Jupyter_STK_Training.ipynb in the file list.
4. Click **Open** to open the Python_Jupyter_STK_Training notebook in a new Web Browser window.

Using the integrated Jupyter notebook

The Python_Jupyter_STK_Training notebook contains the completed script needed for this tutorial. You can run the script directly from the notebook.

1. Click into the individual cell of code as indicated in the notebook.
2. Click **Run this cell and advance (Shift+Enter)** as specified.
3. Repeat for each cell of code in the notebook.

Building your own notebook

If you want to build your script from scratch, you can do so in a new Python notebook.

Creating a new Jupyter notebook

Start by creating a new notebook file.

1. Select New in the File menu of the Jupyter notebook Web Browser window.
2. Select Notebook in the New submenu.
3. Use the default kernel (for example, Python 3 (ipykernel)) when the Select Kernel dialog box opens.
4. Click **Select** to confirm your selection and to open a new notebook.

Setting up your workspace

With you new notebook file created, you can start writing your script. You will add cells and enter your script in section and run them. You may also use the completed script by adding or writing in any missing lines of code.

1. Set up your workspace by copying the starter code below and pasting it into the first cell of your new notebook:

```
# STK library imports
from agi.stk12.stkdesktop import STKDesktop
from agi.stk12.stkobjects import *
from agi.stk12.stkutil import *
from agi.stk12.vgt import *
# if using astrogator uncomment the below
# from agi.stk12.stkobjects.astrogator
# if using aviator uncomment the below
# from agi.stk12.stkobjects.aviator
# Python helper library imports
import os
```

2. Connect to the STK application instance. Use the appropriate script below depending on how you connect to the STK application.

   Use the STK_PID environment variable when using the Integrated Jupyter Notebooks for Python Plugin and have multiple instances of the STK application running:

```
STK_PID = os.getenv('STK_PID')
stk = STKDesktop.AttachToApplication(pid=int(STK_PID))
```

   Use the STKDesktop.AttachToApplication method without reference to STK_PID when running a single instance of the STK application. It does not matter if you are in a notebook or not.

```
stk = STKDesktop.AttachToApplication()
```

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 will provide 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.stk12.stkobjects.AgStkObjectRoot`.

5. Once the above lines are entered into the cell, click **Run this cell and advance (Shift+Enter)** . This will create a new STK window.

Connecting and designing a new scenario

Now that you have launched the STK application via the Python interface, see if you can create a new scenario and set the time period via Python. Create a new scenario and analysis period and reset the animation time.

1. Click **Insert a cell below (B)** to add a new cell.
2. Copy the following code to create a new scenario:

```
# 1. Define a Scenario object.
scenario = root.CurrentScenario
```

3. Copy the following code to set the analysis period:

```
# 2. Set the analytical time period.
scenario.SetTimePeriod('Today','+24hr')
```

4. Copy the following code to reset the animation time:

```
# 3. Reset the animation time to the newly established start time.
root.Rewind();
```

5. Click **Run this cell and advance (Shift+Enter)** .

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. Click **Insert a cell below (B)** to add a new cell.
2. Copy the 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 IDE but is optional; the object returned will be AgTarget at runtime even without the case.

3. Copy the 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)
```

4. Copy the following code to add a satellite object to the scenario:

```
# 3. Add a Satellite object to the scenario.
satellite  = AgSatellite(root.CurrentScenario.Children.New
(AgESTKObjectType.eSatellite,"LeoSat"))
```

5. Copy the code snippets below. Examine the Connect command before running it. You will use the SetState Classical command to propagate the Satellite object for the length of the scenario. Rather than manually setting the times, you will use the defined scenario Start and Stop times. You will print them first to confirm:

```
print(scenario.StartTime)
print(scenario.StopTime)
```

```
# 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');
```

6. Click Run this cell and advance (Shift+Enter) .

Computing access between objects

You now have a scenario with a Target object and a Satellite object. Determine when the Satellite object can access the Target object.

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();
   ```

4. In the Jupyter notebook, click Insert a cell below (B) to add a new cell.
5. Enter the above text to the new cell and click Run this cell and advance (Shift+Enter) .

Retrieving access data from STK

Now that the scenario is fully built, 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. Click **Insert a cell below (B)** to add a new cell.
2. Copy the following code to add calls to the access data provider:

```
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.

3. Click **Run this cell and advance (Shift+Enter)** .

Retrieving satellite altitude data from STK

Retrieve and view the altitude of the satellite during an access interval.

1. Click **Insert a cell below (B)** to add a new cell.
2. Copy the following code to add a 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 data provider folder, subfolder, and selection.

3. Click **Run this cell and advance (Shift+Enter)** .

Saving your scenario

You have just completed the Integrating STK with Python tutorial using Jupyter notebooks. Don't forget to save your work.

1. When finished, close all Web Browser windows you still have open.
2. Save () your work.

With your scenario safely saved, you can close out of the STK application. You can expand and automate workflows like these for quickly building and analyzing missions.

Part 13: Integrating STK with Excel

> **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

Developers, engineers and operators need to quickly and easily automate repetitive tasks from outside of the STK application, integrate STK with other applications, such as *Microsoft Excel*, *MATLAB*, and *Python*, develop custom applications and leverage the AGI Object Model collection of COM libraries from other applications. You want to be able to create, drive and extract data from a simple STK scenario using *Excel*. You need to understand access to the following Application Programming Interfaces (API): the STK Object Model and Connect. Finally, you require a basic comprehension of STK Programming Help.

Solution

Use *Excel* to create a new instance of STK and expose the object root. Next, build a new scenario, populating it with a Satellite object and multiple Target objects. Create an access between the Satellite object and a selected Target object. Use the data providers, groups, and elements to retrieve access times from STK into *Excel*. Download, install and use the STK Excel Add-in plugin. Learn the power of STK Programming Help.

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. Only a small portion of the STK Programming Help is covered in this tutorial but it's a great starting point.

1. Open your preferred Internet browser.
2. Go to help.agi.com.
3. Select STK Programming Help.

Integrating STK with other applications

STK's Integration capability enables you to automate repetitive tasks from outside the STK application, integrate STK with other applications – such as *Excel* and *MATLAB* – develop custom applications, and leverage the AGI Object Model collection of COM libraries from other applications.

1. Choose Select the Right Technology in the contents list on the left side of STK Programming Help.
2. Select Integrate Technologies.
3. Select **clickable decision tree** on the Integrate With Other Applications page.
4. Take a moment to view the page and click some of the icons.

   There are multiple 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 using integration.

Automating repetitive tasks

To automate repetitive tasks in STK, you can use HTML, Connect, and STK Objects to build tools that are accessible from within STK.

1. Return to STK Programming Help.
2. Choose Select the Right Technology in the contents list.
3. Select Automate Tasks.
4. Select **clickable decision tree** on the Automate Repetitive Tasks page.
5. Take a moment to view the page and click some of the icons.

Extending AGI products

AGI provides a variety of ways to extend our products. These extensibility mechanisms can be divided into two distinct areas: user interface extensibility and engine extensibility.

1. Return to STK Programming Help.
2. Choose Select the Right Technology in the contents list.
3. Select Extend AGI Products.

4. Click **clickable decision tree** on the Extend AGI Products Using Plugins and Custom User Interfaces page.
5. Take a moment to view the page and click some of the icons.

   There are other links throughout that aren't covered here. Take some time later on to become familiar with them.

Visualize

STK X allows developers to add advanced STK 2D, 3D visualization and analytical capabilities to applications with little effort.

1. Return to STK Programming Help.
2. Choose Library Reference in the contents list.
3. Select Controls.
4. Select STK X.
5. Take some time to familiarize yourself with the contents of this page.

Using Core Libraries

The STK API consists of two major sub-systems: Connect and the STK Object Model.

- Connect is a library of string commands for STK, originally designed to operate over a TCP/IP socket.
- The STK Object Model is an object-oriented interface to STK, built on Microsoft COM technology.

1. Return to STK Programming Help.
2. Choose Using Core Libraries in the contents list.
3. Note the advantages and disadvantages of using each library.

   The choice of Connect or Object Model should be driven by the requirements of the application and the needs of the developer. It is important to note, however, that this need not be an exclusive choice. Connect commands can be used in a COM application side-by-side with Object Model Code.

Naming conventions

The STK Object Model contains naming conventions for classes, interfaces and enumerations.

1. Return to STK Programming Help.
2. Choose Library Reference in the contents list.
3. Select STK Object Model.
4. Select Naming Conventions.
5. Note the various naming conventions described on the Naming Conventions Used In the Various Core Libraries page.
6. Select Abbreviations in the contents list.
7. Note the abbreviations most commonly used in the names of classes, interfaces and other types in the STK Object Model.

Model diagrams

For the 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 pages.

1. Return to STK Programming Help.
2. Choose Using Core Libraries in the contents list.
3. Select STK Object Model.
4. Select STK Objects.
5. Select Diagrams.
6. Select STK Object Model Diagram (PDF) on the STK Objects Object Model Diagrams page.
7. Zoom into the page for clarification if desired.
8. Take some time to look at the STK 10.0 Object Model Diagrams PDF.
9. Close the STK Object Model Diagram (PDF) when finished.
10. Keep STK Programming Help open.

STK Excel Add-in

The STK Excel Add-in is a set of VBA libraries, macros, forms and toolbars that let *Excel* users interact with STK and STK scenarios directly from *Excel*. You can pull data from STK into *Excel* or pass certain data types from

Excel into STK. The STK Excel add-in may already do what you require. At the end of this tutorial, the STK Excel Add-in tool will be used, so now is a good time to install it on your computer.

1. Go to https://www.agi.com/ in your preferred browser.
2. Click **SIGN IN** at the top of the page.
3. Select Sign In in the shortcut menu.
4. Enter your credentials in the Sign In page.
5. Click **SIGN IN** .

Downloads

1. After signing in, at the top of the page, select Support.
2. Select Downloads once the menu opens.
3. If presented, read the Software License Agreement. Otherwise, go to step 5.
4. Click **ACCEPT** .
5. On the Software Downloads page, locate Data and Auxiliary Extensions.
6. Select STK Excel Add-in (v12.0.0).

   This will download the install file as a zipped folder to your designated download location.

Installing the STK Excel Add-in

1. Go to your download location.
2. Unzip (Extract All) the STKExcelAdd-in_v12.0.0.zip folder to your preferred location.
3. Open the extracted (unzipped) STKExcelAdd-in_v12.0.0 folder.
4. Double-click STKExcelAdd-in_v12.0.0.
5. If you receive a security warning, click **Run** .
6. Run the installer.
7. Keep STK Programming Help open.
8. Close any open folders and web pages that your desire.

Configuring Excel

Enable the Developer tab and the STK Excel Add-in.

1. Launch *Excel*.
2. Click Blank workbook once *Excel* launches.

Excel Developer Tab

Turn on the Developer tab in *Excel* if you want to create a macro, export and import XML files or insert controls. If it's not already turned on, complete the following steps.

1. Right-click anywhere on the ribbon.
2. Select Customize the Ribbon...
3. Select Main tabs (if necessary) under Customize the Ribbon shortcut menu in the *Excel* Options dialog box.
4. Select the Developer check box.
5. Click **OK** .

Turning on the STK Excel Add-in

1. Select the Developer tab.
2. Click **Excel Add-ins** .
3. Select the Stk Add-in 12 X64 check box in the Add-ins dialog box.
4. Click **OK** .
5. Close *Excel*. This is required.

Creating an Excel Workbook

1. Launch *Excel*.
2. Click Blank Workbook.
3. Select the File menu when the workbook opens.
4. Click Save As in the contents list on the left.
5. Select Browse.
6. Select Desktop in the Save As dialog box.
7. Select New folder in the menu bar.
8. Rename New folder to STK_Excel.
9. Click **Open** .
10. Type STK_Excel_Workbook in the File name: field.
11. Open the Save as type: shortcut menu.
12. Select Excel Macro-Enabled Workbook.
13. Click **Save** .

Form Control Button

1. Select the Developer tab.
2. Click **Insert** .
3. Click Button (Form Control) in the shortcut menu.
4. Hold down your left mouse button and draw out a small box (2 cells by 2 cells) in the *Excel* workbook.
5. Type Launch_STK in the Macro name: field when the Assign Macro tool dialog box opens.
6. Click **New** .

Change the Button's Name

1. Bring the *Excel* workbook to the front.
2. Highlight the button name (e.g. Button 1) in the button.
3. Type Launch STK.

Creating an STK Instance from Excel

You will use the *Excel* script file to build a simple STK scenario from which you will extract data into *Excel*. You can use VB script or the STK object model and for automation you can use your preferred method. STK Connect will be demonstrated in this tutorial. All the components of Integration are fully documented, and AGI maintains a GitHub repository of Code samples to help you get started with the STK Object Model (located here: **https://github.com/AnalyticalGraphicsInc/STKCodeExamples**). There is a learning curve, so start with STK Programming Help.

1. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
2. Create global variables by Typing or copying and pasting the following code inside the Visual Basic editor above Sub Launch_STK (). This will create global variables.

```
Dim app
Dim root
Dim scenario
```

3. Create a new instance of STK. Type or copy and paste the following Code within Sub Launch_STK().

```
' TASK 1
' Create new instance of STK using the CreateObject Method
Set app = CreateObject("STK.Application")
app.Visible = True
' Expose the STK application root using the Personality2 Property
Set root = app.Personality2
```

4. Click Save STK_Excel_Workbook.xlsm (Ctrl+ S) ().

Understanding Task 1

Set app = CreateObject("STK.Application")

The CreateObject Method expects a string, in this case STK.Application.

1. Bring STK Programming Help to the front.
2. Choose Library Reference in the Contents list.
3. Select Application Object Model.
4. Select STK UI Application.
5. In Classes, Select AgUiApplication on the Project Overview page.
6. In Interfaces, select IAgUiApplication on the AgUiApplication Object page.
7. In Public Methods, select CreateObject on the IAgUiApplication Interface page.
8. In Syntax, look at the [Visual Basic .NET] Syntax on the CreateObject Method (IAgUiApplication) page.

Making STK Visible

app.Visible = True

Visible Property (IAgUiApplication) makes the main window visible.

1. Select Library Reference in the contents list.
2. Select Application Object Model.
3. Select STK UI Application.
4. Select Properties.
5. Select Visible Property (IAgUiApplication).

Exposing the STK Application Root

Set root = app.Personality2

Personality2 Property (IAgUiApplication) returns a new instance of the root object of the STK Object Model.

1. Select Library Reference in the contents list.
2. Select Application Object Model.
3. Select STK UI Application.
4. Select Properties.
5. Select Personality2 Property (IAgUiApplication).

Running the Sub Routine

You can run the sub routine by clicking the Launch STK button or by selecting the sub routine and clicking the green Run button.

1. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
2. Click Save STK_Excel_Workbook.xlsm (Ctrl+ S) ( ).
3. Click inside the Sub Launch_STK () code.
4. Click Run Sub/UserForm (F5) ( ).

   The STK application window appears.

Creating a New STK Scenario

After launching STK, create a new scenario and set the time period.

1. Return to the *Excel* workbook.
2. Click **Insert** in the Developer tab.
3. Click Button (Form Control) in the Form Controls window.
4. While in the *Excel* workbook, hold down your left mouse button and draw out a new button below the Launch STK button.
5. Type Create_Scenario in the Macro name: field when the Assign Macro dialog box opens.
6. Click **New** .
7. Return to the *Excel* workbook.
8. Change the name of Button 2 to Create Scenario.

9. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.

10. Type or copy and paste the following Code within Sub Create_Scenario ().

```
' TASK 2
' Create a new scenario.
root.NewScenario("New_Scenario")
Set scenario = root.CurrentScenario
' Set the analytical time period.
notUsed = scenario.SetTimePeriod("Today", "+24hr")
' Reset the animation time.
notUsed = root.Rewind()
```

    You will create a new scenario called New_Scenario.

11. Click Save STK_Excel_Workbook.xlsm (Ctrl+ S) ( ).

Understanding Task 2

This section will show you another way to use STK Programming Help.

Understanding NewScenario Method

NewScenario Method (IAgStkObjectRoot) creates a new scenario and expects a scenario name as a string.

You use this method in line 3 of Task 2: **root.NewScenario ("New_Scenario")**

1. Return to STK Programming Help.
2. Type NewScenario in the search field.
3. Click Enter.
4. Select STKObjects~IAgStkObjectRoot~NewScenario.

Understanding CurrentScenario Property

CurrentScenario Property (IAgStkObjectRoot) returns a Scenario object or null if no scenario has been loaded yet.

You use this method in line 4 of Task 2: **Set scenario = root.CurrentScenario**

1. Return to STK Programming Help.
2. Type CurrentScenario in the search field.
3. Click Enter.
4. Select STKObjects~IAgStkObjectRoot~CurrentScenario.

Understanding SetTimePeriod()

SetTimePeriod Method (IAgScenario) sets the Scenario time period. startTime/stopTime use DateFormat Dimension.

You use this method in line 6 of Task 2: **notUsed = scenario.SetTimePeriod("Today", "+24hr")**

1. Return to STK Programming Help.
2. Enter SetTimePeriod in the search field.
3. Click Enter.
4. Select STKObjects~IAgScenario~SetTimePeriod.

Understanding Rewind()

Rewind Method (IAgAnimation) is used to stop and reset the animation.

You use this method in line 8 in Task 2: **notUsed = root.Rewind()**

1. Return to STK Programming Help.
2. Type Rewind in the search field.
3. Click Enter.
4. Select STKObjects~IAgAnimation~Rewind.C

Summary of Task 2

In Task 2, you are creating a Scenario object named New_Scenario and setting its analysis time from midnight local time based on your time zone (Today) and running the scenario for 24 hours. Then you are essentially clicking the red Reset button in the Animation Toolbar of STK to reset your scenario time.

STK Programming Help Search Examples

These were two examples of using STK Programming Help. When you are looking at Code snippets and such, in the search window, you can type something from the Code such as Personality2 and it will provide choices to select from. It would take a long time to browse the STK Programming Help throughout this tutorial, so hyperlinks are used to simplify and speed things up. However, practice makes perfect, so feel free to search on your own.

Running Both Sub Routines

1. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
2. Click Save (Ctrl+ S) ( ) in STK_Excel_Workbook.xlsm.
3. Bring the *Excel* workbook to the front.
4. Click **Launch STK** .
5. Click **Create Scenario** once STK opens.
6. Bring STK to the front.

Viewing the Scenario object properties

1. Right-click on New_Scenario ( ) in the Object Browser.
2. Select Properties ( ).
3. Note the Analysis Period Start: and Stop: times when the Properties Browser opens.
4. Click **Cancel** to close the New_Scenario's ( ) properties ( ).
5. Look at Current Scenario Time in the Animation Toolbar. You set it for midnight local time (Today). You will see that the code reset it to the equivalent UTCG date and time.
6. Return to the *Excel* workbook.

Configuring Target1 Objects

Populate the scenario by inserting 25 Target () objects.

1. Type the following column headers into *Excel*:

Cell	Value
F1	Target Name
G1	Latitude
H1	Longitude

2. Select cell F2.
3. Enter Target.
4. Select cell G2.
5. Copy and paste the following Excel RAND function into the Formula Bar:

   =(RAND()-0.5)*180

6. Click **Enter** . The Excel RAND function returns a random number between 0 and 1. This Code will return a random number between -90 and 90.
7. Select cell H2.
8. Copy and paste the following Excel RAND function into the Formula Bar:

   =(RAND()-0.5)*360

9. Click **Enter** . This Code will return a random number between -180 and 180. Your values will be different from the image.

Creating Multiple Targets and Random Latitudes and Longitudes

1. Select cell F2.
2. Hold the Shift key and select cell H2.
3. Place your cursor on the small black box located at the lower right corner of cell H2.

4. Hold down your left mouse button and drag the black box to cell H26 and release your mouse button. This will create 25 targets and 25 random latitudes and longitudes (your values will be different than the example).

5. Click Save (Ctrl+ S) (💾) while in the *Excel* workbook.

Inserting the Targets into STK

Insert the targets into STK using Target (◎) objects.

1. Click **Insert** in the Developer tab.

2. Click Button (Form Control) in the Form Controls window.

3. While in the *Excel* workbook, hold down your left mouse button and draw out a new button below the Create Scenario button.

4. Type Insert_Targets in the Macro name: field of the Assign Macro dialog box.

5. Click **New** .

6. Return to the *Excel* workbook.

7. Change the name of Button 3 to Insert Targets.

8. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.

9. Type or copy and paste the following Code within Sub Insert_Targets().

```
' TASK 3
' Add a target object to the scenario.
For i = 2 to 26
Dim target
Set target = root.CurrentScenario.Children.New(23, Range("F" & i).Value)
' Move the Target object to a desired location.
notUsed = target.Position.AssignGeodetic(Range("G" & i).Value, Range("H" &
i).Value, 0.0)
Next
```

10. Click Save (Ctrl+ S) (💾) in STK_Excel_Workbook.xlsm.

Understanding Task 3

Review the following sections to learn more about each line of Task 3 and discover resources in the STK Programming Help.

Understanding how to loop through rows in the code

You are looping through rows 2 to 26 of your Excel workbook. You create a new Target object on each line for 25 targets total.

You use a loop in line 3 of Task 3: **For i = 2 to 26**

Understanding the CurrentScenario Method

You use the CurrentScenario Method in line 5 of Task 3: **Set target = root.CurrentScenario.Children.New(23, Range("F" & i).Value)**

You will navigate to the CurrentScenario Method in the STK Programming Help for more information.

1. Return to STK Programming Help.
2. Type currentscenario in the search field.
3. Click **Enter** .
4. Select STKObjects~IAgStkObjectRoot~CurrentScenario from the search results.
5. Select IAgStkObject in the Visual Basic .NET Syntax list.
6. Select Children in Public Properties.

Children Property

Children Property (IAgStkObject) returns a collection of direct descendants of the current object.

1. Select IAgStkObjectCollection from the Visual Basic .NET Syntax list.
2. Select New In Public Methods.

New Method

New Method (IAgStkObjectCollection) creates an STK object using specified class and instance name.

1. Select AgESTKObjectType from the Visual Basic .NET Syntax list.

   AgESTKObjectType Enumeration shows the object type enumerations, associated value and the STK objects.

2. Scroll down the Member list until you locate eTarget. To insert a new Target ( ) object, you can use eTarget or the value 23. Your code uses the value 23.

   Next, STK wants a string for the name. You want the name in cell F2. To use all the target names, in VBA, you use Range. You want cell F and the ampersand (&) allows you to concatenate strings. It will start in F2 and finish in F26.

Assigning Locations of objects

In short, the Position Property gets the position of the target. The AssignGeodetic Method is a helper method to assign the position using the Geodetic representation. Learn more about the Position Property (IAgTarget) and the AssignGeodetic Method (IAgPosition) in the STK Programming Help.

You use both of these method in line 7 of Task 3: **notUsed = target.Position.AssignGeodetic(Range("G" & i).Value, Range("H" & i).Value, 0.0)**

Within the AssignGeodetic method:

- **Range("G" & i).Value** will read cells G2 though G26 for the latitudes.
- **Range("H" & i).Value, 0.0)** will read cells H2 though H26 for the longitudes. The altitude is 0.0 for all the targets which will place them on top of the WGS84 ellipsoid if you don't have terrain or on top of the terrain if you're using terrain such as Terrain Server. In the Code, *Excel* converts the value 0.0 is converted to 0#. (Don't worry about it.)

Running all three sub routines

1. Bring the *Excel* workbook to the front.
2. Click **Launch STK** in the *Excel* workbook.
3. Click **Create Scenario** once STK opens.
4. After the scenario is created, click **Insert Targets** .
5. Bring STK to the front.

You created an instance of STK, created a scenario with a 24 hour analysis period and inserted 25 random Target objects around the Earth.

Inserting a Satellite Object into STK

You have launched STK, created a new scenario, set the time period and inserted 25 randomly placed Target ( ) objects. Now you will insert and propagate a Satellite ( ) object.

1. Return to the *Excel* workbook.
2. Click **Insert** in the Developer tab.
3. Click Button (Form Control) in the Form Controls window.
4. While in the *Excel* workbook, hold down your left mouse button and draw out a new button below the Insert Targets button.
5. Type Insert_Satellite in the Macro name: field of the Assign Macro dialog box.
6. Click **New** .
7. Bring the *Excel* workbook to the front.
8. Change the name of Button 4 to Insert Satellite.
9. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
10. Type or copy and paste the following Code within Sub Insert_Satellite ().

```
' TASK 4
' Add a Satellite object to the scenario.
Dim satellite
Set satellite = root.CurrentScenario.Children.New(18, "LeoSat")
' Propagate the Satellite object's orbit.
root.ExecuteCommand ("SetState */Satellite/LeoSat Classical TwoBody
UseScenarioInterval 60 ICRF ""UseAnalysisStartTime"" 7200000.0 0.0 90 0.0
0.0 0.0")
```

11. Click Save (Ctrl+ S) ( ) in STK_Excel_Workbook.xlsm.

Understanding Task 4

Review the following sections to learn how each line of Task 4 works.

Understanding the Children Property in Task 4

You use the Children property to insert a Satellite ( ) object in this case. This code uses the value 18 which loads a Satellite ( ) object into your scenario called LeoSat. Previously, you used similar code to insert a Target ( ) object into the scenario.

You used this property in line 4 of Task 4: **Set satellite = root.CurrentScenario.Children.New(18, "LeoSat")**

Understanding the ExecuteCommand() Method

You use the ExecuteCommand Method to propagate the Satellite ( ) object. In general, the method executes a custom CONNECT action. Learn more about the connect command ExecuteCommand Method (IAgStkObjectRoot) in the STK Programming Help.

You used this method in line 6 of Task 4: **root.ExecuteCommand ("SetState */Satellite/LeoSat Classical TwoBody UseScenarioInterval 60 ICRF ""UseAnalysisStartTime"" 7200000.0 0.0 90 0.0 0.0 0.0")**

SetState Connect Command

Look at the Alphabetical Listing of Connect Commands to understand the SetState connect command.

1. Return to STK Programming Help.
2. Choose Library Reference in the Contents list.
3. Select Connect Command Library.
4. Click Alphabetical Listing of Connect Commands from the Connect Command Listings page.
5. Click S at the top of the Alphabetical Listing page.
6. Select the SetState Classical command.
7. Scroll through the SetState Classical page and note the following:
    - Syntax will help you understand the input values of your SetState command.
    - The description chart explains each input and its associated unit.
    - In the Examples section, there are multiple examples that can be copied and edited for your particular scenario.

Running all four sub routines

1. Return to the *Excel* workbook.
2. Click Launch STK .
3. Click Create Scenario once STK opens.
4. After the scenario is created, click Insert Targets .
5. After the targets are inserted, click Insert Satellite .
6. Bring STK to the front.

You created an instance of STK, a scenario with a 24 hour analysis period, inserted 25 random Target ( ) objects around the Earth and inserted and propagated a Satellite ( ) object.

Creating an Access

Determine when the LeoSat ( ) can access Target1 ( ).

1. Return to the *Excel* workbook.
2. Click Insert in the Developer tab.
3. Click Button (Form Control) in the Form Controls window.
4. While in the *Excel* workbook, hold down your left mouse button and draw out a new button below the Insert Satellite button.
5. Type Compute_Access in the Macro name: field of the Assign Macro dialog box.
6. Click New .
7. Bring the *Excel* workbook to the front.
8. Rename Button 5 to Compute Access.
9. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
10. Type or copy and paste the following Code within Sub Compute_Access ().

```
    ' TASK 5
    ' Compute Access between the satellite and the target
    Dim sat
    Set sat = root.GetObjectFromPath("Satellite/LeoSat")
    Dim tar1
    Set tar1 = root.GetObjectFromPath("Target/Target1")
    Dim access
    Set access = sat.GetAccessToObject(tar1)
    'Compute access
    access2 = access.ComputeAccess()
```

Understanding Task 5

Review these sections to understand more about Task 5.

Understanding GetObjectFromPath()

In short, the method gets the object instance that matches the path provided. Learn more about the GetObjectFromPath Method (IAgStkObjectRoot) in the STK Programming Help.

You use this method in lines 4 and 6 from Task 5:

Set sat = root.GetObjectFromPath("Satellite/LeoSat")

Set tar1 = root.GetObjectFromPath("Target/Target1")

Understanding the GetAccessToObject()

In short, the method returns an IAgStkAccess object to compute access to. Learn more about the GetAccessToObject Method (IAgStkObject) in the STK Programming Help.

You use this method in line 8 from Task 5: **Set access = sat.GetAccessToObject(tar1)**

Understanding the ComputeAccess() Method

In short, the ComputeAccess() recomputes the access between two objects. Learn more about the ComputeAccess Method (IAgStkAccess) in the STK Programming Help.

You use this method in line 10 in Task 5: **access2 = access.ComputeAccess()**

Data Providers

The content of a report or graph is generated from the selected data providers for the report or graph style. You can view data providers, groups and elements when using the Report & Graph Manager inside of STK. Data providers are also broken out in detail in STK Programming Help.

1. Return to STK Programming Help.
2. Select Library Reference in the Contents list.
3. Select Data Providers Reference.
4. On the Data Providers by Object page, you can spend a lot of time looking at an object's data providers, groups and elements.
5. Keep this page open.

Retrieve and View the Access Data in Excel

You created an access from LeoSat ( ) to Target1 ( ). Retrieve and view the access data in *Excel*.

1. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
2. Type or copy and paste the following Code at the bottom of the Sub Compute_Access().

```
 ' TASK 6
' Pull the data of interest out of the access object
Dim dp
Set dp = access.DataProviders("Access Data")
Dim results
Set results = dp.Exec(scenario.StartTime, scenario.StopTime)
accessNumber = results.DataSets(0).GetValues()
accessStart = results.DataSets(1).GetValues()
accessStop = results.DataSets(2).GetValues()
accessDuration = results.DataSets(3).GetValues()
' Create columns
Range("J1") = results.DataSets(0).ElementName
Range("K1") = results.DataSets(1).ElementName
Range("L1") = results.DataSets(2).ElementName
Range("M1") = results.DataSets(3).ElementName
' Write the data into Excel.
For i = 0 To UBound(accessNumber)
Range("J" & i + 2) = accessNumber(i)
Range("K" & i + 2) = accessStart(i)
Range("L" & i + 2) = accessStop(i)
Range("M" & i + 2) = accessDuration(i)
Next
```

3. Click Save STK_Excel_Workbook.xlsm ().

Understanding Task 6

Review the sections to learn about the different methods and properties in Task 6. Also, you will discover resources in the STK Programming Help to understand these methods and properties further.

Understing the DataProviders Property

The DataProviders Property returns the object representing a list of available data providers for the object. Learn more about the DataProviders Property (IAgStkAccess) in the STK Programming Help.

You use this property in line 4 of Task 6: **Set dp = access.DataProviders("Access Data")**

1. Return to STK Programming Help.
2. Select Access on the Data Providers by Object page. This page provides data providers available for the Access object.
3. Select Access Data in the Available Data Providers list. Access Data is the specified data provider.

4. Select Access in the Data Provider Variants list.
5. Look at the Data Provider Elements list to understand the following code.

Understanding the Exec() Method

The Exec Method computes the data within a specified Start and Stop time. Learn more about the Exec Method (IAgDataPrvTimeVar) in the STK Programming Help.

You use this method in line 6 of Task 6: **Set results = dp.Exec(scenario.StartTime, scenario.StopTime)**

Understanding the DataSets() Property and the GetValues Method

The DataSets() Property returns a collection of data sets within a specified interval. The GetValues retrieves an array of values of the elements in the dataset. In this instance, you're creating an access from the satellite to the target based on the scenario start and stop times.

When pulling data from the data provider elements list, it follows the sequence as seen in the list. For example, the first element, Access Number, is 0 (zero). Start Time is 1 (one), Stop Time is 2 (two) and Duration is 3 (three). In the code, accessNumber = results.DataSets(0).GetValues() simply means that you are you are pulling the results from the data provider (Access Data) and then the data provider element Access Number.

You can learn more about the DataSets Property (IAgDrInterval) and the GetValues Method (IAgDrDataSet) in the STK Programming Help.

You use this property and method in lines 7 to 10 of Task 6:

accessNumber = results.DataSets(0).GetValues()

accessStart = results.DataSets(1).GetValues()

accessStop = results.DataSets(2).GetValues()

accessDuration = results.DataSets(3).GetValues()

Understanding the ElementName Property

You use the DataSets() Property again, but you pair that property with a new one. The ElementName Property returns the name of the dataset. In Task 6, you are writing the element names as headers for columns J1 through M1. Learn more about the ElementName Property (IAgDrDataSet) in the STK Programming Help.

You use this property in lines 12 to 15 of Task 6:

Range("J1") = results.DataSets(0).ElementName

Range("K1") = results.DataSets(1).ElementName

Range("L1") = results.DataSets(2).ElementName

Range("M1") = results.DataSets(3).ElementName

Understanding filling the data in your table

You use a loop to add data to your table in line 17 to 21 in Task 6:

For i = 0 To UBound(accessNumber)

Range("J" & i + 2) = accessNumber(i)

Range("K" & i + 2) = accessStart(i)

Range("L" & i + 2) = accessStop(i)

Range("M" & i + 2) = accessDuration(i)

You will loop through all the data that it returns and write it to the appropriate cell.

Run all five sub routines

1. Bring the *Excel* workbook to the front.
2. Click **Launch STK** .
3. Click **Create Scenario** once STK opens.
4. After the scenario is created, click **Insert Targets** .
5. After the targets are inserted, click **Insert Satellite** .
6. After the satellite is inserted, click **Compute Access** . Your data will look similar to but different than the following example.

Format cells

Excel formats the start and stop times differently than SKT (e.g. day, month, year, time). You can select the cells containing the start and stop times, right-click on the selection, open the Format Cells option and set the format to dd mmm yyyy hh:mm:ss.000. Or you can use the following Code.

1. Bring the Microsoft Visual Basic for Applications - [Module 1 (Code)] window to the front.
2. Type or copy and paste the following Code at the bottom of the Sub Compute_Access().

```
ActiveSheet.Columns("K").Select
Selection.NumberFormat = "dd mmm yyyy hh:mm:ss.000"
ActiveSheet.Columns("L").Select
Selection.NumberFormat = "dd mmm yyyy hh:mm:ss.000"
```

3. Click Save STK_Excel_Workbook.xlsm (Ctrl+ S) ().
4. Bring the *Excel* workbook to the front.
5. Click **Compute Access** . Your data will look similar to but different than the following example.
6. Click Save ().

STK Excel Add-in

At the beginning of the tutorial, you downloaded and installed the STK Excel Add-in. Use that now.

1. Select the Add-ins tab.

Attaching the Excel Workbook to STK

Since you already have a running instance of STK, you can attach *Excel* to your scenario.

1. Select cell P in the *Excel* workbook. You want an empty cell plus three empty cells to the right. You will be choosing four elements.
2. Click Create or Attach to STK 12 () from the Custom Toolbars.
3. Select Attach to an existing STK 12 application instance.
4. Click **OK** to close the information window.
5. Click Extract Data From an STK 12 Data Provider () to open the Generate Data Provider Report tool.

Selecting your data providers

1. Select Satellite/LeoSat from the object list.
2. Expand (⊞) LLA State (Data Provider) in the Data Providers: list.
3. Expand (⊞) Fixed.
4. Select the following data provider elements.
    - Time
    - Lat
    - Lon
    - Alt
5. Use the default Start Time and Stop Time values.
6. Change Step (sec): to 3600. This will make the data easier to read.
7. Click **Generate** .
8. Close the Generate Data Provider Report tool.
9. You can expand columns P/Q/R/S in the *Excel* workbook if desired to see all of the data.

    You can see the Time, Latitude, Longitude, and Altitude of the satellite in one hour increments over the 24 hour analysis period.

Summary

You began by viewing a small portion of the STK Programming Help pages. Then you performed the following:

- Downloaded and installed the STK Excel Add-in.
- Opened *Excel* and enabled the Developer tab and turned on the STK Excel Add-in.
- After closing and then opening *Excel*, you created an *Excel* Macro-Enabled Workbook (*.xlsm).
- Created the first button and code which launched an instance of STK.
- Created the second button and code to create a new Scenario object with a run time of 24 hours and reset the scenario to the new analysis time period.

- In the *Excel* workbook, you created three columns: Target Name, Latitude and Longitude. In the Latitude column, you set a random number formula that entered values between -90 and 90. In the Longitude column, you set a random number formula that entered values between -180 and 180. You created 25 random targets.
- Created the third button and code which loaded 25 random Target objects into the scenario.
- Created a fourth button and code which propagated a LEO Satellite in the scenario.
- Created a fifth button and code that computed access between the Satellite object and a Target object and imported the access times into *Excel*.
- Changed the fifth button code to format the access times to match the format in STK.
- Used the STK Excel Add-in to import selected LLA Data Provider Elements concerning the Satellite object into *Excel*.

Part 14: 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 <STK install folder>\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. This is an introduction to Aviator so you will 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 marked read only ( ).
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 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. The second component of the STK software's *Aviator* capability is the 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 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 Blue Mesa VOR/DME, the aircraft will fly to Cones VOR/DME, and 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 Blue Mesa VOR/DME.
5. Right-click on HBU ( ) in the ARINC424 navaids( ) list.

   Note that HBU ( ) is marked 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 directly 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 Cones VOR/DME.
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 My Styles ( ) 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 (⊞) Flight Profile By Time ( ) in the Data Providers list.
3. Move ( ) the following 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 My Styles ( ).
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 15: 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 of space-based assets. In this scenario, you want to analyze seven days of data to determine the probability of your satellite having a conjunction with any other satellites or debris.

Solution

Use the Advanced Conjunction Analysis Tool (AdvCAT), which is a part of the STK's *Conjunction Analysis* capability, to carry out a close-approach analysis between a primary object (your satellite) and secondary objects (satellites possibly presenting a 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) .

Create a new scenario

1. Launch STK ( ).
2. In the Welcome to STK window, click **Create a Scenario**.
3. Enter the following in the STK: New Scenario Wizard:

Option	Value
Name:	STK_CAT

Location:	Default
Start:	1 Nov 2020 17:00:00.000
Stop:	+ 7 days

4. When you finish, click OK .
5. When the scenario loads, click Save (). STK creates a folder with the same name as your scenario and places it in the location specified above.
6. Verify the scenario name and location and click Save .

Prepare STK

For this scenario, you will not use the Timeline View nor the 2D Graphics window. Close them.

1. Close the Timeline View at the bottom of the STK window.
2. Close the 2D Graphics window.
3. Maximize the 3D Graphics window.

Obtain an archived satellite database

AdvCAT does not add any objects to the scenario. It merely propagates ephemerides for the primary and secondary objects that you select for the specified time frame. Use a local database for the analysis.

With AdvCAT, you can use either satellite objects propagated from your TLE database or analytical objects downloaded and propagated from the AGI server .

1. Select the scenario STK_CAT () and click properties ().
2. In the properties window, select the Basic - Database page.
3. Click **Update Database Files**.
4. When the Update Satellite Database window opens, set the following:

Option	Value
Option	Obtain Archived Database

Database Information	Specific Database: stkAllTLE
Database	Scenario Location (typically C:\Users\username\Documents\STK 12\STK_CAT)
Archive Date:	1 Nov 2020 17:00:00.000 UTCG

5. Click **Update** .
6. Click **OK** to close the Information window.
7. In the Update Satellite Database window, click **Close** .
8. Click **OK** to close STK_CAT's properties.

Primary satellite

The primary satellite used in the analysis is your satellite. For this scenario, your satellite is STARLINK-1611.

1. Using the Insert STK Objects tool ( ), insert a Satellite ( ) object using the From TLE File method.
2. When the Select a TLE File window opens, browse to the location of the downloaded TLE file named stkAllTLE.tce, typically C:\Users\username\Documents\STK 12\STK_CAT.
3. Select stkAllTLE.tce.
4. Click **Open** .
5. When the Question window opens, read the data and click **OK** or **Yes** . Be patient, as this can take a minute or two to load.

Modify the Satellite Database options

1. When the Insert From Satellite Database window opens, click **Modify** .
2. When the Satellite Database: TLE Source window opens, clear the **On propagation, automatically retrieve elements** checkbox.
3. Click **OK** .
4. When the Question pop up opens, click **OK** or **Yes** .
5. When the Insert From Satellite Database window repopulates, enter 46147 in the SSC Number field.
6. Click **Search** .

7. In the Results list, select STARLINK-1611.
8. Click **Insert** .
9. After STARLINK-1611 ( ) has propagated, click **Close** .

Add the AdvCAT Object to the Insert STK Objects tool

1. Bring the Insert STK Objects tool ( ) to the front.
2. Click **Edit Preferences** .
3. In the New Object list, select AdvCAT ( ).
4. Click **OK** .

Create an Advanced CAT object

The AdvCat object ( ) provides a convenient way for you to carry out close-approach analyses for multiple satellites and TLEs.

1. Using the Insert STK Objects tool ( ), insert an AdvCAT ( ) object using the Insert Default method.
2. Open AdvCAT1's properties ( ).

Selecting and defining analysis objects

Most of the Main page of the AdvCAT 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. The Secondary List contains satellites that present a potential risk of collision with, or unacceptably close approach to, satellites in the Primary List.

Analysis object attributes - primary satellite

In analyzing close approaches between a primary and secondary object, AdvCAT assigns to each a threat volume comprising an ellipsoidal shape enclosing the object. The following are key Analysis Object attributes:

- Fixed class specifies the dimensions of the threat volume ellipsoid.
- Tangential is the dimension of the threat volume ellipsoid along the X axis.
- Cross Track is the dimension of the threat volume ellipsoid along the Y axis.
- Normal is the dimension of the threat volume ellipsoid along the Z axis.

Since you're evaluating potential linear conjunctions and not using covariance settings, keep the default Class, Tangential, CrossTrack, and Normal values. Actual threat volumes would be set based on established ellipsoid and covariance screenings.

1. In the Main page of the AdvCAT Properties, go to the Primary List and move (→) Satellite/STARLINK-1611_46147 to the Chosen: list.
2. Use a typical Hard Body Radius of an A-Train satellite. Set the HardBodyRadius value to 3.0 m.
3. Click **Apply** .

Analysis Object attributes - secondary satellites

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 STK in the form of satellite database files and TLEs. AGI updates this data at least once and up to three times a day. Your analysis will check for potential conjunctions between STARLINK-1611 and the complete satellite database.

1. In the Secondary List, move (→) stkAllTLE.tce to the Chosen: list.
2. Set the HardBodyRadius value to 2.0 m.
3. Click **Apply** .

Using prefilters

The basic idea behind close-approach processing is to start with all cataloged orbiting objects and 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 prefilters are usually applied to limit the amount of ephemeris generation that is required.

1. In the AdvCAT Properties, select the Basic - Advanced page.
2. Note the selected Pre-Filters and their values.

- Apogee / Perigee: AdvCAT uses the apogee/perigee prefilter 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.

Compute possible conjunctions

When you compute possible conjunctions, any object threat volumes that fall within the threshold and prefilter distances appear as ellipsoids in the 3D Graphics window after creating a report.

1. In the AdvCAT properties, select the Basic - Main page.
2. Select the **Display Acknowledgment when done** checkbox.
3. Click **Compute** . Be patient. This could take a minute. There is a progress bar visible in the lower right corner of STK.
4. When the Advanced CAT message window appears, read the message and then click **OK** .

Close approach by minimum range report

Right now, you're interested in a few specific data providers. Remember, AdvCAT only reports objects with threat volumes that come within ten (10) kilometers and also pass the prefilters of your primary satellite.

1. In the Object Browser, right-click AdvCAT1 () and select **Report & Graph Manager**.
2. In the Styles section, in the Installed Styles folder, select **Close Approach By Min Range**.
3. Click **Generate** .
4. Look through the report and focus on Object Name, Time In (UTCG), Time Out (UTCG), Min Separation (km) and Min Range (km). The AdvCAT Data Provider is Events by Min Range. The following are the data provider elements used in the report:
    - 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 10 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 10 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.

5. When finished, close the report.

Create a custom report

Add three more **Events by Min Range** elements to your report. This will add situational awareness.

1. In the Installed Styles section, right-click **Close Approach By Min Range** and select Properties ().
2. In the Report Style window, under Report Contents, select a value in Section 6.
3. Under Data Providers, expand (⊞) the Events by Min Range folder.
4. Move () **Time of Close Appr** (TCA) to the Report Contents list.
5. Move () **Relative Velocity** to the Report Contents list.
6. Move () **Collision Probability** (Analytic) to the Report Contents list.
7. Click OK .
8. Read the Warning message and then click OK .

This added new elements to the custom report:

- 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.

Rename and run the report

1. In the Styles section, expand (⊞) the My Styles folder.
2. Rename the custom report to be "My Close Approach By Min Range".

3. Click **Generate** .

4. Look through the report.

Identify and insert a close approach

Focus on the first satellite in the report.

1. In the report, copy the SSC number (Satellite Catalog Number) of the first satellite in the report, for example 45540.

2. Using the Insert STK Objects tool ( ), insert a Satellite ( ) object using the From TLE File method.

3. When the Select a TLE File window opens, browse to the location of the downloaded TLE file named stkAllTLE.tce, which is typically C:\Users\username\Documents\STK 12\STK_CAT.

4. Select stkAllTLE.tce.

5. Click **Open** .

6. When the Question pop up opens, read the data and click **OK** or **Yes** . Be patient, this can take a minute or two to load.

Modify the Satellite Database options

1. When the Insert From Satellite Database window opens, click **Modify** .

2. When the Satellite Database: TLE Source window opens, clear **On propagation, automatically retrieve elements**.

3. Click **OK** .

4. When the Question window opens, click **OK** or **Yes** .

5. When the Insert From Satellite Database window repopulates, paste or type in the first satellite's SSC number from the report into SSC Number field, for example 45540.

6. Click **Search** .

7. In the Results list, select the satellite, for example STARLINK-1368.

8. Click **Insert** .

9. After the Satellite ( ) object has propagated, click **Close** .

Visualize the close approach

1. In the report, right-click on the first Time In (UTCG), select Time In, and then select Set Animation Time.
2. Bring the 3D Graphics window to the front.
3. Right-click on STARLINK-1611_46147 ( ) and select Zoom To.
4. In the Animation Tool Bar, click Decrease Time Step ( ) 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.

   All of your ellipsoids are green. When they enter the threshold of 10 kilometers, they turn yellow. When they intersect, they turn red. Other satellites that enter the threshold are represented by ellipsoids. They aren't analytical objects.
7. Click Start ( ) to animate the scenario.
8. Run the animation through the conjunction. After the conjunction, when the two satellites turn green again, click Pause ( ).

Set the animation time to the Time of Close Approach

1. Return to the report.
2. Set the animation time to the Time of Close Appr (UTCG) of the two satellites.
3. Return to the 3D Graphics window.
4. Using your mouse, zoom in so that you can get a good view of both satellites.
5. When finished, Reset ( ) the scenario.
6. In the 3D Graphics window, click Home View ( ).
7. Close the report and the Report & Graph Manager.

Visualize all the satellites

At this time, only the satellites having threat volumes that fell within the 10 kilometer threshold and passing the prefilters 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.
3. In the Visibility section are the primary ellipsoid and secondary ellipsoids that have conjunctions. Enable All.
4. Click OK .
5. In the Object Browser, uncheck both Satellite ( ) objects.
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, the satellites are represented by ellipsoids that are 20 by 10 by 5 kilometers in size!
8. Use your mouse to zoom out until you see the geostationary belt.

Summary

You began by updating your satellite database with an archived database of satellites that matched your scenario time period. Next, you inserted the primary satellite into the scenario. After inserting the AdvCAT object, you moved the primary satellite from the Primary List to the Chosen list. Then you moved the complete satellite database (stkAllTLE.tce) from the Secondary List to the Chosen list. You left the threat volumes and threshold at default values, 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 16: 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 an STK 12.7 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

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 STK's *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 STK scenario; then build from there.

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_Astrogator
Location:	Default
Start:	Default
Stop:	+ 5 days

4. Click **OK** when you finish.
5. Click Save () after the scenario loads. STK 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 () appears in the Insert STK Objects tool.

1. Click **Edit Preferences...** in the Insert STK Objects tool.
2. Select the Launch Vehicle check box when the Preferences dialog box opens.
3. Click **OK** to close the Preferences dialog box.

Creating a Launch Vehicle

Insert a Launch Vehicle () object.

1. Select Launch Vehicle () in the Insert STK Objects tool.
2. Select the Insert Default () method.
3. Click **Insert...** .
4. Right click on LaunchVehicle1 () in the Object Browser.

5. Select Rename in the shortcut menu.
6. Rename LaunchVehicle1 ( ) to LaunchToLEO.

Selecting a Simple Ascent propagator

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. Open LaunchToLEO's ( ) properties ( ).
2. Select the Basic - Trajectory page.
3. Ensure Propagator is set to SimpleAscent.
4. Enter 7.3 km/sec in the Burnout Velocity field. This will keep the resulting orbit near circular.
5. Click **Apply** to accept your change and to keep the Properties Browser open.
6. Select the 2D Graphics - Attributes page.
7. Change the Color: to teal.
8. Click **OK** to accept your change 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 Cape Canaveral.
5. Notice that the trajectory ends at burnout.

Inserting a satellite

Insert a Satellite ( ) object that you will use to create the satellite orbit.

1. Insert a Satellite ( ) object using the Insert Default ( ) method.
2. Rename Satellite1 ( ) to GEO_Sat.

Using Astrogator

Set GEO_Sat's ( ) propagator to *Astrogator*. You will use Astrogator to design your spacecraft trajectory. The STK Astrogator capability contains specialized analysis for interactive orbit maneuver and spacecraft trajectory design.

1. Open GEO_Sat's ( ) properties ( ).
2. Select the Basic - Orbit page.
3. Open the Propagator: shortcut menu.
4. Select Astrogator.

Setting up the Mission Control Sequence

The Mission Control Sequence (MCS) 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.

The MCS is defined by selecting and organizing MCS Segments in a manner that produces your desired trajectory. By default, an Astrogator satellite's MCS contains two segments: an Initial State ( ) segment and a Propagate ( ) segment.

Deleting the Initial State segment

Since you are modeling a Launch Vehicle ( ), remove the Initial State ( ) segment. You will replace it with a Follow ( ) segment .

1. Select Initial State ( ) in the MCS.
2. Click Delete Segment ( ) in the MCS toolbar.
3. Click **Yes** to confirm deletion.

Adding a Follow segment

Use the 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 ( ) MCS.
2. Select Insert Before... .
3. Select Follow ( ) in the Segment Selection dialog box.
4. Click **OK** to close the Segment Selection dialog box.
5. Select Follow ( ) in the MCS.
6. Select the General tab.
7. Click the Leader ellipsis ( ).
8. Select LaunchToLEO ( ) in the Select Leader dialog box.
9. Click **OK** to close the Select Leader dialog box.
10. Open the Joining: shortcut menu in the Additional Options frame.
11. Select Join 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. Also, the separation parameter is automatically set to Separate at End of Leader's Ephemeris.

Setting the Fuel Tank configuration

Set GEO_Sat's ( ) fuel mass and its maximum fuel mass.

1. Select the Fuel Tank tab.
2. Set the following in the order shown:

Option	Value
Maximum Fuel Mass	6000 kg
Fuel Mass	5000 kg

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

Running the Mission Control Sequence

Run the Mission Control Sequence 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 (0.5 day).
3. Click Run Entire Mission Control Sequence ( ) in 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 engine model

Use a realistic engine model in order to produce accurate results.

1. Return to GEO_Sat's ( ) propterties ( ).
2. Click Component Browser ( ) in the MCS Toolbar.
3. Open the Show Component Type shortcut menu.
4. Select Astrogator Components.
5. Select Engine Models ( ) 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 ( ) in the Engine Models toolbar.
8. Type Test Engine in the Name: field when the Field Editor dialog box opens.
9. Click OK to close the Field Editor dialog box.
10. Double-click Test Engine ( ) in the Engine Models list.

Modifying Constant Thrust and Isp

Modify the Constant Thrust and Isp engine model to specify the thrust and Isp for your engine.

1. Double-click Thrust when the Test Engine dialog box opens.
2. Enter 13500 N in the Real Number: field when the Real Number Field dialog box opens.
3. Click **OK** to close the Real Number Field dialog box.
4. Double-click Isp in the Test Engine dialog box.
5. Enter 2000 s in the Real Number: field when the Real Number Field dialog box opens.
6. Click **OK** to close the Real Number Field dialog box.

Adding a component to your collection

You can save a custom-built component and make it available in other scenarios.

1. Click **Add to Collection** in the Test Engine dialog box.
2. Click **Close** to close the Component Browser.

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.

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 ( ) to Prop to TOI.

Updating the Stopping Condition

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 frame.
2. Select AscendingNode ( ) in the New Stopping Condition dialog box.
3. Click OK to close the New Stopping Condition dialog box.
4. Select Duration in the Stopping Conditions frame.
5. Click Delete ( ) in 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 accept your changes and to keep the Properties Browser open.
8. Click Run Entire Mission Control Sequence ( ) in the MCS toolbar.

Defining a Target Sequence - Start Transfer

Insert a Target Sequence ( ). You will use a Target Sequence ( ) to calculate the Delta-V required to move GEO_Sat ( ) into a transfer orbit.

1. Return to GEO_Sat's ( ) properties ( ).
2. Right-click Prop_To_TOI ( ) in the MCS.
3. Select Insert After... in the shortcut menu.
4. Select Target Sequence ( ) in the Segment Selection dialog box.
5. Click OK to close the Segment Selection dialog box.
6. Rename Target Sequence ( ) to Start Transfer.

Inserting a Maneuver segment

Insert a Maneuver ( ) segment. This first maneuver Delta-V will be solved to achieve a particular altitude at the end of the transfer ellipse.

1. Right-click Start Transfer ( ) in the MCS.
2. Select Insert After... in the shortcut menu.
3. Select Maneuver ( ) in the Segment Selection dialog box.
4. Click OK to close the Segment Selection dialog box.

   The Maneuver ( ) Segment appears below the Return ( ) Segment.

5. Drag and drop the Maneuver ( ) Segment below Start Transfer ( ).
6. Right click on Maneuver ( ) in the MCS.
7. Select Properties... in the shortcut menu.
8. Type TOI in the Name: field when the Edit Segment dialog box opens.
9. Open the Color: shortcut menu.
10. Select white.
11. Click OK to accept your changes and to close the Edit Segment dialog box.

    A Maneuver ( ) segment isn't visible in the 2D and 3D Graphics windows. Propagate ( ) segment colors are visible. By making Maneuver ( ) segments white is just a way to focus on Propagate ( ) segment colors.

Insert a Propagate Segment

Insert a Propagate ( ) segment that will be used to determine the stopping condition.

1. Right click on TOI ( ) in the MCS.
2. Select Insert After... .
3. Select Propagate ( ) in the Segment Select dialog box.
4. Click OK to close the Segment Select dialog box.

5. Right click on Propagate ( ) in the MCS.
6. Select Properties... in the shortcut menu.
7. Enter Transfer in the Name: field when the Edit Segment dialog box opens.
8. Open the Color: shortcut menu.
9. Select yellow.
10. Click OK to close the Edit Segment dialog box.
11. Click Apply to accept your changes and to keep the Properties Browser open.

Setting the engine model

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 frame.
4. Select Test Engine ( ) in the Select EngineModel dialog box.
5. Click OK to close the Select EngineModel dialog box.

Selecting the control parameter

Select Delta-V Magnitude as the independent variable.

1. Select the Attitude tab.

   The default Attitude Control: is Along Velocity Vector (body X).

2. Click 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.

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

Propagating to apoapsis

Update Transfer's ( ) Stopping Condition to stop at apoapsis.

1. Select Transfer ( ) in the MCS.
2. Click New... ( ) in the Stopping Conditions frame.
3. Select Apoapsis ( ) in the New Stopping Condition dialog box.
4. Click **OK** to close the New Stopping Condition dialog box.
5. Select Duration in the Stopping Conditions frame.
6. Click Delete ( ) in the Stopping Conditions toolbar.
7. Click **Apply** to accept your changes and to keep the Properties Browser open.

Selecting the results variable

Set the radius of the orbit, R Mag, as the equality constraint. Astrogator will use R Mag to determine the required Delta-V magnitude.

1. Select Transfer ( ) in the MCS.
2. Click **Results...** at the bottom of the MCS.
3. Expand (⊞) Spherical Elems ( ) in the Available Components: list when the User - Selected Results - Transfer dialog box opens.
4. Select R Mag ( ).
5. 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 close the User - Selected Results - Transfer dialog box.

Setting up the targeter

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: shortcut menu.
3. Select Run active profiles.

   Run active profiles runs the mission control sequence allowing the active profiles to operate.
4. Select Differential Corrector in the Profiles frame.
5. Click Properties... ( ).

Setting the control parameter

Use Delta-V Magnitude as the control parameter (independent variable).

1. Select the ImpulsiveMnvr.Pointing.Spherical.Magnitude Use check box in the Control Parameters frame when the Differential Corrector dialog box opens.

Setting the equality constraints (Results)

Use R_Mag as the equality constraint (dependent variable), and set the radius of orbit goal to be 42238 km.

1. Select the R_Mag Use check box in the Equality Constraints (Results) frame.
2. Click the Desired Value cell.
3. Enter 42238 km in the Desired Value cell.
4. Click OK to close the Differential Corrector dialog box.
5. Save ( ) your scenario.

Running the Entire Mission Control Sequence

1. Click Run Entire Mission Control Sequence ( ) in the MCS toolbar.
2. When complete, look at the top of STK.

   You will see a message informing you whether or not running the entire MCS converged or didn't converge.
3. Look at the StartTransfer.Differential Corrector data dialog box.

   This shows you data based on running the active profile.

4. Close the StartTransfer.Differential Corrector data dialog box.
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.

Defining a Target sequence: Finishing the transfer

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.

1. Return to GEO_Sat's ( ) properties ( ).
2. Right-click the bottom Return ( ) Segment in the MCS.
3. Select Insert Before... in the shortcut menu.
4. Select Target Sequence ( ) in the Segment Selection dialog box.
5. Click OK to close the Segment Selection dialog box.
6. Rename Target Sequence ( ) to 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 Finish_Transfer ( ) in the MCS.
2. Select Insert After... in the shortcut menu.
3. Select Maneuver ( ) in the Segment Selection dialog box.
4. Click OK to close the Segment Selection dialog box.

   The Maneuver ( ) Segment appears below the return ( ) Segment.
5. Drag and drop the Maneuver ( ) Segment below the Finish_Transfer ( ).
6. Right click on Maneuver ( ) in the MCS.

7. Select Properties... in the shortcut menu.
8. Type SOI in the Name: field when the Edit Segment dialog box opens.
9. Open the Color shortcut menu.
10. Select white.
11. Click OK to close the Edit Segment dialog box.

Inserting a Propagate segment

Insert a Propagate ( ) Segment will be used to determine the stopping condition.

1. Right click on SOI ( ) in the MCS.
2. Select Insert After... .
3. Select Propagate ( ) in the Segment Select dialog box.
4. Click OK to close the Segment Select dialog box.
5. Right click on Propagate ( ) in the MCS.
6. Select Properties... in the shortcut menu.
7. Type Prop 1 Rev in the Name: field when the Edit Segment dialog box opens.
8. Open the Color: shortcut menu.
9. Select blue.
10. Click OK to close the Edit Segment dialog box.
11. Click Apply to accept your changes and to keep the Properties Browser open.

Setting the engine model

Use the Test Engine model you updated earlier.

1. Select SOI ( ) in the MCS
2. Select the Engine tab.
3. Click the Engine Model ellipsis ( ) in the Propulsion Type frame.

4. Select Test Engine ( ) in the Select EngineModel dialog box.
5. Click **OK** to close the Select EngineModel 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: shortcut menu.
3. Select Thrust Vector.
4. Click the X (Velocity) target ( ).
5. Click the Y (Normal) target ( ).
6. Click **Apply** to accept your changes and to keep the Properties Browser open.

Propagating for one day

Update the Prop 1 Rev ( ) Duration Stopping Condition to stop after one day.

1. Select Prop 1 Rev ( ) in the MCS.
2. Enter 86400 sec in the Trip: field in the Stopping Conditions frame.
3. Click **Apply** to accept your change and to keep the Properties Browser open.

Selecting the results variable

Set Eccentricity and Inclination as the equality constraints.

1. Select Prop 1 Rev ( ) in the MCS.
2. Click **Results...** at the bottom of the MCS.
3. Expand (⊞) Keplerian Elems in the Available Components: list when the User - Selected Results - Prop 1 Rev dialog box opens.
4. Move ( ) Eccentricity ( ) to the selected components list.

5. Move (→) Inclination () to the selected components list.
6. Click OK to close the User - Selected Results - Prop 1 Rev dialog box.

Setting up the targeter

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: shortcut menu.
3. Select Run active profiles.

Setting the control parameter

Use Delta-V vector's Cartesian X (Velocity) and Y (Normal) as the Control Parameters (independent variables).

1. Select Differential Corrector in the Profiles frame.
2. Click Properties... () in the Profiles toolbar.
3. Select the ImpulsiveMnvr.Pointing.Cartesian.X Use check box in the Control Parameters frame when the Differential Corrector dialog box opens.
4. Select ImpulsiveMnvr.Pointing.Cartesian.Y Use check box.

Set 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 Eccentricity Use check box in the Equality Constraints (Results) frame.
2. Enter 0.001 in the Tolerance: field.
3. Select the Inclination Use check box.
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: shortcut menu in the Scaling frame.

8. Select By tolerance.
9. Click OK to close the Differential Corrector dialog box.
10. Click Apply to accept your changes and to keep the Properties Browser open.
11. Save ( ) your scenario.

Running the Entire Mission Control Sequence

1. Click Run Entire Mission Control Sequence ( ) in the MCS toolbar.
2. Look at the Finish Transfer.Differential Corrector: Finished data window.
3. Close both the Start Transfer.Differential Corrector:Finished and Transfer.Differential Corrector: Finished data 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 iterations

Remove the visible iterations in the 3D Graphics window.

1. Return to GEO_Sat's ( ) properties ( ).
2. Right-click the Bottom Return ( ) Segment.
3. Select Insert Before... in the shortcut menu.
4. Select Propagate ( ) in the Segment Selection dialog box.
5. Click OK to close the Segment Selection dialog box.
6. Click Run Entire Mission Control Sequence ( ) in the MCS toolbar.
7. Close both the Start Transfer.Differential Corrector:Finished and Transfer.Differential Corrector: Finished data windows.
8. Bring the 3D Graphics window to the front.

Generating an MCS Segment Summary 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 ( ) in the MCS Toolbar.
4. Look at the data in the report.
5. When finished, close ( ) the summary report.
6. Select Finish_Transfer ( ).
7. Click Summary ( ) in the MCS Toolbar.
8. Look at the data in the report.
9. When finished looking through the report, close any reports, tools and properties you still have open.
10. Save ( ) your work.

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 17: 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.

Important: Required Capability Install: This lesson requires an additional installation for the STK software's *EOIR* capability. The *EOIR* installer is included in the STK Premium software download, but requires a separate installation process. Read the Readme.htm found in the STK software install folder for installation instructions. You can obtain the necessary install by visiting **https://support.agi.com/downloads** or calling AGI support.

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

Capabilities covered

This lesson covers the following Ansys Systems Tool Kit® (STK®) digital mission engineering software capabilities:

- STK Pro
- Electro-Optical Infrared Sensor Performance (EOIR)
- STK SatPro

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 need to model the telescope specifications and take cloud cover, 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 Maui Space Surveillance Complex (MSSC) 1.6-meter (m) telescope at the Air Force Maui Optical Station (AMOS) observatory in Maui, Hawaii, that tracks a polar satellite in LEO.

What you will learn

Upon completion of this tutorial, you will understand:

- How to configure Sensor objects to use the EOIR type
- How to create an EOIR sensor scene
- How to view data in the EOIR Scene Visual Details dialog box
- How to create a custom EOIR signal-to-noise (SNR) graph

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** .

Inserting MSSC as a Facility object

Add MSSC as a Facility object to your scenario.

Inserting Facility object

Use the Insert STK Objects tool to insert a Facility object from the Standard Object Database.

1. Select Facility () in the Insert STK Objects tool.
2. Select the From Standard Object Database () method.
3. Click **Insert...** .

Selecting an MSSC Facility object

Add the MSSC_1_6m Facility object to your scenario.

1. Type MSSC in the Name field in the Search Standard Object Data dialog box.
2. Click **Search** .
3. Select MSSC 1.6m in the Results list that uses the Local Database Data Source.
4. Click **Insert** .
5. Click **Close** to close the Search Standard Object Data dialog box.

Opening the EOIR toolbar

Open the EOIR toolbar.

1. Extend the View menu.
2. Select Toolbars in the shortcut menu.
3. Select EOIR in the second shortcut menu.

Inserting a Sensor object

Add a sensor to MSSC_1_6m.

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 ( ) to Telescope.

Using the EOIR sensor type

Start by setting sensor type to EOIR.

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 for the Sensor Type.
5. Click Apply to accept your change and to keep the Properties Browser open.

Setting the spatial properties

You are modeling the Advanced Electro-Optical System (AEOS) telescope. Start by setting the Spatial properties of the Sensor object. Use the spatial properties to define the total field-of-view angles and the number of pixels on the sensor detector. The default input setting is Field-of-View and Number of Pixels. The Related Detector Parameters and Instantaneous Field of View values are then based on the Spatial and Optical properties and are updated when you apply your changes. These are read-only fields.

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

3. Click Apply .

Setting the Spectral Properties

Define the spectral range of the Sensor object. 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 time.

1. Select the Spectral tab on the Basic - Definition page.
2. Set the following parameters in the Spectral Band Edge Wavelengths panel (you have to set High first):

Option	Value
High	1.0 um
Low	0.7 um

   The telescope observes the long infrared waveband.

3. Click Apply .

Setting the Optical Properties

Next, set the Optical properties. The Image Quality property models wave front error through the optics. The Negligible Aberrations setting introduces 7% wave front error.

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 cm

4. Select Negligible Aberrations for the Image Quality.

5. Click **Apply** .

Setting the Radiometric Properties

Radiometric properties define the noise floor and the saturation ceiling. You can 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 in the Sensitivity panel:

Option	Value
Integration Time	100
Equivalent Value	1e-16

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 (mrad) in the Line of Sight Jitter field in the Jitter panel.

   This introduces a Gaussian vibration along the sensor boresight.

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

Setting the 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 EOIR sensor scenes

Now you are ready to generate sensor scenes. These scenes accurately portray sensor images for the processing level selected.

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.

3. Right-click 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.

   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.

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. Cclose 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.

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

Option	Value
Type	Circular
Satellite Name	LEO_Sat
Inclination	98 deg
Altitude	700 km
RAAN	24 deg

3. Click OK to accept your changes and to close the Orbit Wizard.

Viewing the LEO satellite and the ground site

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.

   When the target satellite passes over the AMOS facility, AMOS is in darkness while the satellite is illuminated. This scenario gives good lighting conditions for imaging.

7. Click Reset ( ) when finished.

> **Note:** Although this tutorial is a ground-to-space example, it is possible to host an EOIR sensor on air and space vehicles. The work flow of setting up an *EOIR* sensor model is similar for all supported STK objects.

Examining the LEO satellite's basic EOIR shape

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

Add the satellite to 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 report

Create an access report between Telescope and LEO_Sat.

1. Right-click on Telescope () in the Object Browser.
2. Select Access... ().
3. Select LEO_Sat () in the Associated Objects list.
4. Click Compute .
5. Click Advanced... to open the Advanced Options dialog box.
6. Clear Use Light Time Delay in the Light Time Delay panel.

   Light time delay is not used in EOIR analysis.
7. Click OK to close the Advanced Options dialog box.

Generating an Access Report

Generate an Access report.

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 second shortcut menu.

   This sets the Current Scenario Time in the Animation toolbar to the first access time.
5. Close () the access report.
6. Close () the Access tool.

Creating the EOIR Sensor Scene

Create 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** .

Performing EOIR sensor scene analysis

You can view the EOIR sensor scene details.

1. Move the slider bar to the right in the Brightness panel to help see LEO_Sat ( ) against the background stars.
2. Click around the scene to view information on each pixel.
3. Click one of the stars and the target satellite to get more details on those objects.
4. Decrease ( ) the animation Time Step to 0.5 seconds.
5. Animate ( ) the scenario until you see the satellite come into the scene.
6. Click Pause ( ) to stop the scenario animation.

   The dot that represents the satellite moves across the scene while the stars stay relatively still.
7. Click **Close** to close the EOIR Scene Visual Details dialog box when finished.
8. Close ( ) the EOIR sensor scene.

Creating custom graphs for EOIR Sensors

EOIR 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... ( ) to open the Report & Graph Manager.
3. Select My Styles ( ) in the Styles panel.
4. Click Create new graph style ( ) in the Styles toolbar.
5. Type 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: are 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 ( ) EOIR Sensor To Target Metrics in the Data Provider list.
2. Move ( ) In-band target irradiance to the Y Axis list.
3. Move ( ) Signal to noise ratio to the Y2 Axis list.
4. Set the following parameters:

Option	Name
Time Axis Title	Time
Y Axis - Axis	In-band target irradiance

Y2 Axis - Axis	Signal to Noise Ratio (SNR)

5. Select EOIR Sensor to Target Metrics-In-band target irradiance in the Y Axis list
6. Click **Units...** below the Y2 Axis box to open the Units dialog box.
7. Clear Use Defaults.
8. Select Power in the Dimension column.
9. Select Watts (W) in the New Unit Value list.
10. Click **OK** to close the Units dialog box.

Setting the step size

Set the graph's step size.

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.

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 you are able to see the target in the scene. Keep the graph open.

Defining the EOIR 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: 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.
   - MODTRAN Atmosphere: MODTRAN is a community standard, and the MODTRAN-derived atmosphere model is one of the highest-fidelity atmospheric models available in EOIR.
4. Select the Simple Atmosphere option.
5. Set the following parameters:

Option	Value
Aerosol Models	Maritime
Visibility	40 (km)
Humidity	70 (%)

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 Click Refresh (F5) ( ) in the graph toolbar.

Degradation is due to the atmosphere effects. Keep the graph open.

Turning off the Simple 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.

Redefining the EOIR Properties

Define the material and shape properties of your satellite and the properties of your EOIR sensor.

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

Notes:

- LeoComm: based on the 3D model iridium.glb.
- Static: STK applies this temperature to the entire shape. This then applies for the entire EOIR scene time period.
- Aluminum MLI: multi-layer insulation.

4. Click **OK** .

Regenerating the EOIR Sensor Scene

Regenerate the EOIR scenario scene with the updated 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.
7. Select the BGRY option in the Color Map panel.
8. Click **Apply** .

Performing EOIR sensor scene analysis

You can view the EOIR sensor scene details.

1. Move the slider bar to the right in the Brightness panel to help see LEO_Sat ( ) against the background stars.
2. Click around the scene to view information on each pixel.
3. Click one of the stars and the target satellite to get more details on those objects.

4. Animate ( ) the scenario until you see the satellite come into the scene.
5. Click Pause ( ) to stop the scenario animation.
6. 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 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. In this analysis, the satellite is holding a nadir-pointing attitude profile. Because of this, the sensor sees a near-constant satellite cross-section during the overhead pass.

Analyzing the Light Signature

Now you will analyze the light signature of a tumbling satellite to compare to the nadir pointing satellite.

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 the 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. Click Start ( ) to animate the scenario.
5. Click Pause ( ) to stop the scenario animation when finished.

Changing the scaling of the Y axis

Change the Y axis scale for In-band target irradiance.

1. Return to the custom graph.
2. Double-click the In-band target irradiance title to open the Sensor Telescope dialog box.
3. Select the Axis tab.
4. Enter the following in the Y Axis panel:

Option	Value
Min	0
Max	3e-010

5. Click **OK** to close the Sensor Telescope dialog box.
6. Click Click Refresh (F5) ( ) in the graph toolbar.
7. Enter 0.1 sec in the Step field.
8. Select the Enter key. Be patient.

   The plot is jagged, thus confirming that the spacecraft is tumbling. As the spacecraft rotates, various panels reflect varying amounts of light and create an jagged plot.

Summary

This was an introduction to the STK software's *EOIR* capability. You modeled, simulated, and analyzed a Maui Space Surveillance Complex (MSSC) 1.6 meter (m) telescope at the Air Force Maui Optical Station (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 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 EOIR configuration, you applied MODTRAN 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*, *Astrogator*, *Aviator*, *Communications*, *Coverage*, and *Radar* capabilities.

What's in the test?

The STK Master Certification test consists of four scenario development exercises. There are multiple-choice questions for each exercise. You have 14 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 a personalized STK Master Certification certificate and a custom STK Master Certification pin. Register now to take the Level 2 - STK Master Certification test at **https://www.agi.com/training/#cert**.

Upon registration for the STK Master Certification test, you will receive an email confirmation with an attachment for a 14-day demo license. This license provides you access to all the capabilities needed to complete the certification.