

STK Level 1 and Level 2 Training Manual

STK VERSION 12.8, MAY, 2024

© ANSYS, INC AND/OR ITS AFFILIATED COMPANIES. ALL RIGHTS RESERVED. UNAUTHORIZED USE,
DISTRIBUTION, OR REPRODUCTION IS PROHIBITED. 2024

Table of Contents

Table of Contents	2
Level 1 - Beginner Training	4
Part 1: Build Scenarios	1
Part 2: Objects and Properties	8
Part 3: Access Reports and Graphs	27
Part 4: Movies and Visual Data Files	38
Part 5: Introduction to Connect	48
Become Level 1: STK Certified	56
STK Level 2 - Advanced	58
Part 6: Introduction to the Pro Capabilities	61
Part 7: Customize Analysis with Analysis Workbench	76
Part 8: Compute Coverage Over Regions	92
Part 9: Introduction to the AzEl Mask Tool and Volumetrics	111
Part 10: Perform Trade Studies with Analyzer	133
Part 11: Introduction to Communications	144
Part 12: Introduction to Radar	156
Part 13: Integrating STK with MATLAB	173
Part 13: Integrating STK with Python	195
Part 13: Integrating STK with Excel	203
Part 14: Model Aircraft Missions with Aviator	231
Part 15: Introduction to the Advanced CAT Tool	248
Part 16: Design Trajectories with Astrogator	258
Part 16: Ground-based SSA with EOIR	275
Become Level 2: STK Master Certified	292

Level 1 - Beginner Training

STK Level 1 - Beginner training is designed to familiarize first-time users with STK workflow and provide a basic understanding of STK software capabilities. This training is designed to enable you to model and incorporate your own systems and missions throughout the lessons.

Evaluation license

This training covers the basics of STK that can be completed with an Evaluation license.

The Level 1 - Beginner training is a series of tutorials designed to get you started using STK.

Once you have completed these tutorials, you will be ready to take the free Level 1 STK Certification test! Visit www.agi.com/training-and-certification#cert/.

Tutorial	Capabilities	Required License	Required Install
Lesson One: Build Scenarios (Design Reference Missions (DRM)) Learn how to create a scenario in STK.	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 STK.	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 STK.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro



Note: For the STK 10 version of this training, visit <http://help.agi.com/StartTraining/StartTraining1013.htm>

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.

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\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 reset 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.

Problem statement

Engineers and operators need to quickly add realistic analytical and visual properties to objects in STK. 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 STK to insert STK objects into your scenario. Then update the object's properties to define its characteristics 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 () (e.g. STK_Objects_Properties). 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 and you can change these properties later. For instance, when this training says to "Insert a Default Object," that means insert an object using the default method. You should pick the method for the demands of your scenario, as we will show later in this training.

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.

From Standard Object Database

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 using Local Database, you would need to ensure you've updated your satellite database.

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

View 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 () .

View 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, make any adjustments to the time step.
6. Click Start () to animate the scenario.
7. When finished, click Reset () .

ISS_ZARYA_25544's Properties

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

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

Orbit Wizard

The Orbit Wizard is a satellite-level tool designed to assist you in creating any one of several standard orbits, or designing your own satellite orbit. The configurable options available 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 if desired.
8. When finished, click Reset () .

RGT_Sat's Properties.

1. Right-click on RGT_Sat () in the Object Browser.
2. Select properties () .
3. Note that the propagator is set to J2 Perturbation.

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

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. Enable Body Axes Show option.
5. Change the Body Axes color if desired.
6. Click **Apply** .

View 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.

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. Insert an Aircraft () object using the Insert Default () method.

Renaming objects

It's usual to rename objects in STK. If you choose not to rename your objects, STK 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.
-

Aircraft Object's Properties

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

Clicking on 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.

Modify 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** .

Insert 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**.

View 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 (.

Modify 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.

Modify 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.

Modify a Waypoint's Speed

Modify 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.

Modify 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**.

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 STK. 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** .

Set 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.

View 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

STK is set at the Standard attitude option which allows you define a vehicle's attitude profile.

Attitude changes can be applied to all moving objects in STK (e.g. Aircraft, Ground Vehicles, Missiles, Satellites, Ships).

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

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: shortcut menu in the Basic frame.
2. Select Coordinated Turn.
3. Leave the Time Offset at 10 sec.
4. Click **Apply** .

View in 3D

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

2D Graphics properties

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** .

3D Graphics properties

3D Graphics Properties - Vector is used 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.

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** .

View in 3D

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

Animate 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.

3D Graphics Model

The 3D Graphics - Model page is used to specify a model to represent a given vehicle, facility, place, or target in the 3D Graphics window.

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: ellipses () in the Model frame. All the models shown in the File dialog box come with the STK install.
4. Select any model (DAE File or Simulink Model) you'd like to use in the scenario.
5. Click **Open** .
6. Click **OK** .
7. Bring the 3D Graphics window to the front. You have a new model type representing the Aircraft () object.

Area Target object

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.

Define the Area Target

Define 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** .

View 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

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.

Enter 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.

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.

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** .

View 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.
-

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.

Move Place1 by Clicking on 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.

Set 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** .

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 shortcut menu.
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.

Fix 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 hand corner of the building.
4. Click Object Edit Start/Accept () to accept the change.

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.

Sensor Orientation

1. Open WS_Sensor's () properties () .
2. Select the Basic - Definition page.
3. In the Simple Conic frame, enter 90 deg in the Cone Half Angle: field.
4. Click **Apply**.

Basic Constraints

1. Select the Constraints - Basic 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. In this instance, 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. Select the Max: check box in the Range frame.
4. Enter 1000 km Max: field.

Range is measured as the distance between the two objects.
5. 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.

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 basic 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 - Basic 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 - Basic page.
2. Select the Max: check box in the Range frame.
3. Enter 1000 km in the Max: field.
4. 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.

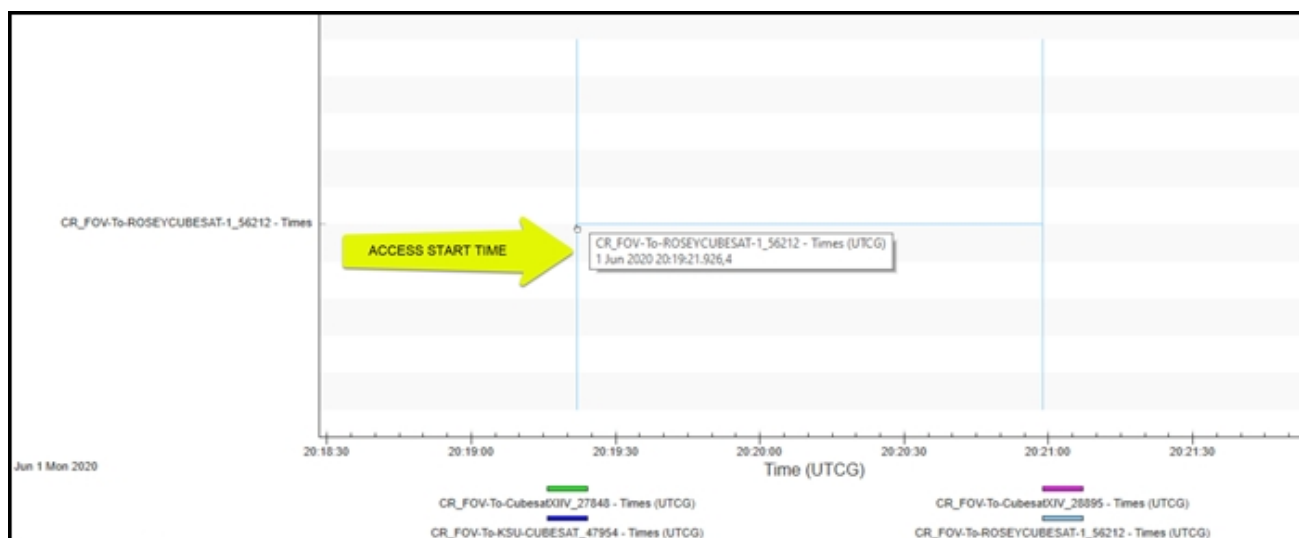
Generating an Access Graph

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 - Basic page when the Properties Browser opens.
 3. Enter 1500 km in the Range frame's Max: field.
 4. 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.

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 to 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 on 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 on PhotoMission () in the Object Browser.
 2. Select Properties ()
 3. Select the Basic - Route page when the Properties Browser opens.
 4. Open the Reference: shortcut 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 on the 2D Graphics window method to create waypoints for PhotoMission (). By clicking on 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. We'll fix that shortly.

5. Return to PhotoMission's (🚁) properties by clicking on the PhotoMission... tab (📄) below the 2D Graphics window.

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

Adjust the altitude, speed, and turn radius using **Set All...** .

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: shortcut 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.mdl 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 one of many.

The second point (waypoint) is when PhotoMission () is overhead 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 on 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:

- Let's say 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**
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.

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 shortcut menu.
 3. Select Movie Timeline in the second shortcut menu.
 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. If you follow the sequence from top to bottom, you will make a fairly effortless movie.

Filename and format

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 the 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 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: shortcut menu in the Choose the resolution of the recorded movie: frame.
2. Select Large.
3. Click **Next**.

Video time and length

Take your time on this page. It's important to follow the following 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 Movie time range: Start time: field.
4. Highlight the time and units in the Movie playback length Shorter / Longer field.
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 Movie time range: End Time: to twenty (20) seconds past the Start time.
7. Select the black lock () check box.
8. Click **Next**.

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. Anti-aliasing is a technique for diminishing jagged - stair step-like lines that you want to be smooth.

1. Extend the Anti-aliasing: pull-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 if required. 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

Using the Movie Wizard, when you create the start and stop time of your movie, it 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 to close the Properties Browser.

Visual Data File (VDF)

You can convert your STK 3D scenarios to Visual Data (.vdf) files that can be:

- Displayed in STK Viewer.
 - Loaded and edited in STK Viewer.
 - Loaded, edited and saved 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 Save in: 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.

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

Now that you have completed Fundamentals training, you are 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 STK.

What's in the Test?

The STK Certification test consists of one exercise scenario and you have 14 days from registration date to complete Certification. The following objectives are tested:

- Model Your Systems - KML, Aircraft, Satellite, Sensor, Constraints
- Analyze Your Systems - Access Tool, Report & Graph Manager, Quick Reports
- Visualize Your Data - 3D Models, Stored Views, Timeline View
- Extend STK - Connect and Object Model
- Share Your Work - VDF, STK Data Federate, Movies, Snapshots

If you pass your L1 STK Certification, you will receive a personalized Level 1 STK Certification certificate, an Ansys lanyard, and a custom STK pin. Register now to take the Level 1: STK certification test (<http://www.agi.com/training/#cert>).



STK Level 2 - Advanced

STK Level 2 - Advanced training builds off of STK Level 1 - Beginner. You will take simulations from STK Fundamentals a step further with advanced analysis capabilities and tools to quantify and measure mission effectiveness.

Evaluation License

You can obtain the necessary license for the training by visiting <http://licensing.agi.com/stk/evaluation> or contacting AGI Support at support@agi.com or 1-800-924-7244.

The Level 2 - Advanced Training is a series of tutorials designed to take a user through STK's advanced capabilities.

Once you have completed these tutorials, you will be ready to take the level 2 STK Master Certification test! Visit www.agi.com/training-and-certification#cert/ (free for U.S. and Canada). Upon registration for the level 2 Master Certification, 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. This license is for use with STK 12.2 or older. If you are using STK 12.3 or newer to complete the STK Certification test, please contact AGI Support at support@agi.com or 1-800-924-7244 for an updated license file.

Optional third-party programs

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

Tutorial	Capabilities	Required License	Required Install
Introduction to the Pro Capabilities Learn how to add terrain and imagery to a scenario.	STK Pro	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Customize Analysis with Analysis Workbench Learn how to use STK's Analysis Workbench capability to build custom geometric, temporal, and logical operations through STK.	STK Pro, <i>Analysis Workbench</i>	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, <i>Coverage</i>	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, <i>Analysis Workbench, Coverage</i>	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Perform Trade Studies with Analyzer Learn how to use STK's Analyzer capability to automate STK trade studies and parametric analyses.	STK Pro, <i>Analyzer</i>	STK Premium (Air or Space) or STK Enterprise	STK Pro + STK Premium
Introduction to Communications Learn how to define and analyze detailed communication systems.	STK Pro, <i>Communications</i>	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, <i>Radar, Coverage</i>	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Integrating STK with MATLAB Learn how to control STK through an external application like MATLAB.	STK Pro, <i>Integration</i>	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Integrating STK with Python Learn how to control STK through an external application like Python.	STK Pro, <i>Integration</i>	STK Pro, STK Premium (Air or Space), or STK Enterprise	STK Pro
Integrating STK with Excel Learn how to control STK through an external application like Excel.	STK Pro, <i>Integration</i>	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, <i>Aviator</i>	STK Premium (Air) or STK Enterprise	STK Pro
Introduction to Advanced CAT Tool Learn how to use STK's Conjunction Analysis capability to avoid a satellite collision.	STK Pro, <i>CAT</i>	STK Premium (Space) or STK Enterprise	STK Pro
Design Trajectories with Astrogator Learn how to use STK's Astrogator capability to place a satellite in orbit.	STK Pro, <i>Astrogator</i>	STK Premium (Space) or STK Enterprise	STK Pro
Ground-based SSA with EOIR Learn how to set up an EOIR sensor in STK to track an object in space and determine the attitude profile of the target based on its light signature.	STK Pro, <i>EOIR</i> , <i>SatPro</i>	STK Premium (Space) or STK Enterprise	STK Pro + STK Premium



Note: For the STK 10 version of this training, visit <http://help.agi.com/StartTraining/StartTraining1013.htm>

Part 6: Introduction to the Pro Capabilities



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.

Credits: ISERV, NASA

Problem statement

Engineers and operators require a quick way to determine if local terrain is affecting visibility between ground sites and satellites for a variety of purposes such as communications, imaging, and general situational awareness. In this tutorial, there is a GPS monitoring station in the vicinity of Mount St. Helens. A USGS Digital Elevation Model (DEM) file contains terrain data for the area requiring analysis. Raw satellite imagery is available of Mount St. Helens which will provide situational awareness of the analytical area when converted as an inlay. A further need exists to create color elevation imagery of the area to be used in a briefing and in documentation.

Solution

Using STK, load a DEM file to analyze the impact of local terrain on accesses between a ground station and GPS satellites. Create a color elevation image, and convert a raw satellite image that can be used as an inlay for visualization and situational awareness. Change the DEM file into a STK Terrain File (pdtt) which can be used to visualize terrain in the 3D Graphics window.

Creating a new scenario

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

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_Pro
Start:	Default
Stop:	Default

- When you finish, click **OK**.
- When the scenario loads, click Save () . A folder with the same name as your scenario is created for you.
- Verify the scenario name and location in the Save As window.
- Click **Save**.

U.S. Geological Survey Data

- Create a folder on your Desktop called Terrain.
- Using File Explorer, browse to the location of the DEM file using the following:
 - STK 11: C:\Program Files\AGI\STK 11\CodeSamples\CodeSamples.zip\SharedResources\Scenarios\Events
 - STK 12: C:\Program Files\AGI\STK 12\CodeSamples\CodeSamples.zip\SharedResources\Scenarios\Events
- Locate and copy the file named hoquiam-e.dem.
- Paste the hoquiam-e.dem file into the Terrain folder you created on your desktop.

Analytical Terrain

The *Pro* capability adds realism to system models. Building on the fundamental capabilities of STK, *Pro* introduces more sophisticated modeling through advanced access constraints, flexible sensor shapes, complex visibility links, more object tracks and digital terrain data.

Load the locally available DEM terrain data file to be used for analysis and visualization into the scenario.

- Right click on STK_Pro () in the Object Browser.
- Select Properties () .

3. Select the Basic - Terrain page when the Properties Browser opens.
4. Clear Use terrain server for analysis.

You're using a local terrain file for both analysis and visualization. By turning off streaming terrain, you are simulating what you'd see in a setting that doesn't have an Internet connection.

5. Click **Add** in the Custom Analysis Terrain Sources: frame.
 6. Open the file type shortcut menu.
 7. Select USGS DEM (DEM).
 8. Browse to the Terrain folder on your desktop that contains the hoquiam-e.dem file.
 9. Select hoquiam-e.dem.
 10. Click **Open** .
 11. Click **OK** to accept your changes and to close the Properties Browser.
-

2D Graphics imagery

Display a background image and inlay in the 2D Graphics window.

1. Bring the 2D Graphics window to the front.
 2. Click Properties () in the 2D Window Defaults tool bar.
 3. Select the Imagery page when the Properties Browser opens.
 4. Select the Image File: option.
 5. Ensure Basic.bmp is selected (default).
-

2D Graphics details

The following will outline international borders and all the states of the United States and provinces of Canada.

1. Select the Details page.
2. Click on and highlight the following (leave RWDB2_Coastlines highlighted) in the Map Details field:
 - RWDB2_International_Borders
 - RWDB2_Provincial_Borders

Create a yellow outline of the hoquiam-e.dem file's location.

3. Select the Show Extents check box in the Terrain Regions Display frame.
4. Click **OK** to accept your changes and to close the Properties Browser.

Viewing your changes in the 2D Graphics window

1. Bring the 2D Graphics window to the front.
2. Zoom in () to the state of Washington.

The yellow box in the southwest corner of the state is the location of the hoquiam-e.dem file. Analysis inside the box can be done using local terrain. Analysis outside the box will take place on the WGS84.

3. Zoom in () to the terrain extents and move it to the right of your map.

Creating Color Elevation Imagery utility

The Create Color Elevation Imagery utility allows you to create images from terrain data that show elevation data colored by height (for example, blue for sea level, green for land, and white for mountain tops). This is useful if you need to visualize terrain but lack imagery or if you want to put the emphasis on height rather than the content of the image.

1. Open the Utilities menu.
2. Select Create Color Elevation Imagery.
3. Move the Create Color Elevation Imagery utility next to the terrain region display in the 2D Graphics window.
4. Enter the following extents which match the hoquiam-e.dem extents:

Option	Value
North Lat	47 deg
West Lon	-123 deg
East Lon	-122 deg
South Lat	46 deg

5. Click **Get Min/Max Alt** .

6. Set the following Min Altitude: values in the Color Options frame:

Option	Value
Hue	0.25
Saturation	0.75
Value	0.20

7. Select the JPEG 2000 Image (jp2) Format: in the Output File frame.
 8. Click the Directory: ellipses () .
 9. Browse to the location of your scenario (e.g. C:\Users\username\Documents\STK 12\STK_Pro).
 10. Click **Select Folder** .
 11. Type StHelensColor in the Filename: field.
 12. Click **Convert** .
 13. Close the Create Color Elevation Imagery utility.
-

2D Graphics imagery

2D Graphics Imagery is used to display a background image and inlay images in 2D Graphics windows.

1. Bring the 2D Graphics window to the front.
 2. Open the 2D Graphics window properties () .
 3. Select the Imagery page.
 4. Click Add... in the Inlay Images frame.
 5. Select StHelensColor.jp2 in the Map Images dialog box.
 6. Click **Open** .
 7. Click **OK** to accept your changes and to close the Properties Browser.
-

Viewing the inlay in the 2D Graphics window

1. Bring the 2D Graphics window to the front.

Imagery converter

Convert an ISS SERVIR Environmental Research and Visualization System (ISERV) LEVEL-R (RAW) imagery .jpg into JPEG 2000 Image (jp2) inlay.

1. Open the Utilities menu.
 2. Select Imagery and Terrain Converter.
 3. Select the Single Image page when the Properties Browser opens.
-

Input data

Select the image file to be converted.

1. Click the Image Filename: ellipsis () in the Input Data frame.
2. Browse to the location of the ISERV imagery (Typically, <STK install folder>\Data\Resources\stktraining\imagery\) in the File dialog box.
3. Select ISERV_Imagery.
4. Click **Open** .
5. Select the file named IPR201407092040194622N12217W.JPG.
6. Click **Open** .

The extents are automatically read from the metadata contained in the supporting files and entered into the Image: fields located in the Image Extent frame.

Output data

You can select a JPEG 2000 image.

1. Set the following in the Output Data / Image file frame:

Option	Value
Format:	JPEG 2000 Image (jp2)
Directory:	Location of the scenario (e.g. C:\Users\username\Documents\STK 12\STK_Pro
Filename:	ISERV_StHelens

2. Click **Convert**.
-

Terrain converter

Use the Terrain Region converter to create terrain inlays (pdtt files) for a specific region. Select your input data.

1. Select the Terrain Region page.
 2. Open the Terrain Source: shortcut menu in the Input Data frame.
 3. Select the path to the hoquiam-e.dem file.
-

Output data

1. Set the following in the Output Data frame:

Option	Value
Directory:	Location of the scenario (e.g. C:\Users\username\Documents\STK 12\STK_Pro
Filename:	StHelensTerrain

2. Click **Convert**.
3. Click **Close** to close the Imagery and Terrain Converter when finished.

Globe Manager

Globe Manager allows you to customize scenario globes with imagery and terrain data as well as manage that data once it has been applied. The Hierarchy window is used to add central bodies, image and terrain items to a scenario.

1. Bring the 3D Graphics window to the front.
 2. Click Globe Manager () in the Globe Manager toolbar.
 3. Click Add Terrain/Imagery () In the Hierarchy toolbar.
 4. Select Add Terrain/Imagery... () in the shortcut menu.
-

Globe Manager: opening Terrain and Imagery Data

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: shortcut menu 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\STK_Pro).
3. Select both the Image (ISERV_StHelens.jp2) and Terrain files (StHelensTerrain.pdtt).
4. Click **Add**.
5. Select **No** in the Use Terrain for Analysis dialog box.

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

Viewing the Imagery and Terrain files in the 3D Graphics window

1. Right-click on ISERV_StHelens.jp2 in Globe Manager.
2. Select Zoom To.
3. Use your mouse to view the image and surrounding terrain.

You can see the satellite image ISERV_StHelens.jp2 which is placed on top of the StHelensTerrain.pdtt. Terrain outside of the image is still visible. If desired, you can return to Globe Manager and turn Aerial.ve back on to use Bing Maps. You can also, toggle ISERV_StHelens.jp2 or StHelensTerrain.pdtt on and off to see the visual difference. The ISERV_StHelens.jp2 image can also be used as an inlay in the 2D Graphics window.

4. When finished, ensure the ISERV_StHelens.jp2 and StHelensTerrain.pdtt are turned on.

Inserting the GPS Tracking Device

Insert a Place () object which will simulate the GPS satellite tracking device.

1. Select Place () in the Insert STK Objects tool.
2. Select the Insert Default () method.
3. Click **Insert...**
4. Right click on Place1 () in the Object Browser.
5. Select Rename in the shortcut menu.
6. Rename Place1 () to TrackingDevice.
7. Open TrackingDevice's () properties ()
8. Select the Basic - Position page when the Properties Browser opens.
9. Set the following:

Option	Value
Latitude:	46.204 deg
Longitude:	-122.188 deg

10. Set Height Above Ground: to 5 ft. This represents the height of the tracking device's stand.
11. Click **Apply** to accept your changes and to keep the Properties Browser open.

Defining an Azimuth-Elevation Mask

The AzElMask properties enable you to define an azimuth-elevation mask for the GPS tracking device.

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

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

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

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

Displaying the Azimuth-Elevation Mask

For situational awareness, you can display the Azimuth-Elevation Mask in both the 2D Graphics and 3D Graphics windows.

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

Option	Value
Show	Select
Number of Steps:	10
Minimum Range:	0 km
Maximum Range:	50 km

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

Viewing the Azimuth Elevation Mask in the 3D Graphics window

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 Enable in the Label Declutter frame.
5. Click **OK** to accept your change and to close the Properties Browser.

Label Declutter is used to separate the labels on objects that are in close proximity for better identification in small areas. It also keeps object labels from being hidden by terrain.

6. Right-click on TrackingDevice () in the Object Browser.
7. Select Zoom To in the shortcut menu.
8. Using your mouse, zoom out until you can see the visual representation of the Azimuth-Elevation Mask.

Each ring represents a 10 kilometer range out to 50 kilometers. Around the edge of the view, you can see indications of North (N), South (S), East (E), and West (W). The view indicates that the view to the east, west, and south are poor. Visibility to the north is good.

Constellation objects

Load the GPS Constellation using orbital elements from GPS almanac files. The Almanac files can be stored in local directories or pulled from AGI Servers (internet connection required). STK creates a constellation that includes all of the satellites in the almanac. Keep in mind, you can build Constellation () objects yourself by loading an empty Constellation () object and assigning objects required for your analysis.

1. Insert a Satellite () object using the Load GPS Constellation () method.

Once loaded, you will see each individual Satellite () object and a Constellation () object containing all of the satellites.

2. Open GPSConstellation's () properties () .
3. Select the Basic - Definition page when the Properties Browser opens.

You can see each GPS satellite has been moved to the Assigned Objects list. If required, you could remove any satellites from the list that are non-operational or are not required for analysis.

4. Select the Constraints - Basic page.

For the purposes of this problem, you require access to at least four GPS satellites at any given time. It doesn't matter which four. Also, take into account that when performing an access, the access will be analyzed from the satellite to the GPS tracing device.

5. Open the Set 'From' access position: shortcut menu in the Logical Restriction frame.

6. Select At Least N.

7. Change the value to 4.

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

Chain objects

A Chain object is a list of objects (either individual or grouped into constellations) in order of access.

1. Insert a Chain () object using the Insert Default () method.
2. Rename Chain1 () to GPStoDevice.

Defining the start and end objects

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

1. Open GPStoDevice's () properties ()
2. Select the Basic - Definition page when the Properties Browser opens.
3. Click the Start Object: ellipses ()
4. Select GPSConstellation () in the Select Object dialog box.
5. Click **OK** to close the Select Object dialog box.
6. Click the End Object: ellipses ()
7. Select TrackingDevice () in the Select Object dialog box.
8. Click **OK** to close the Select Object dialog box.

Choosing the Chain object's connections

After you choose the start and end objects in your chain, you need to build the chain's connections.

1. Click **Add** in the Connections frame.
2. Click the From Object: ellipses ()
3. Select GPSConstellation () in the Select Object dialog box.
4. Click **OK** to close the Select Object dialog box.
5. Click the To Object: ellipses ()
6. Select TrackingDevice () in the Select Object dialog box.
7. Click **OK** to close the Select Object dialog box.
8. Click **OK** to accept your changes and to close the Properties Browser.

Viewing the accesses in the 3D Graphics window

1. Bring the 3D Graphics window to the front.
2. Zoom To TrackingDevice ().
3. Using your mouse, zoom out until you can see accesses from the ground site to the GPS satellites.
4. Click Start () in the Animation toolbar to animate the scenario.

Notice how the accesses don't appear until they're above the Az El Mask. Conversely, they disappear when they reach the Az El Mask. Terrain is being taken into account for the accesses.

5. Click Reset () in the Animation toolbar to reset the scenario when you are finished.

Completing Chain Access Report

The Complete Chain Access report shows times during which access among all objects in a chain is possible through one or more strands.

1. Right-click on GPSToDevice () in the Object Browser.
2. Select Report & Graph Manager... () in the shortcut menu.
3. At the top of the Styles list clear Show Reports when the Report & Graph Manager opens.
4. Select the Complete Chain Access () graph in the Installed Styles list.
5. Click **Generate...** .

The Complete Chain Access is telling you exactly what you need to know. If there is a solid line across the report, that means you always have access to at least four or more GPS satellites during the entire analytical period. If there is a break in the line, that's a period when three or less GPS satellites are being accessed.

6. Close the Graph.

Individual Strand Access

A Individual Strand Access report represents one possible access pathway through the chain. For a chain that consists of a series of individual objects, only a single strand is possible. If a chain contains one or more constellations, multiple strands are possible, but constellation constraints (ANY OF, ALL OF, AT LEAST N, EXACTLY N) can affect the possible number of strands.

1. Return to the Report & Graph Manager.
2. Select the Individual Strand Access () graph.
3. Click **Generate...** .

This graph shows you when the ground site can see individual GPS satellites when that satellite is paired up with at least three more satellites.

4. Keep the graph open.

Turning off the Azimuth Elevation Mask constraint

You can verify that analytical terrain is being used in your analysis.

1. Open TrackingDevice's () properties () .
2. Select the Constraints - Basic page.
3. Clear Use in the Az - El Mask frame.

4. Click **Apply** to accept your changes and to keep the Properties Browser open.
5. Return to the Individual Strand Access graph.
6. Click Refresh (F5) () in the graph toolbar.

Notice how the individual strands grow larger? The surrounding terrain is no longer being used in the analysis.

7. Return to TrackingDevice's () properties ()
8. Select Use in the Az - El Mask frame.
9. Click **OK** to accept your changes and to close the Properties Browser.
10. Return to the Individual Strand Access graph.
11. Click Refresh (F5) () in the graph toolbar.

The individual strands shrink because analytical terrain is being used in the analysis.

Saving your work

1. Reset () the scenario and close any reports or tools that are still open when you are finished.
2. Save () your work.

Summary

You began by loading a USGS DEM terrain file into your scenario which you used as analytical terrain. You used the Create Color Elevation Imagery tool to create an image from the terrain data that showed elevation data colored by height in the 2D Graphics window. You used the Imagery and Terrain Converter utility to create an ISERV image which you displayed in the 3D Graphics window and the terrain converter to create a pdtt terrain file from the USGS DEM file. You used this file to visualize the terrain features in the 3D Graphics window. You inserted a Place object into the scenario that simulated a GPS tracking device. You set up the tracking device to use the terrain analytically. You propagated GPS Satellites from GPS almanac and created a Constellation object which contained the GPS Satellites. Finally, you inserted a Chain object into the scenario and created accesses from the GPS Constellation to the tracking device. You required at least four or more satellites in your analysis which you graphed using a Complete Chain Access graph and an Individual Strand Access graph.

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.

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.

Create 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 2021 16:00:00.000 UTCTG
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**.
-

Disable 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.
-

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.

Recently Launched Satellite

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

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:	New_Sat
Inclination:	50 deg
Altitude:	800 km
RAAN:	-85 deg

5. Click **OK**.
-

Calculate 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 () in the Access Tool's Associated Objects list.
4. Click Compute ( **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.
-

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.

Open 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.

Display 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.

Show 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 () to animate the scenario in the Animation toolbar. The Sun Vector follows the Sun.
6. Click the Reset () when you are finished.

Show 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. Click the Expand () button beside To Vectors () in the Components for: list on the right of the Add Components window.
5. Select New_Sat.
6. Click **OK** to close the Add Components window.

Show 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 Tracking_Station's () properties () .
2. Ensure New_Sat Vector is selected in the Vectors list on the 3D Graphics - Vector page.
3. Select Show Magnitude.
4. Click **Apply** .

View 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 () in the Animation toolbar 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.

Create 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.

Open 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.
4. Ensure Tracking_Station () is selected in the Object list.
5. Click Create new Angle () to open the Add Geometry Component window.

Create a Between Vectors Angle

Select the Between Vectors angle type. You will create an angle between the Sun Vector and the New_Sat Vector.

1. Ensure the Type: is set to Between Vectors. This is the default Type.
2. Enter TrackingAngle in the Name: field.

Select 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 ellipsis button () beside the From Vector: field. The Select Reference Vector window opens.
2. Select Tracking_Station () in the Objects list.
3. Scroll up in the Vectors for: Tracking_Station list on the right.
4. Click the Expand () button beside To Vectors () .
5. Select New_Sat.
6. Click **OK** to close the Select Reference Vector window.

Select 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. Return to the Select Reference Vector window.
2. Note that To Vector: already has the Tracking_Station Sun vector selected.
3. Click **OK** to close the Add Geometry Component window.
4. Note the TrackingAngle vector is in My Components () .
5. Keep the *Analysis Workbench* () open.

Display 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 TrackingAngle in the My Components (📁) folder in the the Components for: list.
 6. Click **OK** to close the Add Components window.
 7. Note Show is selected for TrackingAngle.
 8. Click **OK** to accept the changes and close the Properties Browser.
-

View Tracking Angle

View the tracking angle in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
 2. Click Start (▶) in the Animation toolbar 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.
-

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 (📊).

1. Return to the *Analysis Workbench* (Σ).
2. Select the Calculation tab to open the Calculation Tool.
3. Select Tracking_Station (📍) in the Object list.
4. Click Create new Scalar Calculation (📊) to open the Add Calculation Component window.

Define 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. Ensure the Type: is set to Angle. This is the default Type.
 2. Enter Scalar_TrackingAngle in the Name: field.
 3. Click the ellipsis (⋮) beside the Input Angle: field. The Select Reference Angle window opens.
 4. Select Tracking_Station (🚪) in the Objects list.
 5. Select TrackingAngle in the My Components (📁) folder in the Angles for: Tracking_Station list.
 6. Click **OK** to close the Select Reference Vector window.
 7. Click **OK** to close the Add Calculation Component window.
 8. Note Scalar_TrackingAngle is in My Components (📁).
-

Condition

Create a new Condition (📊).

1. Return to the *Analysis Workbench* (Σ) Calculation tab.
2. Select Tracking_Station (🚪) in the Object list.
3. Click Create new Condition (📊) to open the Add Calculation Component window.

Define 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. This is the default Scalar Bounds.
2. Enter Below_10_Degrees in the Name: field.

Define the Scalar

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

1. Click the ellipsis (⋮) beside the Scalar: field. The Select Reference Scalar Calculation window opens.
2. Select Tracking_Station (🚪) in the Objects list.

3. Select `Scalar_TrackingAngle` in the My Components (📁) folder in the `Scalar_TrackingAngle` for: Tracking_Station list.
4. Click **OK** to close the Select Reference Scalar Calculation window.

Define the Calculation Conditions

Define the calculation conditions.

1. Select `Below Maximum` in the Operation drop down list.
2. Enter `10 deg` in the Maximum: field.
3. Click **OK** to close the Add Calculation Component window.

Create 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 *Analysis Workbench* (🔍) Calculation tab.
2. Select `Tracking_Station` (📶) in the Objects list.
3. Expand (⊕) `Below_10_Degrees` in the My Components (📁) Components for: 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 the *Analysis Workbench* (🔍) open.

View 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 *Analysis Workbench* () .
 2. Select SatisfactionIntervals under Below_10_Degrees in the My Components () Components for: list.
 3. Drag and drop SatisfactionIntervals into the Timeline View's Time Display.
 4. Observe the Timeline View.
-

Time Tool

Using the Time Tool, you will create a custom 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 the *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 window.
5. Enter Optimal_Tracking_Times in the Name: field.

Define 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. Select Merged () in the Select Component Type window list.
3. Click **OK** to close the Select Component Type window.

Define 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 window.
2. Select MINUS in the Operation drop down list.

Remove 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.

Define 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...** to open the Select Time Intervals window.
2. Select Facility-Tracking_Station-To-Satellite-New_Sat (🔑) in the Object list.
3. Select AccessIntervals in the Installed Components (📁) Components for: list.
4. Click **OK** to close the Select Time Intervals window.
5. Click **Add...** on the Add Time Component window.
6. Select Tracking_Station (📡) in the Object list.
7. Expand (⊕) Below_10_Degrees in the My Components (📁) Components for: list.
8. Select SatisfactionIntervals.
9. Click **OK** to close the Select Time Intervals window.
10. Click **OK** to close the Add Time Component window.

View Intervals in the Timeline View

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

1. Return to the *Analysis Workbench* () Time tab.
 2. Select *Optimal_Tracking_Times* in the My Components () Components for: list.
 3. Drag and drop *Optimal_Tracking_Times* into the Timeline View's Time Display.
 4. Observe the Timeline View to see the merged intervals.
 5. Return to the *Analysis Workbench* () .
 6. Click **Close** .
-

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.

Insert a Sensor on Tracking Station

Insert a sensor object on the tracking station.

1. Select Sensor () in the Insert STK Objects tool.
2. Select the Define Properties () method.
3. Click **Insert...** .
4. Select *Tracking_Station* () on the Select Object window.
5. Click **OK** .

Model Limited Field-of-View

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

1. Select the Basic - Definition page.
2. Ensure the Sensor Type: is set to Simple Conic. This is the default type.
3. Enter 2 deg in the Cone Half Angle: field.
4. Click **Apply** .

Target the Sensor

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

1. Select the Basic - Pointing page.
 2. Select Targeted in the Pointing Type: drop down list.
 3. Select New_Sat () in the Available Targets list.
 4. Click Move () to move New_Sat () to the Assigned Objects list.
 5. Click **OK**.
 6. Rename Sensor1 () Tracking_Sensor.
-

Calculate 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 Access Tool's Associated Objects list.
 4. Click Compute ( Compute).
 5. Click **Access...** in the Reports section.
 6. Note the multiple accesses between Tracking_Station () and New_Sat () .
 7. Keep the Access tool () and the report open.
-

Add 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 Targeted pointing type to your sensor object using the custom components created earlier in this lesson.

1. Open Tracking_Sensor's () properties ()
 2. Select the Constraints - Temporal page.
 3. Select Use in the Intervals section.
 4. Click **Import...** to open the Select Time Intervals window.
 5. Select Tracking_Station () in the Object list.
 6. Select Optimal_Tracking_Times in the My Components () Components for: list.
 7. Click **OK** to close the Select Time Intervals window.
 8. Note the intervals have populated in the Intervals list.
 9. Click **OK** to close the Properties Browser.
-

Create 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.
-

Solar Exclusion Angle

What you have built and used in this lesson is a Solar Exclusion Angle. Set that directly in the sensor constraints.

1. Open TrackingSensor's () properties ()
 2. Select the Constraints - Temporal page.
 3. Disable the Use option in the Intervals list.
 4. Select the Constraints - Sun page.
-

5. Enable the Boresight - Solar Exclusion Angle.
6. Set the value to ten (10) deg.
7. Click OK to close the Properties Browser.

The access times update and just like before, you can see the times that 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.

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:

1. The percentage of the Earth's surface the satellite payloads survey during a 24-hour period and daylight hours only
2. How many times the satellite payloads survey points on the ground during a 24-hour period and daylight hours only
3. 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	1 Jul 2020 16:00:00.000 UTCG
Stop	2 Jul 2020 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.
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.
 4. Clear the **Use terrain server for analysis** check box.
 5. Click **OK** to accept the changes and 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 window:

Option	Value
Type	Circular
Satellite Name	Circ_Sat
Inclination	55 deg
Altitude	700 km

5. Click **OK** .
-

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. 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 window:

Option	Value
Type	Repeating Ground Trace
Satellite Name	Repeat_Sat

5. 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. 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 window:

Option	Value
Type	Sun Synchronous
Satellite Name	Sun_Sat

5. 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

Insert a Sensor () on Circ_Sat () . Define a rectangular sensor with a 20-degree vertical half-angle and 10-degree horizontal half-angle.

1. Select Sensor () in the Insert STK Objects tool.
2. Select the Define Properties () method.
3. Click **Insert...** .
4. Select Circ_Sat () in the Select Object window.
5. Click **OK** .
6. Go to the Basic - Definition page.
7. Set the following:

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

8. Click **OK** .

Reusing the Sensor object

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

1. Select the Sensor () attached to Circ_Sat () 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.
7. Expand () Repeat_Sat () and Sun_Sat () .

Renaming the Sensor Objects

Rename the three sensors.

1. Right-click the Sensor () object attached to Circ_Sat () in the Object Browser.
 2. Select Rename.
 3. Enter Circ_Sens.
 4. Press Enter on your keyboard.
 5. Repeat the steps above to rename Repeat_Sat's sensor () Repeat_Sens and Sun_Sat's sensor () Sun_Sen.
-

Adding a Coverage Definition object

The Coverage Definition object defines a coverage area for analysis.

Inserting a Coverage Definition object

Insert a Coverage Definition () into your scenario.

1. Select Coverage Definition () in the Insert STK Objects tool.
2. Select the Insert Default () method.
3. Click **Insert...**
4. Right-click CoverageDefinition1 () in the Object Browser.
5. Select Rename.
6. Enter World_Cov.
7. Press Enter on your keyboard.

Defining the coverage grid

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

1. Right-click World_Cov' () in the Object Browser.
2. Select Properties () .
3. Go to the Basic - Grid page.
4. Set the following parameters:

Option	Value
Type	Global
Point Granularity - Lat/Lon	4 deg

5. Click **Apply** .
6. Select the 2D Graphics window to bring it to the front. The grid now covers the entire Earth and the grid spacing is 4 degrees instead of 6 degrees.
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. Use the Ctrl key to select all three Sensor () objects in the Assets field.
4. Click **Assign** .
5. Click **Apply** .

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. Go to the Basic - Advanced page.
 2. Clear the **Automatically Recompute Accesses** check box.
 3. Click **OK** .
-

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. There is a progress bar in the lower-right corner of STK.
4. Right-click World_Cov () in the Object Browser when computing accesses is complete.
5. Select Report & Graph Manager () .
6. Select Coverage By Latitude graph () in the Installed Styles list.
7. Click **Generate...** .

The Coverage By Latitude graph is a quick way to see the percentage of time covered by latitude.

8. Close () the graph.
 9. Click **Close** on the Report & Graph Manager window.
-

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. Select Figure Of Merit () in the Insert STK Objects tool.
2. Select the Insert Default () method.
3. Click **Insert...** .
4. Select World_Cov () in the Select Object window.
5. Click **OK** .
6. Rename the Figure Of Merit () 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. Right-click Simple_Cov's () in the Object Browser.
2. Select Properties ().
3. Note the default Definition - Type is set as Simple Coverage on the Basic - Definition page.
4. Click **Cancel** .

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 ().
3. Select Percent Satisfied report () in the Installed Styles list.
4. Click **Generate...** .
5. Note the % Satisfied value at the bottom of the report (e.g., 78%).

78% of the globe is covered by at least one of the three sensors during the analysis time period.

6. Close () the report when finished.
7. Close () the Report & Graph Manager.

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. This will be the grid constraint object.

1. Select Target () in the Insert STK Objects tool.
2. Select the Define Properties () method.
3. Click **Insert...**
4. Go to the Constraints - Sun page.
5. Select Lighting.
6. Ensure Direct Sun is set.
7. Click **OK**.
8. Rename the Target () Constraint_Template.
9. 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 Target () object as the grid constraint. It contains the constraint you want to apply to the coverage grid.

1. Right-click World_Cov' () in the Object Browser.
2. Select Properties ()

3. Click **Grid Constraint Options...** on the Basic - Grid page.
 4. Ensure Reference Constraint Class is set to Target on the Grid Constraint Options window. 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.
 7. Click **OK** to close the Grid Constraints Options window.
 8. Click **OK** to accept the changes and close World_Cov's () properties (.
-

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. There is a progress bar in the lower-right corner of STK.
 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 (.
3. Select Percent Satisfied report () in the Installed Styles list.
4. Click **Generate...**
5. Note the % Satisfied value at the bottom of the report (e.g., 45 %).

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 () the report when finished.
 7. Close () the Report & Graph Manager.
-

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. Select Figure Of Merit () in the Insert STK Objects tool.
 3. Select the Define Properties () method.
 4. Click **Insert...**
 5. Select World_Cov () in the Select Object window.
 6. Click **OK**.
 7. Select Number Of Accesses as the Definition - Type on the Basic - Definition page.
 8. Keep Compute set to the default Total.
 9. Click **OK**.
 10. Rename the Figure Of Merit () 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 ()
3. Select Grid Stats report () in the Installed Styles list.
4. Click **Generate...**
5. Note the Maximum value (e.g., 8).

That means at least one point in the grid was accessed on eight 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 when finished.
 8. Close () the Report & Graph Manager.
 9. 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. Select Area Target () in the Insert STK Objects tool.
 2. Select the Select Countries and US States () method.
 3. Click **Insert...**
 4. Clear US States on the Select Countries And US States window.
 5. 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.
 6. Click **Insert**.
 7. Click **Close**.
-

Defining a new Coverage Definition

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

Inserting a Coverage Definition

Insert a new Coverage Definition () .

1. Select Coverage Definition () in the Insert STK Objects tool.
2. Select the Define Properties () method.
3. Click **Insert...** .
4. Rename the new Coverage Definition () 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 the Coverage Definition () Properties () .
2. Set Type as Custom Regions in the Grid Area of Interest section on the Basic - Grid page.
3. Select Area Targets in the drop-down menu below Custom Regions.
4. Use the Ctrl key to select Canda and United_States_of_America in the Area Targets list on the left.
5. Click Move () to move the Area Targets to the Selected Regions list on the right.
6. Set the Point Granularity - Lat/Lon to 1 deg.
7. Click **Apply** .
8. Bring the 2D Graphics window to the front.
9. 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.

10. 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. Select Repeat_Sens () in the Assets list.
4. Click **Assign** .
5. Click **Apply** .

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 Show - Grid field. This turns off the visual grid inside the Area Target () objects. Analytically, they're still there.
3. Click **OK** .

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.
3. Select Compute Accesses. There is a progress bar in the lower-right corner of STK.

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_Time (.

1. Select Figure Of Merit () in the Insert STK Objects tool.
2. Select the Define Properties () method.
3. Click **Insert...**
4. Select Country_Cov () in the Select Object window.
5. Click **OK**.
6. Rename the Figure Of Merit () 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. Set the Definition - Type to Coverage Time on the Basic - Definition page.
2. Change Compute to Total Time Above.

Total Time Above computes the amount of time over the entire coverage interval during which a point is covered by at least the specified number of assets based on the entire coverage interval.

3. Click **Apply**.

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 (.
3. Select Grid Stats report () in the Installed Styles list.
4. Click **Generate...**

5. Note the Maximum time (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. Go to the 2D Graphics - Static page.
2. Set Filled Area - % Translucency to 30 in the Show Points As field.
3. Select Show Contours in the Display Metric field.
4. Click **Remove All** in the Level Attributes field.
5. Set the following in the Level Adding field:

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. Change the Color Ramp Start Color to red in the Level Attributes field.
8. Change the End Color to blue in the Level Attributes field.

Points with no coverage will be red and any points at or above your highest Level Attribute value will be blue.

9. Select Natural Neighbor in the Contour Interpolation (points must be filled) field. Color is applied smoothly over all points in the grid to differentiate contour levels.
10. Click **Apply**

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. Return to Cov_Time's (A*) properties (📄) 2D Graphics - Static page.
2. Click **Legend...** .
3. Click **OK** to close Cov_Time's (A*) properties (📄).
4. Click **Layout...** in the Static Legend for Cov_Time window.
5. Set the following on the Figure of Merit Legend Layout window:

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

6. Click **OK** .
7. Close (✕) the Static Legend for Cov_Time window.
8. Bring the 2D and 3D Graphics windows to the front.
9. 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 (A*) 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 (A*) 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 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 - Basic page.
 2. Select the Use check box in the Az-El Mask frame.
 3. 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** .
-

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 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 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 AzEI Mask tool.

Opening the AzEI Mask Tool

Use the AzEI 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 AzEI Mask... in the shortcut menu.

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

Preparing the AzEI Mask Tool

Start by setting up the AzEI Mask Tool prior to creating a .bmsk file. Set Comm_Radome () as the obscuring object and the window dimension to 500.

1. Move the AzEI Mask dialog box (AzEIMask for Radar_FOV) to the right so that it isn't on top of the Az/EI Mask View window.
2. Select Comm_Radome () in the AzEI 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 AzEI Mask window and the Az/EI Mask View window when the computation is complete.
-

Constraining the sensor with the AzEI 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 AzEI 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 AzEI Mask constraint

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

1. Select the 2D Graphics - Projection page.
2. Leave AzEIMask selected in the Field of View - Use Constraints frame.
3. Scroll down the Field of View - Use Constraints list until you locate SensorAzEIMask.
4. Press **Ctrl** on your keyboard and select SensorAzEIMask.
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 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 Ansys 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 Volumetrics 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** .
-

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.

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 () 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.

Problem Statement

Engineers and operators need to quickly determine communication link budgets. Factors such as communications unobstructed or obstructed by terrain, trees or man made structures need to be taken into consideration. Other factors that need to be included and analyzed are rain models, atmospheric losses, and RF interference sources. In this scenario, a team of scientists is monitoring glacial meltwater in a remote, mountainous location. Prior to setting up camp, they need to determine how their location will impact a link budget between them and a low earth orbit (LEO), Earth observation satellite which is downloading data to the team.

Solution

Use STK Pro and STK's *Communications* capability to model and analyze a link budget between the ground site and the Earth observation satellite. The satellite transmitter will be analyzed using an isotropic, omnidirectional antenna pattern. The ground team will employ a small parabolic antenna steered by a servo motor that can track the satellite. After establishing a link budget based on line of sight only, terrain obstruction, rain, atmospheric absorption and system noise temperature will be factored in until the final link budget analysis is complete.

Create a New Scenario

Create a new scenario for a twenty four (24) hour time period starting 1 Jul 2020 06:00:00.000 UTCG.

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_Communications
Location:	Default
Start:	1 Jul 2020 06:00:00.000 UTCG
Stop:	2 Jul 2020 06:00:00.000 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.

Add 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. Right-click on STK_Communications () and select Properties (.
2. Select the Basic - Terrain page.
3. Disable Use terrain server for analysis.
4. Click OK to accept the changes and close the Properties Browser.
5. Bring the 3D Graphics window to the front.
6. In the 3D Graphic window toolbar, click Globe Manager (.
7. In the Globe Manager's Hierarchy toolbar, click Add Terrain/Imagery (.
8. Select the Add Terrain/Imagery () option in the drop-down.
9. In the Globe Manager: Open Terrain and Imagery Data window, click the ellipsis button () in the Path field.
10. Navigate to <STK Install Folder>\Data\Resources\stktraining\imagery (e.g. This PC then C:\Program Files\AGI\STK 12\Data\Resources\stktraining\imagery).
11. Click OK.
12. Select RaistingStation.pdtt.

13. Click Add .

14. When the Use Terrain for Analysis prompt displays, click Yes to enable terrain for analysis.

Declutter the 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. Open the 3D Graphics window properties () .
 3. On the Details page, locate the Label Declutter section.
 4. Select Enable.
 5. Click OK to accept the changes and close the Properties Browser.
-

Insert the Earth Observation Satellite

The Earth observation satellite is in a sun-synchronous orbit. Sun-synchronous 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. Using the Insert STK Objects tool, insert a Satellite () object using the From Standard Object Database method.
 2. In the Search Standard Object Data window, enter Rapideye in the Name or ID: field.
 3. Click Search.
 4. In the Results: field, select RapidEye 2 using the AGI's Standard Object Data Service Data Source.
 5. Click Insert.
 6. Close the Search Standard Object Data window.
-

Insert the Scientists' Camp Site

The camp site sits in a valley next to a river being fed by mountain glaciers.

1. Using the Insert STK Objects tool, insert a Place () object using the Define Properties method.
2. On the Basic - Position page, set the following:

Option	Value
Latitude:	47.5605 deg
Longitude:	11.5027 deg
Height Above Ground:	6 ft (simulates antenna height)

3. Click OK to accept the changes and close the Properties Browser.
 4. Rename the Place () object Camp_Site.
-

Simple Transmitter Model

The Simple Transmitter model is convenient when you do not have all the information necessary to model the transmitter in detail; for example, during the system engineering process.

1. Using the Insert STK Objects tool, insert a Transmitter () object using the Insert Default method.
 2. When the Select Object window appears, select Rapideye2_33312 ().
 3. Click OK.
 4. Rename the Transmitter () object Download_Tx.
-

Transmitter Model Specs

Using a Simple Transmitter Model type, the Model Specs tab allows you to set the transmitter's frequency, EIRP (Effective Isotropic Radiated Power), data rate, and polarization. The Simple Transmitter model defaults to an isotropic antenna pattern. An isotropic antenna pattern is an ideal spherical pattern antenna with constant gain.

1. Open Download_Tx's () properties.
2. On the Basic - Definition page, ensure the Model Specs tab is selected.
3. Set the following:

Option	Value
Frequency:	1.7045 GHz
EIRP:	10 dBW
Data Rate:	4.2 Mb/sec

4. Select Use in the Polarization field.
5. Set the polarization to Right-hand Circular.
6. Click OK to accept the changes and close the Properties Browser.

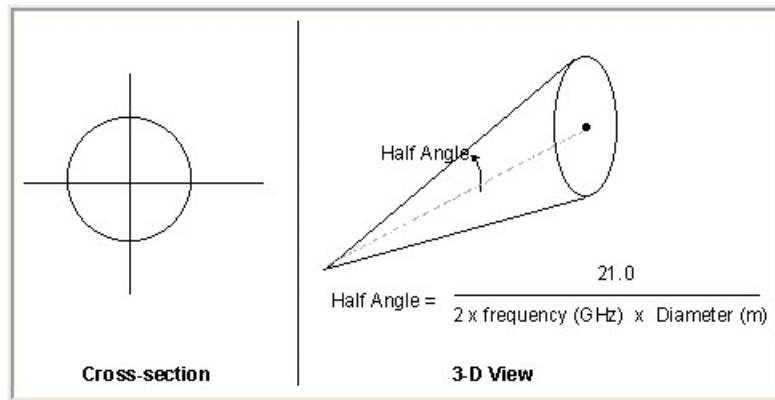
Receiver Antenna Orientation

The receiver antenna is steerable. To create a steering device (e.g. servo motor) in STK, you use a Sensor () object.

1. Zoom To Camp_Site () .
2. Using the Insert STK Objects tool, insert a Sensor () object using the Insert Default method.
3. When the Select Object window appears, select Camp_Site () .
4. Click OK.
5. Rename the Sensor () object Servo_Motor.

Half Power Sensor Patterns

Half Power sensor patterns are designed to visually model parabolic antennas. The sensor half angle is determined by frequency and antenna diameter.



HALF POWER SENSOR

1. Open Servo_Motor's properties (📄).
2. On the Basic - Definition page, set the following:

Option	Value
Sensor Type:	Half Power
Frequency:	1.7045 GHz
Diameter:	1.6 m

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

Targeted Sensor

The Targeted pointing type causes the sensor to point to other objects in the scenario.

1. Select the Basic - Pointing page.
2. Set the Pointing Type: to Targeted.
3. Set the Track Mode: value to Receive.
4. Move (➡) Rapideye2_33312 (🛰️) from the Available Targets list to the Assigned Targets list.
5. Click Apply to accept the changes and keep the Properties Browser open.

Target Times

Use Target Times to schedule or view access times from the sensor to the selected targets. This is a quick way to determine an access without using the Access Tool.

1. Click Target Times.
2. There are seven (7) opportunities to download data from the satellite to the ground site.
3. Click Cancel to close the Target Times window.
4. Click OK to close Servo_Motor's () properties.
5. Bring the 3D Graphics window to the front.
6. In the Timeline View, slowly move the gray pointer until the sensor accesses the satellite. Use your mouse to change the view so that you can view the access between Servo_Motor () and Rapideye2_33312 ()).

You can see that Servo_Motor () is targeting Rapideye2_33312 () . Rapideye2_33312 () has two (2) Sensor () objects attached to it that were downloaded with the Satellite () object if the satellite was inserted from AGI's Standard Object Data Service Data Source.

Complex Receiver Model

The Complex Receiver model allows you to select among a variety of analytical and realistic antenna models, and to define the characteristics of the selected antenna type.

1. Using the Insert STK Objects tool, insert a Receiver () object using the Insert Default method.
2. When the Select Object window appears, select Servo_Motor () .
3. Click OK.
4. Rename the Receiver () object Download_Rx.

Receiver Model Specs

1. Open Download_Rx's () properties () .
2. On the Basic - Definition page, click the ellipsis button (Component Selector) () beside the Type field.
3. Select Complex Receiver Model.
4. Click OK.

By default, the frequency is set to Auto Track. 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. LNA refers to Low Noise Amplifier. If you have those specifications, you can add them.

Receiver Antenna

You can select to embed an antenna model from the Component Browser or you can link to an antenna object. You will use a parabolic antenna.

1. Select the Antenna tab.
2. Click the ellipsis button (Component Selector) () beside the Type field.
3. Select Parabolic.
4. Click OK.
5. Set the following:

Option	Value
Design Frequency:	1.7 GHz
Diameter:	1.6 m

6. Click Apply to accept the changes and keep the Properties Browser open.
-

Receiver Antenna Polarization

The receiver polarization type is the same as the transmitter's polarization.

1. Select the Polarization sub-tab.
2. Select Use.
3. Change the polarization to Right-hand Circular.
4. Click OK to accept the changes and close the Properties Browser.

Simple Link Budget

Creating a Link Budget in the Access tool is referred to 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 window.

1. In the Object Browser, right click on Download_Rx () and select Access () .
2. When the Access Tool opens, in the Associated Objects list, expand () Rapideye2_33312 () .
3. Select Download_Tx () .
4. Click Compute ( Compute) .
5. In the Reports section, click Link Budget...

Take some time to look at the Simple Link Budget report. As the satellite rises over the horizon of the central body (WGS84) you receive transmissions. When the satellite falls below the horizon, you lose transmissions.

6. Leave the Link Budget Report open.

Take Terrain Into Consideration

Use Terrain Mask. If this check box is selected, access to the object is constrained by any terrain data in the line of sight to which access is being calculated.

1. Open Download_Rx's () properties () .
2. Select the Constraints - Basic page.
3. Select Terrain Mask.
4. Click OK to accept the changes and close the Properties Browser.
5. Return to the Link Budget Report and click Refresh (F5) () .

You'll see that all accesses blocked by analytical terrain (RaistingStation.pdtt) have been removed from the report. You began with seven (7) accesses but taking terrain into account you now have four (4) accesses.

6. Close the Link Budget Report.
7. Return to the Access Tool.

Detailed Link Budget

The Detailed Link Budget Report is a modified Link Budget Report with added Access Data Provider Elements.

1. In the Access Tool, click Report & Graph Manager...
2. When the Report & Graph Manager opens, go to the Styles - Installed Styles list and select Link Budget - Detailed.
3. Click Generate.
4. Change the report Step: to 30 sec.
5. Press Enter on your keyboard.

Take some time to look at the report. Looking at the first access, note the values for Atmos Loss (dB) and Rain Loss (dB). They're currently zero (0). You haven't loaded any RF Environment models. Also note the values in the C/N (dB) (carrier to noise ratio), Eb/No (dB) (energy per bit to noise ratio), and BER (Bit Error Rate) columns. For the purposes of your analysis due to downloading data, you require a BER equal to or lower than $1.000000e-13$. You can see from the report that you only have a few minutes during each pass to download data.

6. Leave the report open.
-

Rain Model

Environmental factors can affect the performance of a communications link. Apply rain and atmospheric absorption models to the analysis. Rain models are used to estimate the amount of degradation (or fading) of the signal when passing through rain.

1. Open STK_Communications () properties ()
2. Select the RF - Environment page.
3. Select the Rain & Cloud & Fog tab.
4. Select Use In the Rain Model section. Leave the default ITU (International Telecommunication Union) model.
5. Click Apply to accept the changes and keep the Properties Browser open.

Atmospheric Absorption Model

Atmospheric Absorption models estimate the attenuation of atmospheric gases on terrestrial and slant path communication signals.

1. Select the Atmospheric Absorption tab.
2. Select Use. Leave the default ITU model.
3. Click OK to accept the changes and close the Properties Browser.
4. Return to the Link Budget - Detailed report.
5. Observe the C/N (dB), Eb/No (dB) and BER columns.
6. Click Refresh (F5) () . There was a slight change in the C/N (dB) and Eb/No (dB) values.
7. Scroll to the Atmos Loss (dB) and Rain Loss (dB) columns.

Losses were minimal, therefore the slight losses in C/N (dB) and Eb/No (dB) values.

8. Leave the report open.

System Noise Temperature

The Receiver's System Noise Temperature allows you to specify the system's inherent noise characteristics.

1. Open Download_Rx's () properties () .
2. On the Basic - Definition page, select the System Noise Temperature tab.
3. Select Compute.
4. Select Compute in the Antenna Noise section.
5. Select Sun, Atmosphere, Rain and Cosmic Background.
6. Click OK to accept the changes and close the Properties Browser.
7. Return to the Link Budget - Detailed report.
8. Observe the C/N (dB), Eb/No (dB) and BER columns.

9. Click Refresh (F5) () . There were increases in the C/N (dB) and Eb/No (dB) values and decreases in BER.

10. Look at the following columns and observe their values:

- Tatmos (K)
- Train (K)
- Tsun (K)
- Tearth (K)
- Tcosmic (K)
- Tantenna (K)
- Tequivalent (K)

These are the calculated noise temperatures. Calculating system noise temperature improved your Link Budget Report and extended the time available for downloading data.

Summary

You inserted a Place () object into the scenario which simulated the location of a scientific team monitoring glacial meltwater in mountainous terrain. You used a local terrain file both analytically and visually for your analysis and situational awareness. You inserted and propagated an actual Earth Observation Satellite () object which the scientists need to communicate with for data download. You were introduced to the Simple Transmitter Model which was attached to the Satellite () object. The Simple Transmitter uses an isotropic antenna. Next, you created a servo motor using a Sensor () object which was used to steer the ground site receiver antenna by locking onto the Satellite () object. The Receiver () object used a Complex Receiver Model. This model allowed you to input a Parabolic Antenna Pattern. The first analysis, using the Access Tool, focused on a Simple Link Budget. The only constraint applied was line of sight which is basically horizon to horizon using the central body. Next, using Terrain Mask, you took into account local terrain and lost multiple accesses to the Satellite () object. You moved to the Report & Graph Manager and generated a Detailed Link Budget. You enabled an ITU rain model and atmospheric absorption model which had a slight detrimental impact on the Link Budget Report. Finally, you added System Noise Temperature analysis to the Link Budget Report which had a positive impact to the link budget.

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.

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:

1. Create an airfield radar site
2. Model an airport surveillance radar
3. Build a monostatic radar
4. Test various settings against multiple targets
5. Determine probability of detection

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 New Scenario Wizard:

Option	Value
Name:	STK_Radar
Location:	Default
Start:	1 Oct 2020 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**.
-

Turn Off Terrain Server

This is an introduction to *Radar*. Terrain will not be used in this analysis.

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

Turn On Label Declutter

Turn on Label Declutter to reposition object labels so they do not obstruct one another while in close proximity.

1. Open the 3D Graphics Window's properties (.
2. Select the Details page.

3. Select Enable for Label Declutter.
4. Click **OK**.

Insert 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. Rename the Aircraft () Target_Acft.

Create the Target Aircraft's 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.
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 Altitude: and Speed: in the Set All Grid Values window.
7. Set the following:

Option	Value
--------	-------

Altitude:	25000 ft
Speed:	330 mi/hr

8. Click **OK** to close the Set All Grid Values window.
9. Click **Apply** to accept the changes and keep the Properties Browser open.

Specify 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 transport turboprop aircraft.

1. Select the RF - Radar Cross Section page.
2. At the top of the page, clear Inherit. This allows you to set the RCS settings for the Aircraft () instead of inheriting the settings from the Scenario () object.
3. Set the Constant RCS Value: to 19 dBsm (decibels referenced to a square meter).

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**.

Display 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 Show Volume in the Volume Graphics section.
3. Click **OK**.
4. Bring the 3D Graphics window to the front.
5. Right-click on Target_Acft () in the Object Browser.
6. Select Zoom To.
7. Use your mouse to zoom out until you can see the RCS sphere.

Insert the Radar Site

Use a Place (📍) object as the radar site location.

1. Select Place (📍) in the Insert STK Objects tool.
2. Select the Insert Default method.
3. Click **Insert...**
4. Rename the Place (📍) Radar_Site.

Define 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.
3. Set the following:

Option	Value
Latitude:	35.7517 deg
Longitude:	139.3562 deg
Height Above Ground:	50 ft

4. Click **OK**.

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. Right-click on Radar_Site (📍) in the Object Browser.
7. Select **Zoom To**.
8. Use your mouse to obtain situational awareness of the radar site's location.

Insert 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. Select Sensor () in the Insert STK Objects tool.
2. Select the Insert Default method.
3. Click **Insert...**
4. Select Radar_Site () in the Select Object window.
5. Click **OK**.
6. Rename the Sensor () Servo_System.

Define the Sensor Field of View

Define Servo_System's () 3 degree 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 a target.

1. Open Servo_System's () properties ()
2. Select the Basic - Definition page.
3. Set the Simple Conic - Cone Half Angle: value to 3 deg.
4. Click **Apply**.

Target the Aircraft

Use the Targeted pointing type to point Servo_System () to Target_Acft()

1. Select the Basic - Pointing page.
 2. Set the Pointing Type: to Targeted.
 3. Select Target_Acft () in the Available Targets list.
 4. Move () Target_Acft () to the Assigned Targets list.
 5. Click **Apply** .
-

Set 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.

1. Select the Constraints - Basic page.
 2. Select Min: and Max: in the Elevation Angle section.
 3. Set the Min: value to 0 deg.
 4. Set the Max: value to 30 deg.
 5. Select Max: in the Range section.
 6. Set the value to 150 km.
 7. Click **OK** .
-

Determine Access

Determine when Servo_System () accesses Target_Acft () . This will give you a good idea of when the radar may be able to track the aircraft.

1. Right-click on Servo_System () in the Object Browser.
 2. Select Access () .
 3. When the Access Tool opens, select Target_Acft () in the Associated Objects list.
 4. Click **Compute** .
-

5. Click **Access...** in the Reports section.
6. Click Report Units () in the Access report's toolbar.
7. Select Time Dimension in the Units: Access window.
8. Select Minutes (min) in the New Unit Value list.
9. Click **OK** to close the Units: Access window.

There are two accesses with a total duration of approximately 31 minutes.

10. Close the report.

Generate an Azimuth Elevation Range Report

You have three constraints: a maximum elevation angle of 30 degrees, a minimum elevation angle of 0 degrees, and a maximum range of 150 km. Generate an azimuth-elevation-range (AER) report to see what affect these constraints have on your access report.

1. Return to the Access Tool.
2. Click **AER...** in the Reports section.
3. Look at the Elevation (deg) column.
4. 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.

5. When finished, close the AER report.
6. Close the Access Tool.

Look 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 3.00 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. When finished, click Reset () in the Animation Toolbar.
-

Insert 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. Select *Radar* () in the Insert STK Objects Tool.
2. Select the Insert Default method.
3. Select Servo_System () in the Select Object window.
4. Click **OK** .
5. Rename the *Radar* () ASR.

Model 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.
3. Note that Type: defaults to Monostatic.
4. Note that Mode - Type: defaults to Search Track.

Define the Waveform

The waveform in our 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. Return to ASR's () properties () , Basic - Definition page.
2. Note the Waveform is set to Fixed PRF.
3. Note the default value of 0.001 MHz. We will keep that value since our system's PRF is ~ 1000 Hz.

Define 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. Return to ASR's () properties () , Basic - Definition page.
2. Click the units () drop down next to Pulse Width.
3. Select usec.
4. Set the Pulse Width value to 1 usec.
5. Click **Apply** .

Define the Antenna Pattern

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 on ASR's () properties () , Basic - Definition page.
2. Click the ellipsis button (Component Selector) () beside the Type field.
3. Select Cosine Squared Aperture Rectangular.
4. Click **OK** .
5. Select Use Beamwidth.

6. Set the following:

Option	Value
X Dim Beamwidth:	5 deg
Y Dim Beamwidth:	1.4 deg
Design Frequency:	2.8 GHz
Main-lobe Gain: (if required, turn off Computed)	34 dB
Efficiency:	55 %

7. Click **Apply** .

Define the Radar Transmitter

The transmitter has a frequency range of 2.7-2.9 GHz, a peak power of 20 kW, and uses either linear or circular polarization. We will model linear polarization.

1. Select the Transmitter tab on ASR's () properties () , Basic - Definition page.
2. Select Frequency.
3. Set the value to 2.8 GHz.
4. Set the Power: value to 20 kW.
5. Select the Polarization sub-tab.
6. Select Use.
7. Keep the default setting of Linear.
8. Click **Apply** .

Set 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 on ASR's () properties () , Basic - Definition page.
2. Select the Polarization sub-tab.
3. Select Use.

4. Keep the default setting of Linear.
5. Click **Apply** .

Add the Radar Receiver's System Noise Temperature

Next, let's 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 Compute.
 3. Select Compute in the Antenna Noise section.
 4. Select Sun.
 5. Select Cosmic Background.
 6. Click **OK** .
 7. Save () your scenario.
-

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 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.

Compute Access

Compute Access () between ASR () and Target_Acft () .

1. Right-click on ASR () in the Object Browser.
2. Select Access... () .
3. Select Target_Acft () in the Access Tool's associated objects list.
4. Click **Compute** .

Generate a Radar SearchTrack Report

Now that you calculated Access between ASR () and Target_Acft () , generate a Radar SearchTrack report.

1. Return to the Access Tool.
 2. Click **Report & Graph Manager...** under the Reports section.
 3. Expand (⊕) the Installed Styles list if necessary when the Report & Graph Manager opens.
 4. Select Radar SearchTrack report () .
 5. Click **Generate...** .
 6. Click **Show Step Value** .Show Step Value At the top of the report.
 7. Change the Step: value to 30 sec.
 8. Press Enter on the keyboard or click Refresh () at the top of the report.
-

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**.

Observe 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.

Observe 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.

Observe S/T Ambig Tgt Range

Look at the report to determine if distance from the radar has an effect on the data.

1. Click Report Units () in the Radar SearchTrack report's toolbar.
2. Select Distance Dimension in the Units: Radar SearchTrack window.
3. Select Statute Miles (mi) in the New Unit Value list.
4. Click **OK** to close the Units: Radar SearchTrack window.
5. Scroll over to the right in the Radar SearchTrack report.
6. Locate the S/T Ambig Tgt Range (mi) column. This column tells you how far the aircraft is from the radar site.
7. You see improvement in values discussed above when the aircraft is closer to the radar site.
8. Keep the report open.

Medium Sized Aircraft

Next, simulate a medium sized aircraft.

1. Open Target_Acft's () properties () .
2. Select the RF - Radar Cross Section page.
3. Change the Constant RCS Value: to 10 dBsm.
4. Click **Apply** .
5. Return to the Radar SearchTrack report.

6. Click Refresh () at the top of the report.
 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.
-

Small Sized Aircraft

Simulate a small sized aircraft.

1. Return to Target_Acft's () properties ()..
 2. Change the Constant RCS Value: to 0 dBsm.
 3. Click **Apply** .
 4. Return to the Radar SearchTrack report.
 5. Click Refresh () at the top of the report.
 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.
-

Birds and Stealth

Simulate a bird and a large, somewhat stealthy aircraft.

1. Return to Target_Acft's () properties ()..
2. Change the Constant RCS Value: to -20 dBsm.
3. Click **Apply** .
4. Return to the Radar SearchTrack report.
5. Click Refresh () at the top of the report.
6. Note the S/T Pdet1, S/T Integrated Pdet, S/T SNR1 (dB), and S/T Integrated SNR (dB) changes.
7. Looking at the results, you can only track birds within 10 miles of the airfield.

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.

Load External File

Load an installed Aspect Dependent RCS file.

1. Return to Target_Acft's () properties () .
2. Change Compute Type: to External File.
3. Click the ellipsis button () beside the Filename field.
4. Browse to <STK install folder>\Data\Resources\stktraining\samples\SeaRangeResources\X-47B
5. Select X-47B_Notional_Sample.rcs.
6. Click **Open** .
7. Click **Reload** .
8. Click **OK** .

Visualize RCS Pattern

View the RCS pattern in the 3D Graphics window.

1. Bring the 3D Graphics window to the front.
2. Right-click on Target_Acft's () in the Object Browser.
3. Select Zoom To.
4. Use your mouse to get a good view of the aspect dependent RCS pattern.

View 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 () at the top of the report.
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.

View RCS Data

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 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 (IAgCrdrnIntervalsVectorResult) 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 interval 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 STK with other tools you decide to explore the process.

Solution

Analysis in STK 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 explore how to model a mission with a script.

From this tutorial you will learn how to:

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

Familiarize yourself with the STK Programming Interface Help

Before attempting to code anything, take a moment to familiarize yourself with the STK Programming Interface Help system (<https://help.agi.com/stkdevkit/index.html>).

You will find a wealth of information within the STK Programming 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.



STK Programming Help

- 📁 Welcome to STK Programming Interface Help
- 📁 What's New
- 📁 Migration Guide
- 📁 Tutorials, Code Samples, Code Snippets, and Online Resources
- 📁 Select the Right Technology
- 📁 Using Core Libraries**
 - 📁 STK Object Model
 - 📁 Connect
 - 📁 Development Environments
 - 📁 Library Reference
 - 📁 Getting Help

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

There are several ways to open Jupyter Notebooks. You will try opening it inside STK. You may download the script linked above or follow below. First, create an STK scenario.

1. Create a new STK scenario.
2. Click **Create a Scenario**.

3. Enter the following in the New Scenario Wizard:

Option	Value
Name	STK_Python_Training
Start Time	Default Start Time
Stop Time	Default Stop Time

4. Click **OK** .
-

Integrating STK and Python

Launch the Python Scripting Interface.

1. Click 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 ().
6. If not specified, click the ellipsis button () in the Python Interpreter Path field to set the Python install path.
7. If not 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 be saved to your C:<user path>\Documents\STK 12\PythonScripts.
8. Clear the Create new script on open check box. In this lesson, you will use a notebook file available in your install or from download.
9. Click **Launch** .
10. Browse to the installed Python_Jupyter_STK_Training.ipynb file located at STK Install Folder\Data\Resources\stktraining\scripts. Save the file to the path specified in the **Path to script or folder** option above.

Setting up your workspace

You will be using Jupyter notebooks. You will add cells and enter your script into sections and run them. You may also use the completed script by adding or writing in any missing lines of code. If you are building your script from scratch, use the following commands and paste them into the notebook cell:

1. Set up your workspace.

```
# 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 instance. Use the appropriate scrip below depending on how you connect to STK.

Use STK_PID environment variable when using the Jupiter Notebook plugin and have multiple instances of STK running.

```
STK_PID = os.getenv('STK_PID')

stk = STKDesktop.AttachToApplication(pid=int(STK_PID))
```

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

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

3. Grab a handle on the STK application 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 `stk` 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`.

```
root = stk.Root
```

4. Check that the root object has been built correctly and check the `type()`. The output will be `agi.stk.stkobjects.AgStkObjectRoot`.

```
type(root)
```

5. Once the above lines are entered into the cell, click **Run**. This will create a new STK window.

Connecting and designing a new scenario

Now that you have launched STK 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 cell below** 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**.

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 cell below** to add a new cell.
2. Copy the following code to add a target object to the scenario. 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.

```
# 1. Add a target object to the scenario.
```

```
target = AgTarget(scenario.Children.New  
(AgESTKObjectType.eTarget, "GroundTarget"))
```

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"))
```

```
#Examine the below connect command before running. In it we will be  
using the Set State Classical connect command. Rather than manually  
setting the times we will use the define scenario times. Print them to  
confirm.
```

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

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

5. Copy the following code to propagate the satellite object for the length of the scenario.

```
#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**.

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, expand the following
 - a. 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 JupyterNotebook, click **insert cell below** to add a new cell.
5. Enter the above text to the new cell and click **Run**.

Retrieve 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 cell below** to add a new cell.
2. Copy the following code to add calls to the access data provider. This will retrieve and view the access data in Python.

```
accessDP = access.DataProviders.Item('Access Data')
```

```
results = accessDP.Exec(scenario.StartTime, scenario.StopTime)
```

```
accessStartTimes = results.DataSets.GetDataSetByName('Start  
Time').GetValues()
```

```
accessStopTimes = results.DataSets.GetDataSetByName('Stop  
Time').GetValues()
```

```
print(accessStartTimes,accessStopTimes)
```

3. Click **Run**.
-

Retrieve the satellite altitude data from STK

1. Click **insert cell below** to add a new cell.
2. Copy the following code to add a call to the satellite's LLA state data provider. Retrieve and view the altitude of the satellite during an access interval. In the following lines, note how the data providers must follow the data provider folder, subfolder, and selection.

```
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)
```

3. Click **Run**.
-

Save your scenario

You have just completed the Integrating STK with Python tutorial using Jupyter Notebooks. Don't forget to save your work. With your scenario safely saved, you can close out of STK. 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 you 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.

```
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()
End Sub
```

Task 2 Code

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:
$$=(\text{RAND}()-0.5)*180$$
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:
$$=(\text{RAND}()-0.5)*360$$
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.

Understanding 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 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.

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 use the analytical tool to model real-world aircraft performance that accounts for variations in airframe performance characteristics and mission requirements. In this lesson, mission planners will fly a small commuter jet from Colorado Springs Municipal Airport to Telluride Regional airport. They will quickly add Navigational Aids as waypoints. They will determine how much fuel is required, and how much payload can be carried on board the aircraft in a fast, easy way.

Solution

- Use STK and STK's *Aviator* capability to design a cross country flight route.
- Use the *Aviator* Catalog Manager to:
 - Load airfield runway data into the scenario for takeoff and landing procedures.
 - Load Navigational Aids (NAVAIDs) to simulate physical devices on the ground that aircraft can detect and fly to.
- Use selected Data Providers to determine payload requirements and how much fuel is consumed during the flight.

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_Aviator
Location:	Default
Start:	1 Aug 2020 18:00:00.000 UTCG
Stop:	+ 1 hr

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.
7. Click Save.

Declutter the Labels

Your analysis will take place in very mountainous terrain which can obstruct object labels. Use Label Declutter to separate the labels on objects that are in close proximity for better identification in small areas.

1. Bring the 3D Graphics window to the front.
2. Open the 3D Graphics window properties () .
3. On the Details page, select Enable for Label Declutter.
4. Click OK to accept the changes, and close the Properties Browser.

Aviator Catalog Manager

Aviator provides a catalog structure for the loading and saving of aircraft, airports, nav aids, runways, VTOL points, and waypoints. Each of these mission elements has an associated catalog in STK. 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.

1. Extend the Utilities menu.
2. Select the *Aviator* Catalog Manager.
3. Expand Runway.

4. Select ARINC424 runways.
 5. To the right, click the Use Master Data File ellipses ()
 6. Select My Computer.
 7. Browse to C:\Program Files\AGI\STK 12\Data\Resources\stktraining\samples.
 8. Select the FAANFD18 file.
 9. Click Open.
 10. On the *Aviator* Catalog Manager, click Save.
 11. Keep the *Aviator* Catalog Manager open.
-

Determine the Length of a Runway

The aircraft will fly to and land at Telluride Regional Airport. Determine the length of the runway.

1. In the upper left corner of the *Aviator* Catalog Manager, enter Telluride in the Filter: field.
 2. Click Enter.
 3. Select TELLURIDE RGNL 09 27 in the list under ARINC424 runways.
 4. On the right, you will see data concerning the runway. Locate Length. The runway at Telluride Regional Airport is 7111 feet long.
-

Insert Analytical Objects from the Aviator Catalog Manager

A small commuter jet will takeoff from 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 which exceeds 14,000 feet as well as the runway's location on a plateau with a thousand-foot drop should the aircraft slide off of the runway. Use the *Aviator* Catalog Manager to insert the Colorado Springs Municipal Airport and the Telluride Regional Airport as Place () objects into the scenario.

1. Under ARINC424 runways, right click on TELLURIDE RGNL 09 27.
2. Select Create STK Object from waypoint...
3. Change Type of object: to Place in the Create STK Objects window.

4. Change Color: if desired.
 5. Click OK.
 6. In the upper left hand side of the *Aviator* Catalog Manager, type Colorado Springs in the Filter: field.
 7. Click Enter.
 8. Right click on CITY OF COLORADO SPRINGS MUNI 17L 35R in the list under ARINC424 runways.
 9. Select Create STK Object from waypoint.
 10. Change Type of object: to Place in the Create STK Objects window.
 11. Change Color: if desired.
 12. Click OK.
 13. When finished, close (X) the *Aviator* Catalog Manager.
-

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, and select Zoom To.
 3. Use your mouse to change the view so that you can view the runway and its surroundings.
 4. Right click on TELLURIDE_RGNL_09_27 in the Object Browser, and select Zoom To.
 5. Use your mouse to change the view so that you can view the runway and its surroundings.
-

Insert An Aircraft

Insert an Aircraft () object which you will use to create a flight plan.

1. Using the Insert STK Objects tool, insert an Aircraft () object using the Insert Default () method.
2. Rename the Aircraft () object "Flight_Plan".

3. Open Flight_Plan's () Properties ()

Aviator

Aviator provides an enhanced method for modeling aircraft - more accurate and more flexible than the standard Great Arc 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.

1. On the Basic - Route page, change the Propagator to Aviator.
2. Click Apply.
3. Click Optimize STK for *Aviator* on the Flight Path Warning pop up window. This also changes the animation time to X-Real Animation Time Mode.
4. Click OK to close the Flight Path Warning window.

Mission Window

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.

Initial Aircraft Setup - Select Aircraft

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 window 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. In the Initial Aircraft Setup toolbar, click Select Aircraft.
2. Right click on Basic Business Jet, and select Duplicate.
3. Right click on Basic Business Jet Copy, and select Rename.

4. Rename Basic Business Jet Copy to FlightPlan.
 5. Ensure FlightPlan is selected, and click OK.
-

Initial Aircraft Setup - Aircraft Properties

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.

1. In the Initial Aircraft Setup toolbar, click Aircraft Properties ().

Acceleration Performance Model

The Basic Acceleration performance model is comprised of three tabs:

- Basic: define the basic turning, climb and descent transition, and attitude characteristics of the aircraft
- Aerodynamics: select and define strategies to model attitude characteristics
- Propulsion: select and define strategies to model propulsion characteristics

You want to determine how much fuel is consumed during the flight.

1. Select the Acceleration Built-In Model in the Performance Models section.
2. Ensure the Basic tab is selected.
3. In the Level Turns section, open the pull down menu and select Scale by atmosphere density.
4. In the Climb and Descent Transitions section, select Scale by atmosphere density in the drop down menu.
5. Select the Aerodynamics tab.
6. Change the Strategy: to Basic Fixed Wing.
7. Select the Propulsion tab.
8. Change the Strategy: to Basic Fixed Wing.
9. Click Save.

Notice that the Mode: is set to Jet - Specify Net Thrust. You will use the default values for both aerodynamics and propulsion, but they will have an affect on fuel consumed. All of this data would be changed according to the actual specifications of your aircraft.

Climb Performance Model

The Basic Climb performance model is comprised of a simple set of parameters that define the flight characteristics of the aircraft while climbing.

1. Select the Climb Built-In Model in the Performance Models section.
2. Select Use Aero/Propulsion Fuel Flow.
3. Click Save.

Cruise Performance Model

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 25000 feet and level off.

1. Select the Cruise Built-In Model in the Performance Models section.
2. Change the Default Cruise Altitude: to 25000 ft.
3. Select Use Aero/Propulsion Fuel Flow.
4. Click Save.

Descent Performance Model

The Basic Descent performance model is comprised of a simple set of parameters that define the flight characteristics of the aircraft while descending.

1. Select the Descent Built-In Model in the Performance Models section.
2. Enable Use Aero/Propulsion Fuel Flow.
3. Click Save.

Landing Performance Model

The Basic Descent performance model is comprised of a simple set of parameters that define the flight characteristics of the aircraft during a landing.

1. Select the Landing Built-In Model in the Performance Models section.
2. Enable Use Aero/Propulsion Fuel Flow.

3. Click Save.

Takeoff Performance Model

The Basic Takeoff performance model is comprised of a simple set of parameters that define the flight characteristics of the aircraft during a takeoff.

1. Select the Takeoff Built-In Model in the Performance Models section.
 2. Enable Use Aero/Propulsion Fuel Flow.
 3. Click Save.
 4. Click Close.
 5. Click Apply to apply changes, and keep the Properties Browser open.
-

Initial Aircraft Setup - Configuration

The Configuration window is used to define the aircraft's fuel and payload configuration.

1. In the Initial Aircraft Setup toolbar, click Configuration ()
2. Change the Empty Weight: value to 31000 lb. This is where you add payload weight. This will account for the pilot, instructor, and baggage.
3. Note the Max Landing Weight value. If, at the end of the analysis, the Aircraft () object weighs more than this value, a warning will appear in Message Viewer.
4. Note the Total Weight value. This is the empty weight and the fuel load of the Aircraft () object.
5. Select the Stations tab.
6. Click Internal Fuel.
7. Note the Capacity and Initial state values. After your initial analysis, you may need to adjust the initial state.
8. Click OK.
9. Click Apply to apply changes, and keep the Properties Browser open.

Initial Aircraft Setup - 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.

1. In the Initial Aircraft Setup toolbar, click Mission Wind Model () .
 2. Change the Wind Bearing: value to 180 deg.
 3. Change the Wind Speed: value to 20 nm/hr.
 4. Click OK.
 5. Click Apply to apply changes, and keep the Properties Browser open.
-

Flight Plan

In this scenario, you will use one phase. You can have multiple phases, if you desire. Each phase can contain a separate performance model for the Aircraft () . The aircraft will takeoff from 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 and land at Telluride Regional Airport.

Site Properties - Select 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. In the Mission List window, right click on Phase 1 and select Insert First Procedure for Phase () .
2. Select Runway from Catalog () in the Site Properties / Select Site Type: section.
3. Enter Colorado Springs in the Filter: field.
4. Click Enter.
5. Under ARINC424 runways, select CITY OF COLORADO SPRINGS MUNI 17L 35R.
6. Click Next.

Procedure Properties - Takeoff

A Takeoff procedure launches an aircraft from a runway site into the air.

1. Select Takeoff () in the Procedure Properties window / Select Procedure Type: section.
2. Set the following:

Option	Value
Name:	COS Runway
Use runway heading 172 Mag 180 True (Headwind)	enabled
Runway Altitude Offset:	7 ft
Use Terrain for Runway Altitude	enabled

3. Click Finish.
4. Click Apply to apply changes, and keep the Properties Browser open.

Site Properties - End of Previous Procedure

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/DME.

1. Right click on CITY OF COLORADO SPRINGS MUNI 17L 35R in the Mission List, and select Insert Procedure After ()
2. Select End of Previous Procedure () in the Site Properties window / Select Site Type: section.
3. Change the Name: to Climb.
4. Click Next.

Procedure Properties - Basic Maneuver Climb Before Turning

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 Procedure Properties window / Select Procedure Type: section.
2. Change the Name: to Straight 20 nm.

3. In the Horizontal / Navigation tab settings, note that the strategy: is set to Straight ahead. Open the pull down menu to view other strategies.
4. Set the following :

Option	Value
Strategy:	Straight Ahead (default)
Basic Stop Conditions - Time of Flight:	disable
Basic Stop Conditions - Downrange:	20 nm

5. Select the Vertical / Profile tab.
6. Set the Altitude - Mode: to Specify Altitude Change.
7. Set the Altitude - Relative Altitude Change: value to 10000 ft.
8. Click Finish two times.
9. Click Apply to apply changes, and keep the Properties Browser open.

Site Properties - Navaid Fly to Blue Mesa

If you have ARINC424 navaid data available in the *Aviator* Catalog Manager, you can define a site using a navaid from that data.

1. Right click on Climb in the Mission List, and select Insert Procedure After ()
2. Select Navaid from Catalog () in the Site Properties window / Select Site Type: section.
3. Enter HBU in the Filter: field. HBU is the FAA designator for Blue Mesa VOR/DME.
4. Click Enter.
5. Right click on HBU, and select Create STK Object from waypoint...
6. Set Type of object: to Place.
7. Change the Color: if desired.
8. Click OK.
9. Click Next.

Procedure Properties - Basic Point to Point

A Basic Point to Point procedure is a basic traverse between two waypoints.

1. Select Basic Point to Point () in the Procedure Properties window / Select Procedure Type: section.
2. Set the following:

Option	Value
Name:	Blue Mesa
Navigation Options - Nav Mode:	Fly Direct
Enroute Options - Turn Factor	5.00 (less roll)

3. Click Finish.
4. Click Apply to apply changes and keep the Properties Browser open.

Site Properties - Navaid Fly to Cones

1. Right click on HBU in the Mission List, and select Insert Procedure After ()
2. Select Navaid from Catalog () in the Site Properties window / Select Site Type: section.
3. Enter ETL in the Filter: field.
4. Click Enter.
5. Right click on ETL, and select Create STK Object from waypoint...
6. Set Type of object: to Place.
7. Change the Color: if desired.
8. Click OK.
9. Click Next.

Procedure Properties - Basic Point to Point

1. Select Basic Point to Point () in the Procedure Properties window / Select Procedure Type: section.
2. Set the following:

Option	Value
Name:	Cones

Altitude - Use Aircraft Default Cruise Altitude	clear
Altitude - Altitude:	15000 ft
Enroute Options - Turn Factor	5.00
Enroute Cruise Airspeed - Pull-down menu	Other Airspeed
Enroute Cruise Airspeed - Airspeed:	250 nm/hr

3. Click Finish.
4. Click Apply to apply changes and keep the Properties Browser open.

Site Properties - Select Runway

1. Right click on ETL in the Mission List, and select Insert Procedure After (.
2. Select Runway from Catalog () in the Site Properties / Select Site Type: section.
3. Enter Telluride in the Filter: field.
4. Click Enter.
5. Select TELLURIDE RGNL 09 27 under ARINC424 runways.
6. Click Next.

Procedure Properties - Landing

A Landing procedure brings an aircraft down from the air to a runway site.

1. Select Landing () in the Procedure Properties window / Select Procedure Type: section.
2. Set the following:

Option	Value
Name:	Telluride Runway
Approach Mode:	Intercept Glideslope
Use runway heading 096 Mag 105 True (Headwind)	selected
Runway Altitude Offset:	7 ft
Use Terrain for Runway Altitude	selected

3. Click Finish.
 4. Click Apply to apply changes and leave the Properties Browser open.
-

Message Viewer

STK uses the Message Viewer window to display error messages, warning messages, and informational messages. Currently, there is a warning in Message Viewer.

1. In the STK Menu area, open the View menu and select Message Viewer. Look at the last message. Expand as necessary. The maximum landing weight for the aircraft is 40000 pounds. It is too heavy.
 2. At the bottom of Message Viewer, you'll see a tab named All Messages. Right click on the All Messages tab and select Clear All Tabs.
 3. Close (✕) the Message Viewer.
-

Flight Profile by Time Custom Report

The data provider Flight Profile By Time has the elements required for your analysis. Flight data is sampled using a constant time step between grid points. This report style is only available for *Aviator* propagated vehicles.

1. In the Object Browser, right click on Flight_Plan () , and select the Report & Graph Manager.
2. In the Styles list, right click on the Flight Profile by Time report and select Properties.
3. In the Report Contents list, select Flight Profile By Time-Fuel Consumed. This opens the Flight Profile By Time Data Provider on the left.
4. In the Flight Profile By Time Data Provider, move () Fuel State and Weight elements to the Report Contents list.
5. Click OK.
6. Click OK to close the Warning.
7. In the Styles list, expand the My Styles folder.
8. Right click the Flight Profile by Time report in the My Styles folder, and select Rename.
9. Rename the report "My Flight Profile by Time".
10. Click Generate

Focusing on fuel consumed, fuel state and the weight of the aircraft, you can see that the plane is approximately 3238 pounds above its maximum landing weight of 40000 lbs when it touches down. You landed with approximately 12238 pounds of fuel.

11. Leave the My Flight Profile by Time report open.

Adjust the Initial Fuel State

You have the option of adjusting the initial fuel state and payload. In this instance, adjust the initial fuel state. Keep in mind you want reserve fuel. You landed with approximately 12238 pounds of fuel. Remove 6000 pounds of fuel.

1. Return to Flight_Plan's () Properties ()
2. In the Initial Aircraft Setup toolbar, click Configuration ()
3. Select the Stations tab in the Aircraft Configuration window.
4. Select Internal Fuel.
5. Change the Initial state: value to 14000 lb.
6. Click Apply.
7. Click OK.
8. Click OK to close Flight_Plan's () Properties Browser.

Refresh the My Flight Profile by Time Report

1. Bring the My Flight Profile by Time report to the front.
2. Refresh () the report.

Focusing on fuel consumed, fuel state and the weight of the aircraft, you can see that the plane is approximately 2614 pounds below its maximum landing weight. You landed with plenty of reserve fuel. Furthermore, you needed less fuel to arrive at your destination.

3. Look at Message Viewer. Due to adjusting the weight of the aircraft, there are no new warnings.
4. When finished, close () the report and the Report & Graph Manager.
5. Close () the Message Viewer.

2D Graphics Window Flight Route View

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
-

View the Flight

Use the 3D Graphics window to observe the mission.

1. Bring the 3D Graphics window to the front.
 2. Reset () the scenario.
 3. Zoom To Flight_Plan.
 4. Adjust (, ) the time step as desired.
 5. Start () the scenario.
 6. When finished, Reset () the scenario.
 7. You can Zoom To ETL and HBU to view the actual VOR/DME ground transmitter sites.
-

Summary

You began by loading and saving of aircraft, airports, nav aids, runways, VTOL points, and waypoints using the Aviator Catalog Manager. Using *Aviator*, you mission planned for a small commuter jet taking off from Colorado Springs Municipal Airport and landing at Telluride Regional Airport. During the initial aircraft configuration, you added 1000 pounds to the aircraft to account for increased payload. You tweaked performance models to determine how much fuel was required for the flight. The wind bearing for the entire mission was out of the south at 20 miles per hour. After takeoff, you flew straight ahead to gain altitude due to mountainous terrain. After 20 miles, you turned and flew direct to Blue Mesa VOR/DME. Making a slight turn, you continued to Cones VOR/DME. You were able to line up the aircraft by arriving at Cones with a heading taking you directly to Telluride. After landing in Telluride, you viewed a warning in Message Viewer that informed you of the aircraft's weight being over the maximum allowed weight of 40,000 pounds. You then used a customized Flight Profile by Time report to determine how much the aircraft weighed during landing and how much fuel you still had on

board. Returning to the aircraft configuration settings, you adjusted the initial fuel load. Re-analyzing the mission, you were able to land in Telluride under the maximum allowable weight limit with plenty of reserve fuel.

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

- When you finish, click **OK**.
- 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.
- 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.

- Close the Timeline View at the bottom of the STK window.
- Close the 2D Graphics window.
- 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.

- Select the scenario STK_CAT () and click properties () .
- In the properties window, select the Basic - Database page.
- Click **Update Database Files**.
- 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.

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:

1. A basic launch phase to place a satellite into a parking orbit (low Earth orbit or LEO)
2. A transfer orbit injection (TOI) maneuver to transfer from LEO to GEO
3. A synchronized orbit injection (SOI) maneuver to circularize the orbit at GEO

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_Astrogator
Location:	Default

Start:	Default
Stop:	+ 5 days

- When finished, click **OK**.
- When the scenario loads, click Save () . A folder with the same name as your scenario is created for you in the location specified above.
- Verify the scenario name and location.
- Click **Save**.

Updating the Insert STK Objects tool

Ensure the Launch Vehicle () appears in the Insert STK Objects tool.

- Click **Edit Preferences...** on the Insert STK Objects tool.
- In the New Object list, select the object you want to add.
- Click **OK**.

Creating a Launch Vehicle

Insert a Launch Vehicle object.

- Select Launch Vehicle () in the Insert STK Objects tool.
- Select the Insert Default method.
- Click **Insert...**
- 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.

1. Open LaunchToLEO's () properties ()
 2. Select the Basic - Trajectory page.
 3. Ensure Propagator is set to SimpleAscent.
 4. Set the Burnout Velocity value to 7.3 km/sec. This will keep the resulting orbit near circular.
 5. Click **Apply** to accept the changes and keep the Properties Browser open.
 6. Select the 2D Graphics - Attributes page.
 7. Change the Color: to teal.
 8. Click **OK** to accept the changes and 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. Select Satellite () in the Insert STK Objects tool.
2. Select the Insert Default method.
3. Click **Insert...**
4. Rename Satellite () GEO_Sat.

Using Astrogator

Set GEO_Sat's Propagator to *Astrogator*. You will use *Astrogator* to design your spacecraft trajectory.

1. Open GEO_Sat's () properties () .
 2. Select the Basic - Orbit page.
 3. Set Propagator to *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 () .
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 the Propagate () segment in the MCS.
2. Select Insert Before...
3. Select Follow () in the Segment Selection window.
4. Click **OK**.
5. Ensure Follow () is selected in the MCS.
6. Go to the General tab on the Basic - Orbit page.
7. Click the Leader ellipsis button ()
8. Select LaunchToLEO () in the Select Leader window.
9. Click **OK**.
10. Set Joining to **Join at End of Leader's Ephemeris** in the Additional Options section.

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**.

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. When finished, 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. Click Component Browser () in the MCS Toolbar.
2. Change Show: to *Astrogator* Components on the Component Browser window.
3. Select Engine Models in the Components list on the left.
4. Select Constant Thrust and Isp in the Engine Models list on the right.
5. Click duplicate () .
6. Change Name to Test Engine in the Field Editor window.
7. Click **OK** .
8. 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 in the Engine Models - Test Engine window.
2. Set Real Number: to 13500 N in the Real Number Field window.
3. Click **OK** .
4. Double-click Isp in the Engine Models - Test Engine window.
5. Set Real Number to 2000 seconds in the Real Number field window.
6. Click **OK** .

7. Click **OK** to close the Engine Models - Test Engine window.
 8. 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 Propagate () in the MCS.
3. Select Rename.
4. Change the name to Prop_to_TOI.

Updating the Stopping Condition

Update the Propagate Stopping Condition to stop at the ascending node after two full revolutions in the parking orbit.

1. Click New () in the Stopping Conditions section.
2. Select AscendingNode in the New Stopping Condition window.
3. Click **OK** .
4. Select the Duration stopping condition.
5. Click Delete () .
6. Change Repeat Count to 2 so will stop it on the second ascending node.
7. Click **Apply** .
8. Click Run Entire Mission Control Sequence () .

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...
4. Select Target Sequence () in the Segment Selection window.
5. Click **OK** .
6. Right-click Target Sequence () in the MCS.
7. Select Rename.
8. Change the name 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...
3. Select Maneuver () in the Segment Selection window.
4. Click **OK** .

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

5. Drag and drop the Maneuver () Segment below Start_Transfer ().
6. Right-click the Maneuver () Segment.
7. Select Properties...
8. Change Name to TOI.

9. Change Color to white.

10. Click **OK** .

Insert a Propagate Segment

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

1. Right-click TOI () in the MCS.
2. Select Insert After...
3. Select Propagate () in the Segment Select window.
4. Click **OK** .
5. Right-click the new Propagate () Segment.
6. Select Properties...
7. Set Name to Transfer.
8. Set Color to Yellow.
9. Click **OK** .
10. Click **Apply** .

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 button () in the Propulsion Type section.
4. Select Test Engine () in the Select EngineModel window.
5. Click **OK** .

Selecting the control parameter

Select Delta-V Magnitude as the independent variable.

1. Select the Attitude tab.
2. The default Attitude Control is Along Velocity Vector (body X).
3. Click the target icon () to target the Delta-V Magnitude and make it the independent variable.

This tells *Astrogator* to determine the Delta-V magnitude based on user determined results. You will set up those results in an upcoming section.

4. Click **Apply**.

Propagating to apoapsis

Update the Transfer () Stopping Condition to stop at apoapsis.

1. Select Transfer () in the MCS.
2. Click New () in the Stopping Conditions section.
3. Select Apoapsis () in the New Stopping Condition window.
4. Click **OK**.
5. Select Duration in the Stopping Conditions section.
6. Click Delete () .
7. Click **Apply**.

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 Elms in the User - Selected Results window's Available Components list.
4. 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.
5. Click **OK**.

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. Change Action to **Run active profiles**.
3. Select Differential Corrector in the Profiles section.
4. Click Properties ().

Setting the control parameter

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

1. Locate the Control Parameters section of the Differential Corrector window.
2. Select Use for ImpulsiveMnvr.Pointing.Spherical.Magnitude.

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. Locate the Equality Constraints (Results) section of the Differential Corrector window.
2. Select Use for R_Mag.
3. Select the Desired Value field.
4. Set Desired Value to 42238 km.
5. Click **OK** .
6. Click **Apply** .

Running the Entire Mission Control Sequence

1. Save () your scenario.
2. Click Run Entire Mission Control Sequence ().

3. 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.
 4. Look at the StartTransfer.Differential Corrector data window, which shows you data based on running the active profile.
 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...
4. Select Target Sequence () in the Segment Selection window.
5. Click **OK**.
6. Right-click Target Sequence () in the MCS.
7. Select Rename.
8. Change the name 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...
3. Select Maneuver () in the Segment Selection window.
4. Click **OK**.

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

5. Drag and drop the Maneuver () Segment below the Finish_Transfer () .
6. Right-click the Maneuver () Segment.
7. Select Properties...
8. Change Name to SOI.
9. Change Color to white.
10. Click **OK** .

Inserting a Propagate segment

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

1. Right-click SOI () in the MCS.
2. Select Insert After...
3. Select Propagate () in the Segment Select window.
4. Click **OK** .
5. Right-click the new Propagate () Segment.
6. Select Properties...
7. Set Name to Prop1Rev.
8. Set Color to Blue.
9. Click **OK** .
10. Click **Apply** .

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 button () in the Propulsion Type section.
4. Select Test Engine () in the Select EngineModel window.
5. Click **OK**.

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. Set Attitude Control to Thrust Vector.
3. Click the target icon () next to X (Velocity).
4. Click the target icon () next to Y (Normal).
5. Click **Apply**.

Propagating for one day

Update the Prop1Rev () Duration Stopping Condition to stop after one day.

1. Select Prop1Rev () in the MCS.
2. Change Trip to 86400 sec (1 day) in the Stopping Conditions section.
3. Click **Apply**.

Selecting the results variable

Set Eccentricity and Inclination as the equality constraints.

1. Select Prop1Rev () in the MCS.
2. Click **Results...** at the bottom of the MCS.
3. Expand () Keplerian Elms in the User - Selected Results window's Available Components list.
4. Move () Eccentricity () to the selected components list.

5. Move () Inclination () to the selected components list.
6. Click **OK**.

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. Change Action to Run active profiles.
3. Select Differential Corrector in the Profiles section.
4. Click Properties ()

Setting the control parameter

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

1. Locate the Control Parameters section of the Differential Corrector window.
2. Select Use for ImpulsiveMnvr.Pointing.Cartesian.X.
3. Select Use for ImpulsiveMnvr.Pointing.Cartesian.Y.

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. Locate the Equality Constraints (Results) section of the Differential Corrector window.
2. Select the Use for Eccentricity check box.
3. Set Tolerance to 0.001.
4. Select the Use for Inclination check box.
5. Select the Desired Value field.
6. Set Desired Value to 2 deg.
7. Set Tolerance to 0.01 deg.
8. Set Method to **By tolerance** in the Scaling section.

9. Click **OK** .
 10. Click **Apply** .
-

Running the Entire Mission Control Sequence

1. Save () your scenario.
 2. Click Run Entire Mission Control Sequence () .
 3. Look at the Finish Transfer.Differential Corrector data window, which shows you data based on running the active profile.
 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...
 4. Select Propagate () on the Segment Selection window.
 5. Click **OK** .
 6. Click Run Entire Mission Control Sequence () .
 7. 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. When finished, close () the summary report.
 5. Select Finish_Transfer ()
 6. Click Summary () in the MCS Toolbar.
 7. When finished looking through the report, close any reports, tools and properties you still have open.
 8. Save () your work.
-

Summary

This was a basic introduction to *Astrogator*. In this lesson, you did the following:

1. Used a Launch Vehicle object to launch a vehicle that followed an ascent trajectory from a launch point to an orbit insertion point.
2. 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.
3. Used a Target sequence, a Maneuver segment, and a Propagate segment to place the Satellite orbit into a TOI.
4. Finalized its orbit by creating another Target Sequence that placed the Satellite into a GEO.
5. Used an MCS Segment Summary Report to determine maneuver times, required Delta-V, estimated burn duration, and estimated fuel usage.

Part 16: 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: The EOIR installation package is included with the STK Premium installation package. However, you must install EOIR on your computer separately from the STK application. See the [EOIR capability help](#) for more information.

Capabilities covered

This lesson covers the following STK Capabilities:

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

Define the scenario's assets

You will be using the MSSC 1.6m telescope at the AMOS observatory in Maui, Hawaii, to track a polar satellite in LEO. To create a scenario and define its assets:

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

Option	Value
Name	STK_EOIR
Location	C:\Users\<username>\Documents\STK 12\
Start	1 Jan 2018 16:00:00 UTCG
Stop	+ 10 min

3. When you finish, click **OK** .

4. When the scenario loads, click Save (). A folder with the same name as your scenario is created for you in the location specified above.
 5. Verify the scenario name and location and click **Save** .
-

Insert a Facility object

Use the Insert STK Objects tool to add the MSSC_1_6m facility from the Standard Object Database.

1. Bring the Insert STK Objects tool to the front.
 2. Insert a Facility () object using the From Standard Object Database () method.
 3. Click **Insert...** .
 4. Enter MSSC in the Name field and click **Search** .
 5. Select 'MSSC 1.6m' and click **Insert** .
 6. Click **Close** to return to the Insert STK Objects tool.
-

Verifying that EOIR is installed

Ensure that *EOIR* is installed on your computer.

1. If you do not see the *EOIR* toolbar (), extend the View menu.
 2. Select the Toolbars option
 3. Select *EOIR*.
-

Adding a sensor to the MSSC_1_6m facility

Add a sensor to the facility you just added to the scenario.

1. Insert a Sensor () object using the Insert Default () method.
 2. Select MSSC_1_6m () and click **OK** .
-

3. Attach a sensor to the MSSC_1_6m facility.

Setting the EOIR properties for the sensor

You can set up this sensor to model the large AEOS telescope at AMOS.

SETTING THE SPATIAL PROPERTIES

1. Open the sensor's properties.
2. Set the sensor type to *EOIR* on the sensor's Basic - Definition page.

You will set up this sensor to model the large AEOS telescope at AMOS. The first tab contains the spatial information of the sensor. This is where you define the total field-of-view angles and the number of pixels on the sensor detector.



Note: For a more detailed explanation of the *EOIR* property windows, click the Help button on the bottom of the properties page.

3. Set the following parameters:

Option	Value
Field of View Horizontal Half Angle	7.5 deg
Field of View Vertical Half Angle	7.5 deg



Note: Since you left the default setting for Input, which 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 click **Apply**. These are read-only fields.

SETTING THE SPECTRAL PROPERTIES

1. Select the Spectral tab.

This tab is where you define the spectral range of your sensor. 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.

2. Set the following Spectral Parameters:

Option	Value
Spectral Band Edge Wavelengths - Low	0.4 um
Spectral Band Edge Wavelength- High	0.7 um



Note: The telescope observes the long infrared waveband. If you would like to do your analysis with those parameters, then set the Spectral Band Edge Wavelengths Low to 0.7 μm and High to 1.0 μm .

SETTING THE OPTICAL PROPERTIES

1. Select the Optical tab.
2. In the Input field, select F-Number and Entrance Pupil Diameter.

Option	Value
F/#	200
Entrance Pupil Diameter	367.00 cm
Image Quality	Negligible Aberrations



Note: The Image Quality property models wave front error through the optics. The Negligible Aberrations setting introduces 7% wave front error.

SETTING THE RADIOMETRIC PROPERTIES

1. Select the Radiometric tab.

This is where you define the noise floor and the saturation ceiling. You can define a set of points that relate Integration (Exposure) Time to NEI/SEI. STK linearly interpolates between the points to get correct NEI/SEI for the integration time you set at the bottom of the window.

2. Set the following Sensitivity parameters:

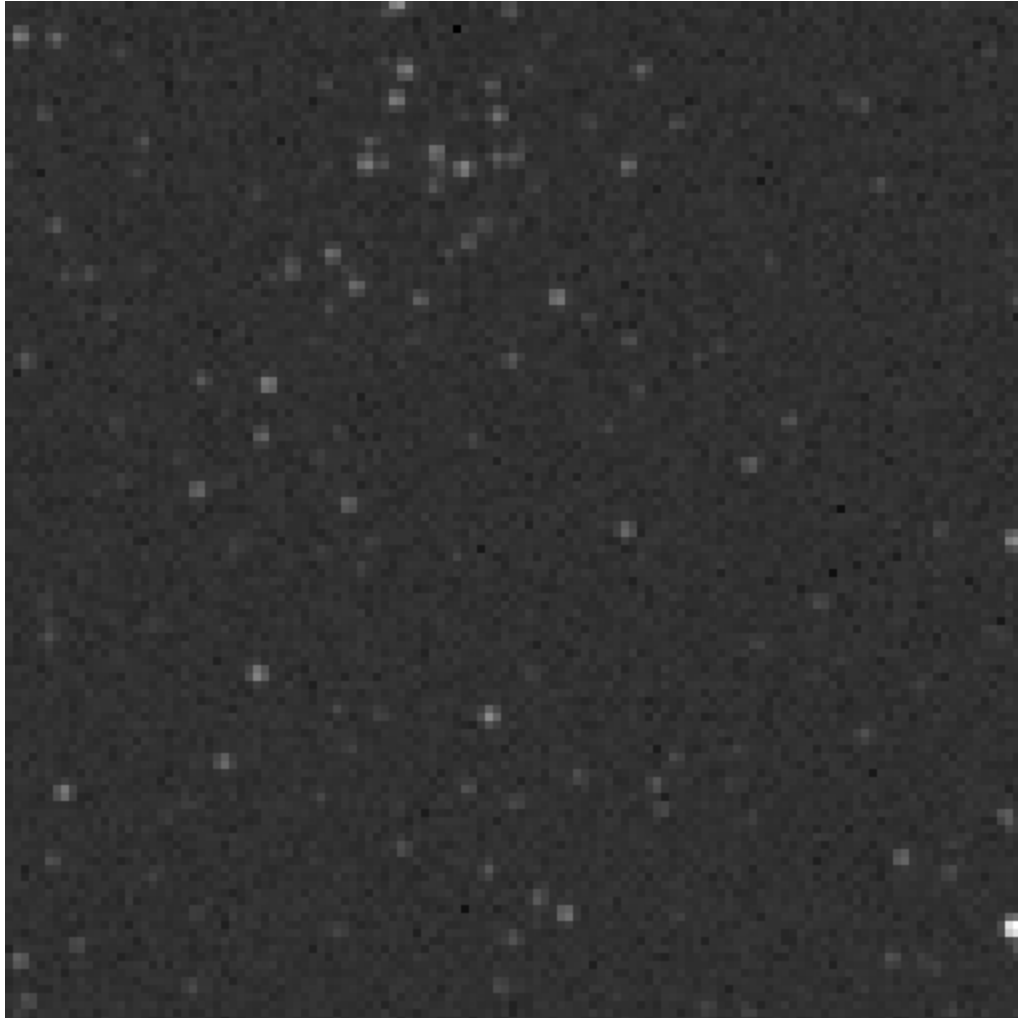
Option	Value
Sensitivity - Integration Time	100
Sensitivity - Equivalent Value	1e-16

3. Set the Processing Level to Sensor Output. The 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. Set the Line of Sight Jitter to 1 mrad to introduce a Gaussian vibration along the sensor boresight.
5. Click **OK** to apply and save your changes.
6. Open the Target Configuration () window. This window defines the objects used in the EO/IR analysis.
7. Verify that Facility/MSSC_1_6m is in the Selected Targets list.

Generate EOIR sensor scenes

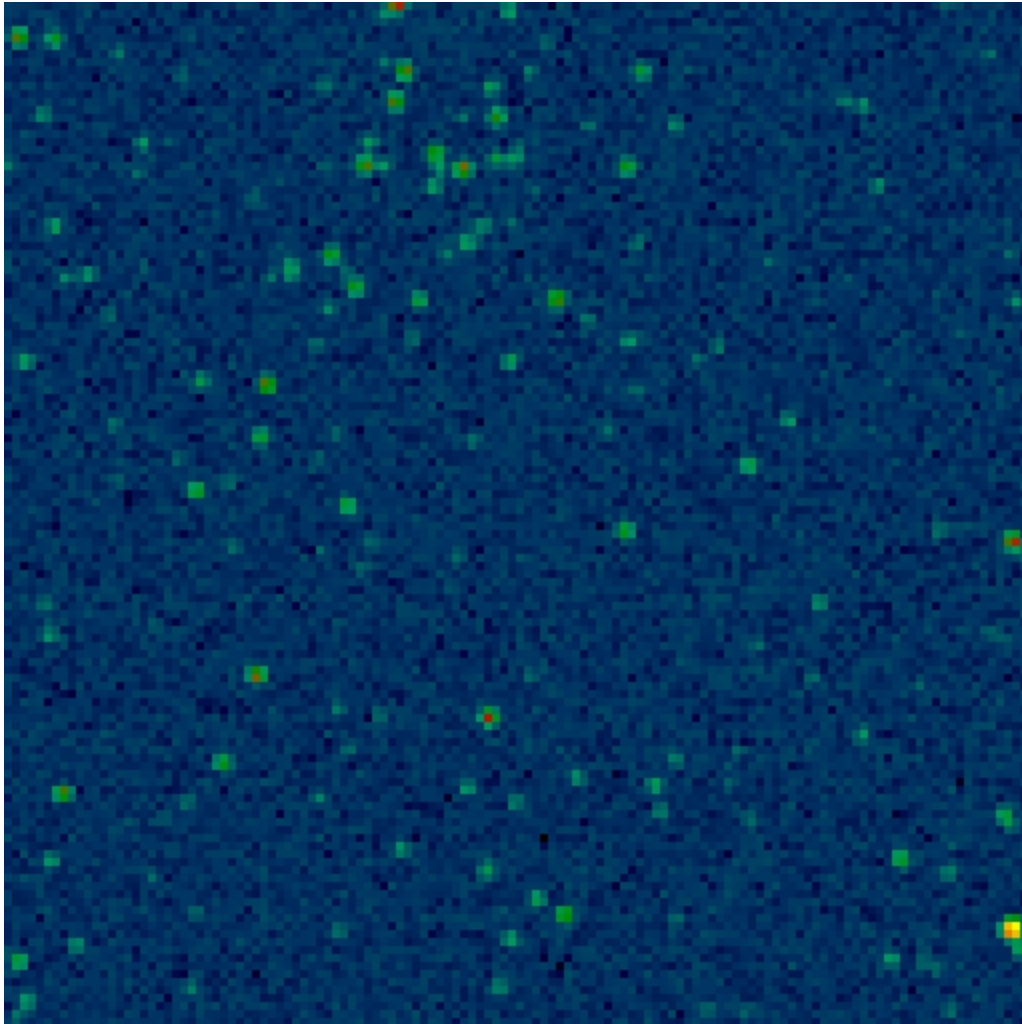
Now you are ready to generate sensor scenes. These scenes accurately portray sensor images for the processing level selected.

1. Select the sensor in the Object Browser and click *EOIR Sensor Scene* (🔧) to generate an image that represents the radiometric input to the sensor. You will see some white dots and gray dots against a black background.



2. Right-click the sensor scene and select *Details....*. Here you can 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. Set the Color Map to BGRY and click **Apply** .
3. 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.



4. Click around the scene to display information on the *EOIR* Scene Visual Details window for each pixel. Click one of the stars to get more details on this object.
5. Close the *EOIR* Scene Visual Details properties page and the *EOIR* Sensor Scene window.

Creating a satellite that flies over the facility

Using the Orbit Wizard, create a satellite that flies over the MSSC_1_6m.

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

Option	Value
Type	Circular (default)
Inclination	98 deg
Altitude	500 km (default)
RAAN	189 deg

3. Click **OK** to insert the new satellite.

Zoom to the satellite

1. Zoom To the satellite in the 3D Graphics window. Pan and zoom around so that you can view both the satellite and the facility.
2. Set the Animation Time Step to one (1) second and 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.



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.

Examine the satellite's basic EOIR Shape

Examine the material and shape properties of your satellite.

1. Open the Satellite's properties.
2. Open the Basic - *EOIR* Shape properties page.
3. Examine the following options:

Option	Value
Shape	Sphere (default)
Radius	1 m

Body Temperature	Static (default)
Temperature	273 K (default)
Material	Gray Body (default)



Note: This temperature is steady and uniform over the body and corresponds to its external temperature in sunlight.

4. Open the Target Configuration () box and add Satellite/Satellite1 to the Selected Targets list.



Note: Available objects include central bodies, satellites, facilities, missiles, and aircraft. Both targets and sensor parent objects must be included as Selected Targets.

5. Click **OK** to close the *EOIR* Configuration window.

Run an Access report

Run an access report between the sensor and the satellite.

1. Right-click the sensor object and select Access... from the Sensor menu.
2. Select Satellite1 from the list.
3. Click **Advanced...** button and disable Use Light Time Delay. Light time delay is not used in *EOIR* analysis.
4. Click **OK** .
5. Click **Access...** in the Reports section.

The Access report shows one access interval at or near the following times:

- 1 Jan 2018 16:05:19.723
- 1 Jan 2018 16:05:37.186

6. Right-click the start time in the Access report and extend the Start Time menu.
7. Select the Set Animation Time option. This sets the animation time to the access time, which will be reflected in the animation field.

Keep the access report open.

Regenerating the EOIR Sensor Scene

1. Select the sensor in the Object Browser and click *EOIR* Sensor Scene () to generate an image that represents the sensor output.
You will see colorful dots against a dark background.
 2. Set the Animation Time Step to 0.5 seconds and animate the scenario until you see the satellite come into the scene. The dot that represents the satellite moves across the scene while the stars stay relatively still.
 3. Right-click the sensor scene and select Details....
 4. Click around the scene to view information on each pixel. Click one of the stars and the target satellite to get more details on those objects.
 5. Close the *EOIR* Scene Visual Details properties page and the *EOIR* Sensor Scene window.
-

Create custom graphs for EO/IR Sensors

EOIR does more than simulate scenes created by an EO/IR 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.

1. Open the Report & Graph Manager for the sensor on the facility.
2. Click the Create a New Graph () button to create a custom graph for the sensor.
3. Name the new graph *EOIR* Sensor-to-TargetMetrics.
4. Open up the graph's Content properties.
5. In the *EOIR* Sensor to Target Metrics data provider, select the following data elements:
 - Y Axis: Effective target irradiance
 - Y2 Axis: Signal to noise ratio
6. Set the following titles:

Option	Name
Time Axis Title	Time
Y Axis Title	Effective Target Irradiance
Y2 Axis Title	Signal to Noise Ratio (SNR)

7. Select *EOIR* Sensor to Target Metrics-Effective target irradiance under Y Axis and click **Units...** below the Y2 Axis box.
 8. Clear the Use Defaults check box
 9. Select Power.
 10. Select Watts (W) as the new unit value.
 11. Click **OK** to close the Units window.
 12. Click **OK** to finalize your custom graph and return to the Report & Graph Manager.
-

Setting the Time properties

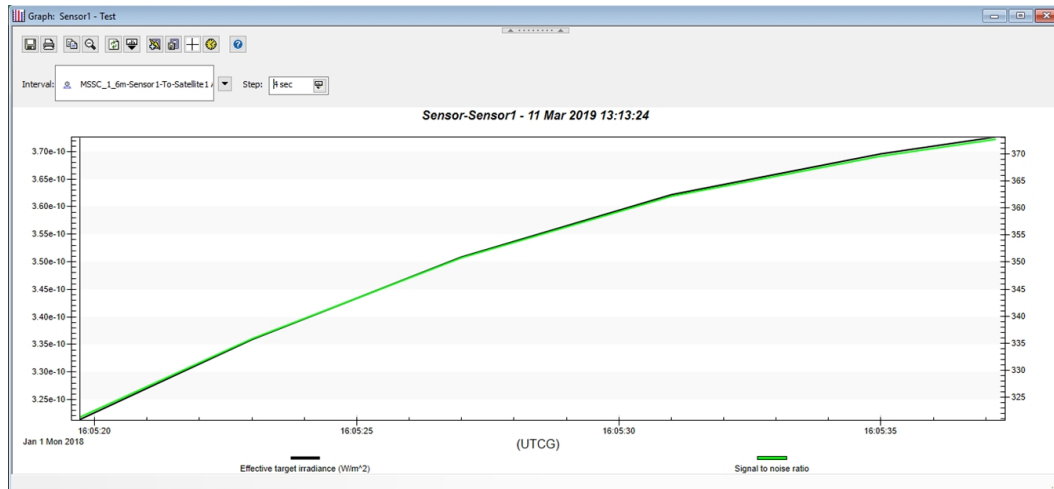
1. Select Specify Time Properties under the Time Properties section.
 2. In the Select type field, select Specify Times.
 3. Click the  button to the right of the Start and Stop times and select Interval Component....
 4. In the Select Time Interval window, select the Access object ().
 5. Under Installed Components, expand Access Intervals.
 6. Select First (since there is only one access interval) and click **OK** to return to the Report & Graph Manager.
 7. Enable the Use step size/time bound option.
 8. Set the Step Size to 4 seconds.
-

Generating the custom graph

1. Generate the graph.
2. Take a look at the graph. You should see a graph similar to the one below.



Note: *EOIR* calculations are computationally expensive. It is recommended that you confine your report and graph times to access intervals and choose your time steps wisely.



This graph shows the signal is large relative to the noise by up to a factor of 373 to 1 which is why you are able to see the target clearly in the scene. Keep the graph open.

Defining more EOIR properties

1. Open the Target Configuration () box.
2. Click **Atmosphere and Textures** to display the *EOIR* Atmosphere Model properties.
3. Take a minute to view the different atmosphere models available with *EOIR*, Simple Atmosphere and MODTRAN Atmosphere
 - The 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.
 - The MODTRAN-based atmosphere model is one of the highest fidelity, community standard, atmospheric models available in *EOIR*. As such, it takes longer to compute.
4. Select Simple Atmosphere and set the following parameters:

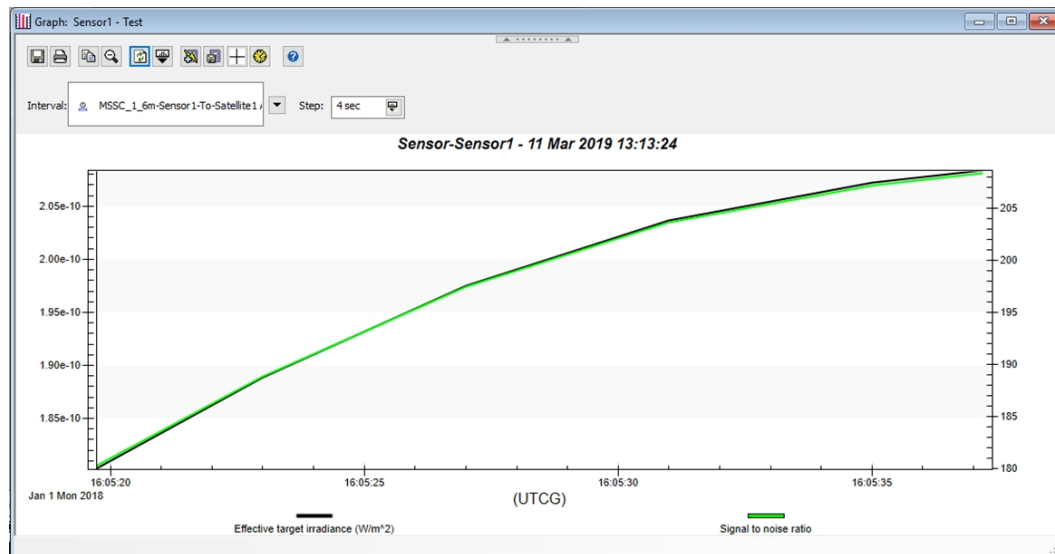
Option	Value
Aerosol Models	Maritime
Visibility	40 km
Humidity	70 %

5. Click **OK** to close the *EOIR* Atmosphere Model; then click **OK** to close the *EOIR* Configuration window.

6. Select the sensor in the Object Browser and click *EOIR* Sensor Scene (🔧) to generate an image that represents the sensor output. Take a moment to view the new sensor scene.

Refreshing the custom graph

1. Refresh the graph. You will see a graph similar to the one below:



This graph shows the signal large relative to the noise by up to a factor of 210 to 1. The degradation is due to the atmosphere effects. Keep the graph open.

Regenerating the EOIR sensor scene

1. Select the sensor in the Object Browser.
2. Click *EOIR* Sensor Scene (🔧) to generate an image that represents the sensor output.
3. Right-click the sensor scene and select Details....
4. Set the Animation Time Step to 0.5 and animate the scenario. You will see only one bright dot that represents the satellite moving against a dark background. After the satellite moves out of the scene, the stars will return to the image.
5. Close the *EOIR* Scene Visual Details properties page and the *EOIR* Sensor Scene window.

Turning off the Simple Atmosphere model

Now that you have seen the effects the atmosphere has on your data, turn the atmosphere off.

1. Open the Target Configuration () box.
 2. Click **Atmosphere and Textures** to reopen the *EOIR* Atmosphere properties
 3. Set the Atmosphere Mode back to Atmosphere Off for the remainder of this tutorial and return to the *EOIR* Configuration window.
 4. Click **OK** to close the *EOIR* Configuration window
-

Redefining the EOIR Properties

Define the material and shape properties of your satellite and the properties of your EO/IR sensor.

Define the Satellite's Basic EOIR Shape

1. Open the Satellite's properties.
2. Select the Basic - *EOIR* Shape page.
3. Set the following options:

Option	Value
Shape	LEOComm
Body Temperature	Static (default)
Temperature	400 K
Material	Aluminum MLI



Note: This temperature is steady and uniform over the body and corresponds to its external temperature in sunlight.

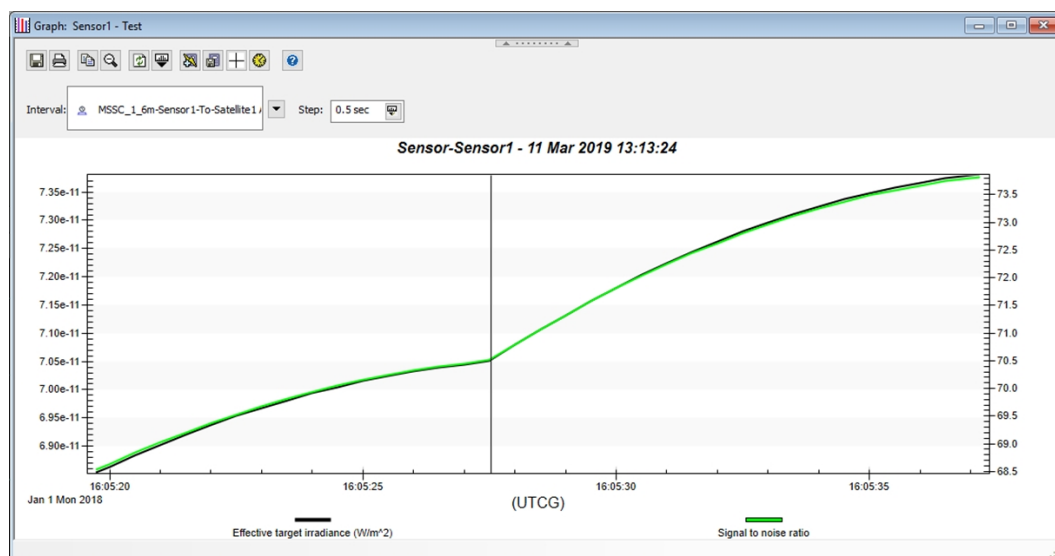
4. Click **OK** .

Regenerating the EOIR Sensor Scene

1. Select the sensor in the Object Browser and click *EOIR Sensor Scene* () to generate an image that represents the sensor output.
 2. Right-click on the sensor scene and select Details....
 3. In general, a satellite will be dimmer than the brightest stars in the FOV. To make that satellite easier to see, slide the Brightness slider all of the way to the right.
 4. Click **Apply** , more objects in the scene will be visible, but which one is the satellite?
 5. Set the Animation Time Step to 0.5 and animate the scenario.
 6. Click around the scene to view information on each pixel. Click one of the stars and the target satellite to get more details on those objects.
 7. Close the *EOIR Scene Visual Details* properties page and the *EOIR Sensor Scene* window.
-

Refreshing the custom graph

1. Bring the Graph window to the front.
2. Set the Step value to 0.5 seconds.
3. Refresh the graph.
You will see a graph similar to the one below:



The curve is smooth, showing a single minimum rate in effective 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 Satellite's properties.
2. Select the Basic - Attitude page.
3. Set the following options:

Option	Value
Type	PrecessingSpin
Body Spin Axis	Type: Cartesian (X,Y,Z) = 0,1,0
Precession - Rate	30 revs/min
Spin - Rate	30 revs/min

4. Click **OK**.
5. Zoom To the satellite.
6. Set the Animation Time Step to 0.1 seconds.
7. Animate the scenario to view the satellite's spinning orbit in the 3D Graphics window.

Refreshing the graph

1. Bring the Graph window to the front.
2. Change the scaling of the Y axis:
 - a. Double-click the Effective Target Irradiance title.
 - b. Select the Axis tab.

c. Select Min/Max in the Y-Axis (default).

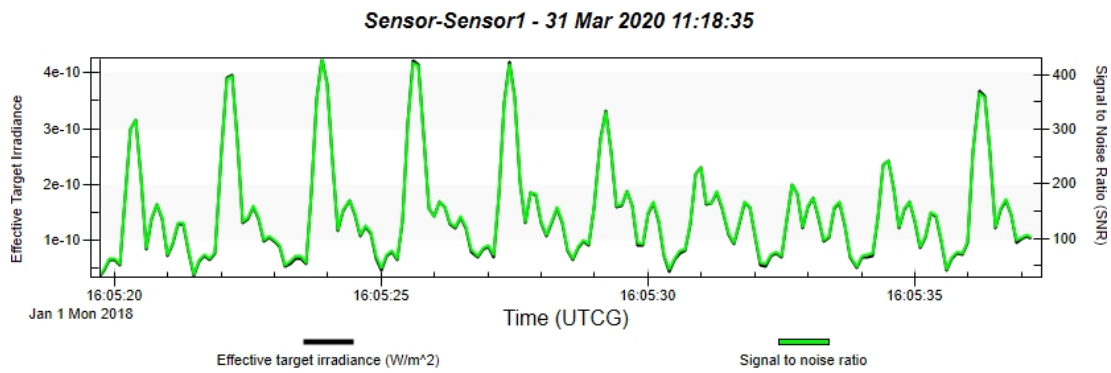
d. Enter the following values:

Option	Value
Min	0
Max	3e-010

e. Click **OK** .

3. Set the Step to 0.1 sec.

4. Refresh the custom Irradiance and SNR graph. You will see a graph similar to the one below; your graph does not need to match exactly:



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 unsmooth plot.

Further study

To become more familiar with all the capabilities of *EOIR*, continue to experiment with changing *EOIR* properties and creating scene settings.

For example, use the ModTran Atmosphere Model to define the atmosphere. To access the model, go to the Target Configuration () box and click Atmosphere and Textures.

Become Level 2: STK Master Certified

Now that you have completed Comprehensive training, you are well-prepared to complete the STK Level 2 Certification test. The STK Master Certification is the second level of certification and validates your ability to perform more advanced STK modeling and analysis through use of STK's *Analysis Workbench*, *Astrogator*, *Aviator*, *Communications*, *Coverage*, and *Radar* capabilities.

What's in the Test?

The Level 2 - STK Master Certification test consists of four scenario development exercises and one bonus exercise. There are multiple choice questions for each exercise. You have 14 days from registration date to complete Master Certification. The following objectives are tested:

- Model 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
- Analyze Your Systems - Access Tool, Report & Graph Manager, Custom Reports, *Coverage*, Figure of Merit, *Analysis Workbench* (Vector Geometry Tool)
- Extend STK - Connect commands to model objects, enable constraints, and generate reports

If you pass your STK Master Certification, you will receive a personalized Level 2 STK Master Certification certificate and a custom L2 STK Master Certification pin. Register now to take the Level 2: STK Master certification test (<http://www.agi.com/training/#cert>).