PhotoMesh™
Version 7.8

User Guide

Skyline Software Systems Inc.

Technical support   support@skylinesoft.com

General information   info@skylinesoft.com

Contact Skyline on the Internet at   http://www.skylinesoft.com

Table of Contents

CHAPTER 1   INTRODUCTION

About PhotoMesh

Skyline's PhotoMesh fully automates the generation of high-resolution, textured, 3D mesh models from standard 2D photographs, Lidar and Full Motion Video (FMV), offering a significant reduction in cost and time compared to traditional modeling methods. PhotoMesh's breakthrough technology is based on the highest-performance photogrammetry, computer vision, and computational geometry algorithms. Combining any number of photographs, in a wide range of formats and resolutions, PhotoMesh generates highly-detailed 3D models that can be viewed and queried using TerraExplorer or other 3D and GIS products.

PhotoMesh employs elaborate tiling mechanisms to efficiently handle projects with even hundreds of thousands of photos. Running on standard hardware, PhotoMesh can also exploit computer clusters and cloud computing to run a single project simultaneously on hundreds of fuser machines. Sharing the workload involved in generating PhotoMesh output vastly accelerates all steps of the build.

PhotoMesh Components

The PhotoMesh application is composed of three components:

- **PhotoMesh Editor** – The main PhotoMesh working area for various workflow actions such as preparing, submitting for processing, monitoring, and reviewing the build.

- **PhotoMesh Fuser** – Worker component of PhotoMesh which allows you to share the workload between several computers on the same network. The master computer, equipped with PhotoMesh fuser, utilizes the computing power of the network's client computers by connecting to PhotoMesh fusers on the client machines.

  Note: Fuser pools can be used to share fusers between several PhotoMesh master computers in your organization. PhotoMesh can also work with Amazon Web Services (AWS), adding and terminating virtual fuser machines, based on resource needs.

- **PhotoMesh Build Manager** – Manager application that is responsible for managing the build process and PhotoMesh fusers. PhotoMesh Build Manager keeps track of all build tasks and fusers, distributing tasks among the fusers and assigning new tasks as fusers complete their tasks and become available. During the build process, PhotoMesh Build Manager monitors the build and provides the user with detailed information about the build's progress.

PhotoMesh Resources

Resources that provide information about the application and about your application license are easily accessible from the PhotoMesh button .

To open PhotoMesh resources:

- Click the **PhotoMesh** button, point to **Resources** and then click one of the following resources:

Resource	Description
PhotoMesh Help (F1)	Provides access to PhotoMesh Help files.
License Manager	Opens the License Manager tool.

PhotoMesh API	Provides access to interactive documentation of the PhotoMesh REST API to automate the production phases, e.g., load photo files or list, set AT and reconstruction areas, begin and manage a build, and create new build versions.
PhotoMesh Start Page	Opens the Start page from which you can create a new project, open an existing one, open the automation dashboard, and more.
About PhotoMesh	Access basic application information, including version number, contact information, and license information.

SkylineGlobe Product Suite

The SkylineGlobe software suite sets the standard for 3D desktop and web-based applications, enabling an enterprise to build and efficiently publish realistic 3D environments that can be explored, queried, analyzed and edited by desktop, web, and mobile clients. The suite includes:

- **Editors** – PhotoMesh, TerraBuilder, and TerraExplorer Pro

- **Cloud Services** – Streaming, Storing, and Cataloging

- **Clients** – Desktop, Mobile, and Web applications
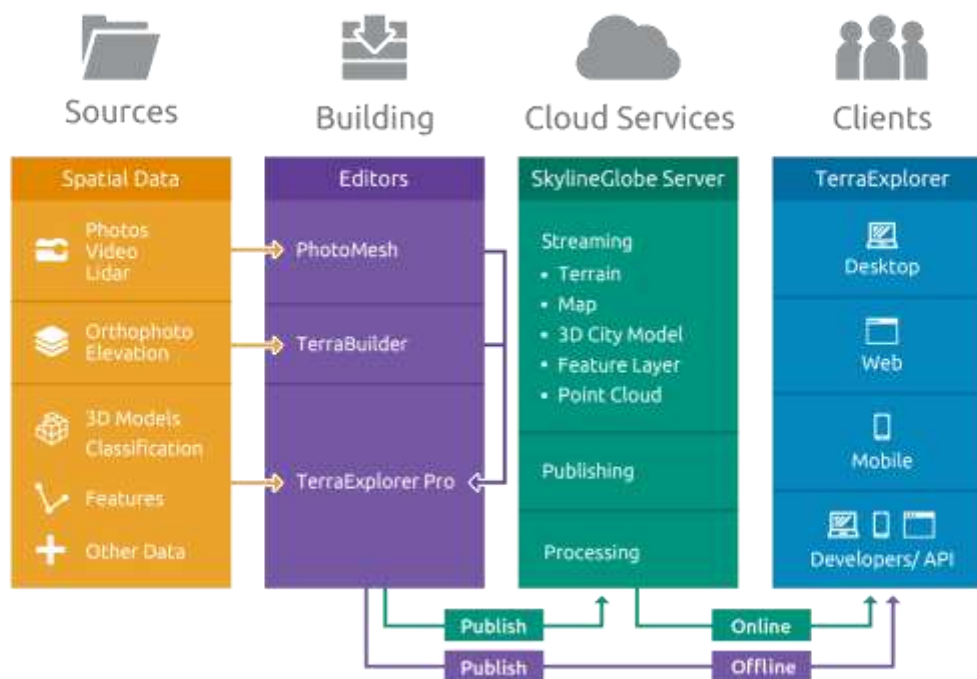
SkylineGlobe Product Suite

The SkylineGlobe Enterprise solution is scalable, with licenses to support from ten to tens of thousands of concurrent users.

Whether deploying a desktop, mobile or web-based application, SkylineGlobe Enterprise provides a full Application Programming Interface (API) allowing you to customize it according to your requirements.

Because SkylineGlobe Enterprise is based on open OGC standards such as WFS and WMS, it can operate as a seamless 3D interface with other existing, legacy systems within the organization.

"On the fly" data fusion from disparate and distributed sources without data pre-processing allows you to keep your 3D environment current and relevant.

Skyline's range of products allows users to design an implementation customized to meet their requirements. Deployment options include the ability to work in a networked or disconnected (off-line) mode and make content available to the public or keep it restricted to secure networks and authorized users. Highly efficient streaming server technology supports large enterprise deployments with minimal hardware requirements.

Editors

Skyline offers powerful applications for generating, editing and publishing 3D terrain and urban model databases from a wide variety of data types.

TerraBuilder

TerraBuilder is a 3D terrain database creator with professional-grade tools for manipulating and merging orthophotos, DEM and DSM raster files of different sizes and resolutions. The resulting photo-realistic, geographically accurate terrain database (MPT) can be made available to local TerraExplorer clients or published to remote clients using the SkylineGlobe Server.

PhotoMesh

Skyline's breakthrough PhotoMesh application fully automates the building of high-resolution, textured, 3D mesh models from oblique and nadir photographs and Lidar captured from street view, drones, UAV and airplanes. PhotoMesh results can be generated in standard 3D model (3DML, OSGB, DAE, OBJ), raster (Orthophoto, DSM, DTM) and point cloud (LAS) formats.

TerraExplorer Pro

TerraExplorer Pro is a powerful GIS tool for merging unlimited, geo-referenced 3D terrain, 3D mesh, Lidar, BIM, raster and feature layers into a 3D high-resolution, geospatial environment. The project and all its individual layers can be directly uploaded and published to SkylineGlobe cloud server from TerraExplorer. Through a single publishing operation, the project and its individual layers are made available to all TerraExplorer clients: Desktop, Mobile, and TE for Web, as well as standard geospatial clients. TerraExplorer Pro also provides powerful tools and a high resolution 3D environment in which to view, query, analyze and present geospatial data.

Cloud Services

SkylineGlobe Server (SGS) is a private cloud solution that provides a comprehensive set of web services for publishing, storing, managing and streaming 3D spatial data. SkylineGlobe Server provides streaming services for all your spatial data types including terrain (MPT/TBP), map (raster-WMS/WMTS), feature (WFS/WFS-T), 3D Mesh (3DML, OGC 3D Tiles, I3S/SLPK), and point cloud (CPT, OGC 3D Tiles). All data on your server is cataloged and referenced so clients can easily find the specific published layer required. Your data is kept safe by multiple security layers, including restriction of each user group to predefined data folders and a robust user authentication mechanism.

The server's built-in complete user access control system allows easy management of users, groups, and administrative roles controlling the server-side storage and client-side view/edit permissions.

SkylineGlobe Server is compliant with the following OpenGIS Implementation Specifications:

- OpenGIS® Web Feature Service (WFS) Implementation Specification

- OpenGIS® Web Feature Service Transactions (WFS-T) Implementation Specification

- OpenGIS® Web Map Service (WMS) Implementation Specification

- OpenGIS® Web Map Tiling Service (WMTS) Implementation Specification
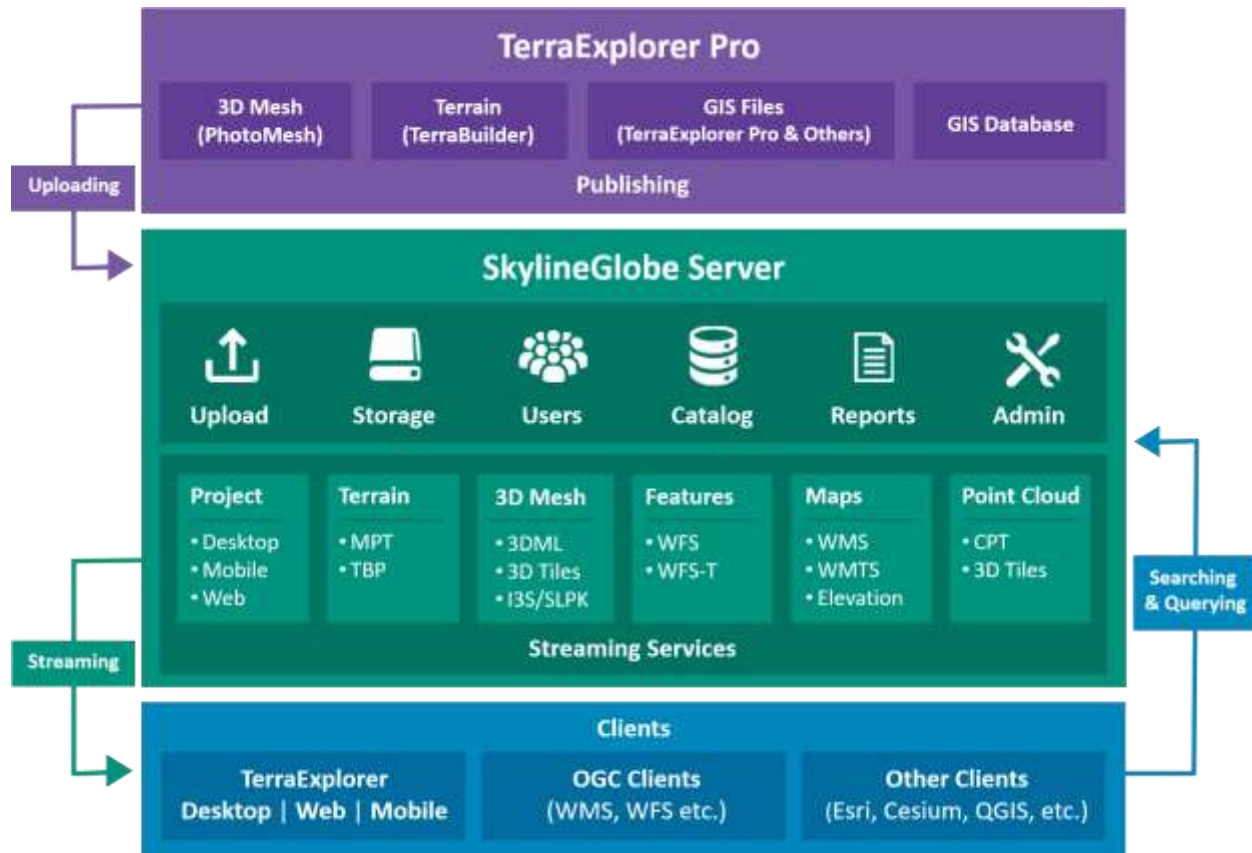
SkylineGlobe Terrain Service

Powerful terrain service technology for streaming terrain data stored either in Skyline optimized terrain databases (MPT) or in terrain databases defined by the TerraBuilder project file (TBP) to remote TerraExplorer clients and any application that reads the standard OGC WMS/WMTS protocol.

SkylineGlobe Feature Service

Powerful feature service technology for streaming point, polyline, and polygon feature layers from files and from the Oracle, SQL server, PostGIS and ArcSDE databases to remote TerraExplorer clients and any application that reads the standard OGC WFS protocol. The Feature Service can stream most TerraExplorer supported feature layer formats. It also supports advanced spatial and attribute queries from TerraExplorer and other WFS clients. With SGS's support for WFS-T (Web Feature Service-Transactional), remote clients can edit (create, delete, and update features) the feature layer and save changes to the data source.

SkylineGlobe Maps Service

Powerful maps service technology for streaming most TerraExplorer supported raster layer formats to remote TerraExplorer clients and any application that reads the standard OGC WMS/WMTS protocol. The layers are streamed as WMS/WMTS tiles in JPEG and PNG formats.

SkylineGlobe 3D Mesh Service

Powerful 3D city model service technology for streaming 3D Mesh Layer (3DML) databases from 3DML files to remote TerraExplorer clients. 3D mesh layers (3DML) are high-resolution, textured, 3D mesh models that have been compressed and stream-optimized to accelerate loading speed and efficiently handle memory. The 3D Mesh Service also exposes the data as 3D tiles for TerraExplorer for Web and Cesium based clients.

SkylineGlobe Point Cloud Service

Powerful point cloud service technology for streaming multi-resolution and stream-optimized point cloud data from CPT files to remote TerraExplorer clients. The Point Cloud Service also exposes the data as 3D tiles for TerraExplorer for Web and Cesium based clients.

Clients

Skyline offers a range of 3D visualization clients that enable users to interact with, edit and annotate a geo-referenced 3D environment. Users can customize the database with data from a network, local drive or the Internet. Overlaying data specific to the user's requirements onto a 3D map creates a targeted, interactive picture that can meet the needs of a diverse user base and the specific requirements of each individual user. Skyline's client applications can be used in a local desktop environment, accessing terrain and content data directly, or in a network environment via the SkylineGlobe server.

TerraExplorer Desktop

A cutting-edge 3D GIS desktop viewer and creator that provides powerful tools and a high resolution 3D environment in which to view, query, analyze and present geospatial data.
With TerraExplorer's robust and extensive capabilities and ever-increasing interoperability, stunningly realistic 3D visualizations can be created by overlaying the terrain with unlimited data layers, 3D models, virtual objects and more.

TerraExplorer for Web

TerraExplorer for Web is a lightweight 3D GIS viewer that enables you to view high-resolution, stunningly realistic SkylineGlobe 3D content in a web browser, without any plug-ins. The employment of HTML5/WebGL standards allows TerraExplorer for Web to provide support for multiple platforms and browsers. Built on the powerful Cesium open source library, TerraExplorer for Web further extends the core functionality with additional capabilities and enhanced performance.

TerraExplorer for Mobile

TerraExplorer for Mobile is a 3D GIS viewer for Android and Apple iOS devices, based on TerraExplorer for Web, that enables you to view high-resolution SkylineGlobe 3D content. The TerraExplorer for Mobile app can display most of your spatial databases including 3D city layers and feature layers, and it provides powerful tools to query, analyze and present your data on the go. TerraExplorer for Mobile seamlessly accesses online data from Skyline's SkylineGlobe server and other OGC-compliant servers, and quickly loads online TerraExplorer projects.

CHAPTER 2   GETTING STARTED

Software and Hardware Requirements

Operating System	Windows® 8 / 10 Windows® Server 2012 R2 / 2016 / 2019 – 64bit required. Ubuntu 18.04 and Debian 10.0 – 64-bit required (Only for PhotoMesh **fusers**).
System Memory	16 GB RAM (32 GB recommended). **Note**: RAM requirements are mostly affected by the number of photos in the AT tile and the reconstruction tile size. **See** "Setting AT Parameters" and "Setting Reconstruction Parameters" in the "Building" chapter for more information.
Video Card	1 GB of video memory (2 GB or more recommended). Pixel and vertex shader v3.0.
Processor	4 cores (8 cores recommended). PhotoMesh works best in a multi-core environment and can utilize multiple CPU's and hyper-threaded processors. In the latest PhotoMesh performance tests, using a single computer, 16 GB RAM and 8 cores, output was between 10-15 gigapixels/day, depending on the machine and resource availability.
Additional Software	.Net Framework 4.6.1 required.

License Mechanism for PhotoMesh

PhotoMesh is protected by a licensing mechanism that controls its operation, license term, and available modules. A license file issued by Skyline determines the licensing mechanism used.

The following licensing mechanisms are available for PhotoMesh:

- Local Hardware Protection (USB Key)

- Local Hardware Protection (HASP Key)

- Workstation Dependent Software License

- Skyline Floating License Server

- HASP Floating License

Local Hardware Protection (USB Key)

A USB key is a physical hardware key delivered with the PhotoMesh software package that plugs into the USB port of the computer – to allow the operation of the product. Verify that the USB key is plugged into the USB port before starting PhotoMesh.

Local Hardware Protection (HASP Key)

In some cases, Gemalto's HASP key solution is used instead of a USB key. The HASP key is a physical hardware key delivered with the PhotoMesh software package that plugs into the USB port of the computer – to allow the operation of the product. Verify that the HASP key is plugged into the USB port before starting PhotoMesh. To use a HASP key, the necessary driver must be downloaded and installed on your computer. **See** "Manual Installation of the HASP Driver" in this chapter for information.

Workstation Dependent Software License

Another type of license is the workstation dependent software license. This license does not require any hardware protection but is limited to a specific computer. Before working with the workstation dependent software license, you must first register to get a license locked to your computer. The license cannot be transferred without repeating the registration process.

Skyline Floating License Server

The license information can also be obtained using a floating license server which enables a number of licenses to be shared among a larger number of users. When an authorized user wants to run PhotoMesh, the user requests a license from the license server, and if one is available, the license server allows PhotoMesh to run. When the user finishes running PhotoMesh, the license is returned to the license server for use by other users. **See** "Configuring PhotoMesh to Use the Skyline Floating License Server (SFLS)" in this chapter for information. The floating license server also supports the PhotoMesh GX mechanism, which allows you to pre-purchase a bank of gigapixel credits, load them to your local Skyline Floating License Server (SFLS), and automatically apply them towards different PhotoMesh projects to allow building of all output types without watermarks. **See** "Applying Gigapixel Credits" in the "Building" chapter for more information.

HASP Floating License

In some cases, Gemalto's NetHASP solution is used instead of Skyline's Floating License Server. The NetHASP key is a physical hardware key delivered with the PhotoMesh software package that plugs into the USB port of the server computer along with installation of the NetHASP License Manager software. It allows a predetermined number of concurrent users on the same network to use PhotoMesh, according to the license level defined by Skyline, without an individual hardware key on each of their computers.

License Manager

The License Manager allows the user to view and update the current license level.

To activate the License Manager:

▪ Click the **PhotoMesh** ⊗ button, point to **Resources**, and then click **License Manager**. The License Manager window is displayed.

PhotoMesh License Options

The following PhotoMesh license options are available:

▪ **PhotoMesh Standard** – Pay per fuser. No gigapixel limit.

▪ **PhotoMesh GX** – Pay per gigapixel in your project. Unlimited managers/fusers. This option is generally implemented with Skyline Floating License Server (SFLS), or using an online gigapixel credit form or request file. Gigapixel credits are loaded to SFLS, and then applied to a project's photos when building the project. See the SFLS Setup Guide for more information.

▪ **PhotoMesh Evaluation** – 1 manager/ 1 fuser. Only 3DML output can be produced, displaying a watermark.

Installing PhotoMesh

The PhotoMesh application installation includes fuser installation. A standalone fuser installation is also available. **See** "Installing PhotoMesh Fusers" in this chapter for information.

To install PhotoMesh:

Note: On some operating systems, administrative privileges may be required.

1. If you have an old version of PhotoMesh running, close it before continuing with the installation.
2. Place the PhotoMesh license file (SLPhotoMesh.lic) next to the PhotoMeshSetup.msi file in the installation folder.
3. On your computer, browse to the **Photomesh.msi** file and double-click. The Welcome to PhotoMesh Setup dialog box is displayed.
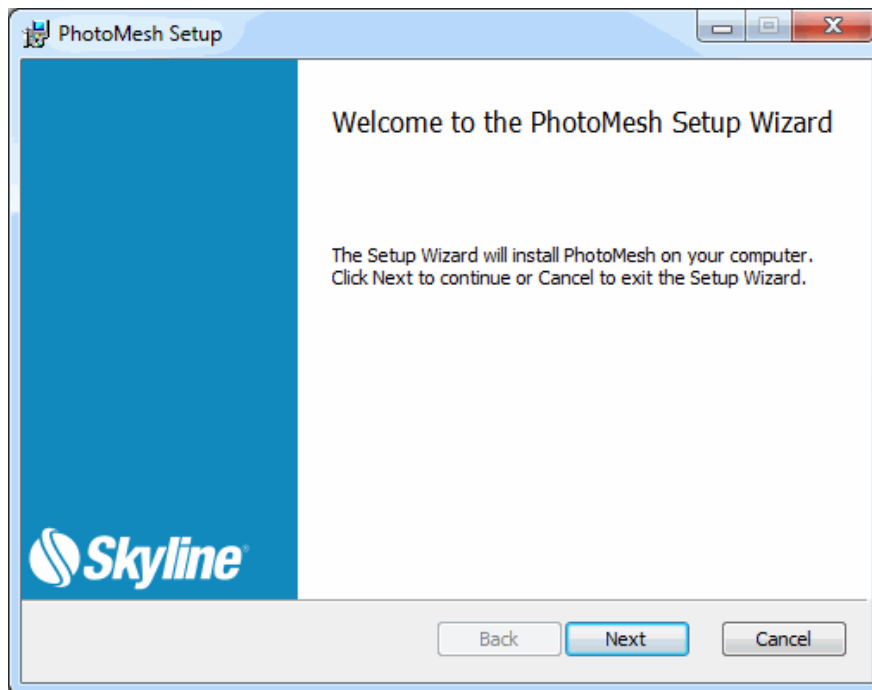
Welcome to PhotoMesh Setup

4. Click **Next**. The End-User License Agreement is displayed.
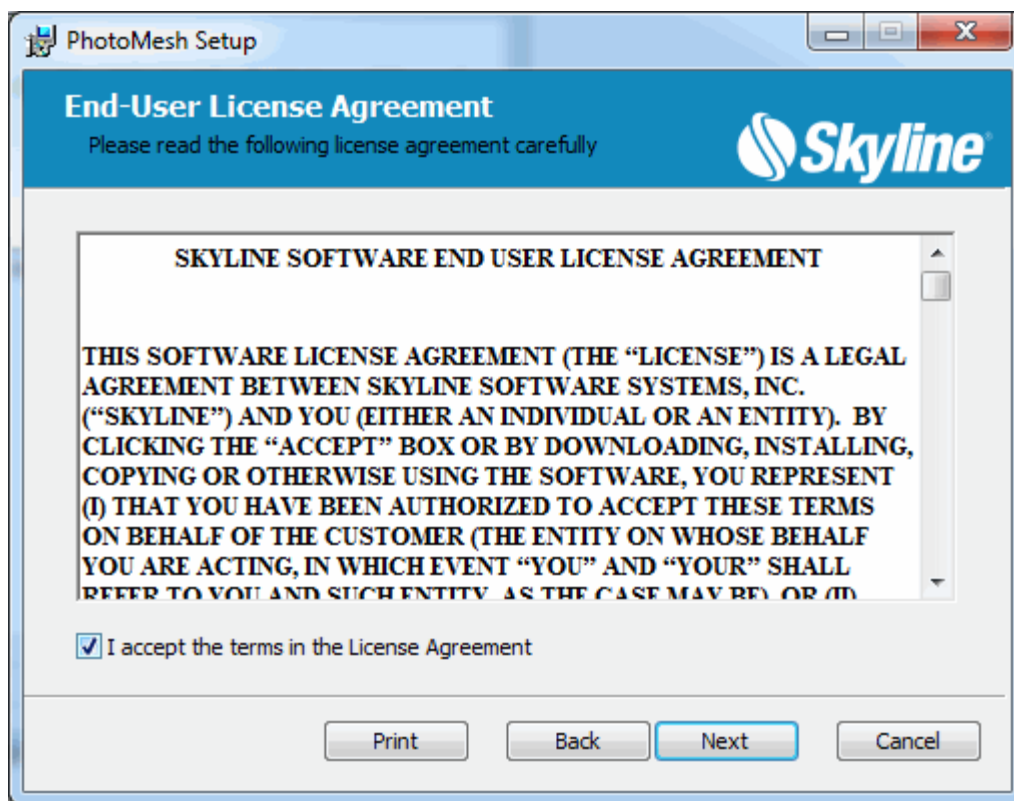
License Agreement

5. Select the check box to agree to the license agreement terms and then click **Next** to continue the installation.
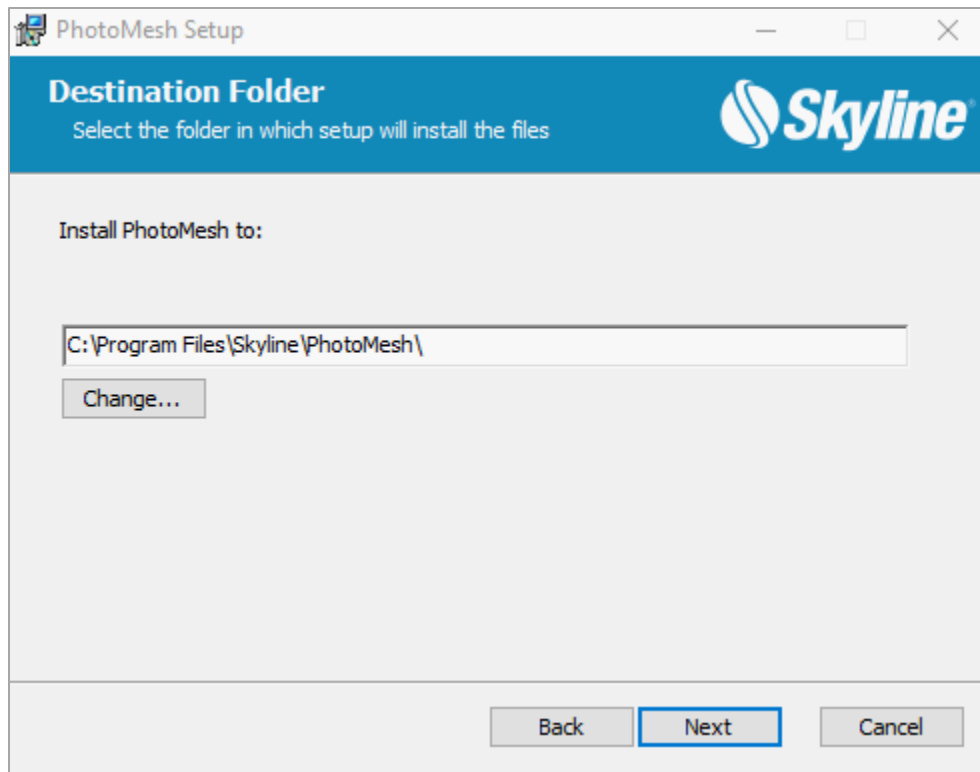
Destination Folder

6. Select the folder in which setup should install the files, and then click **Next**. The Ready to Install PhotoMesh dialog box is displayed.
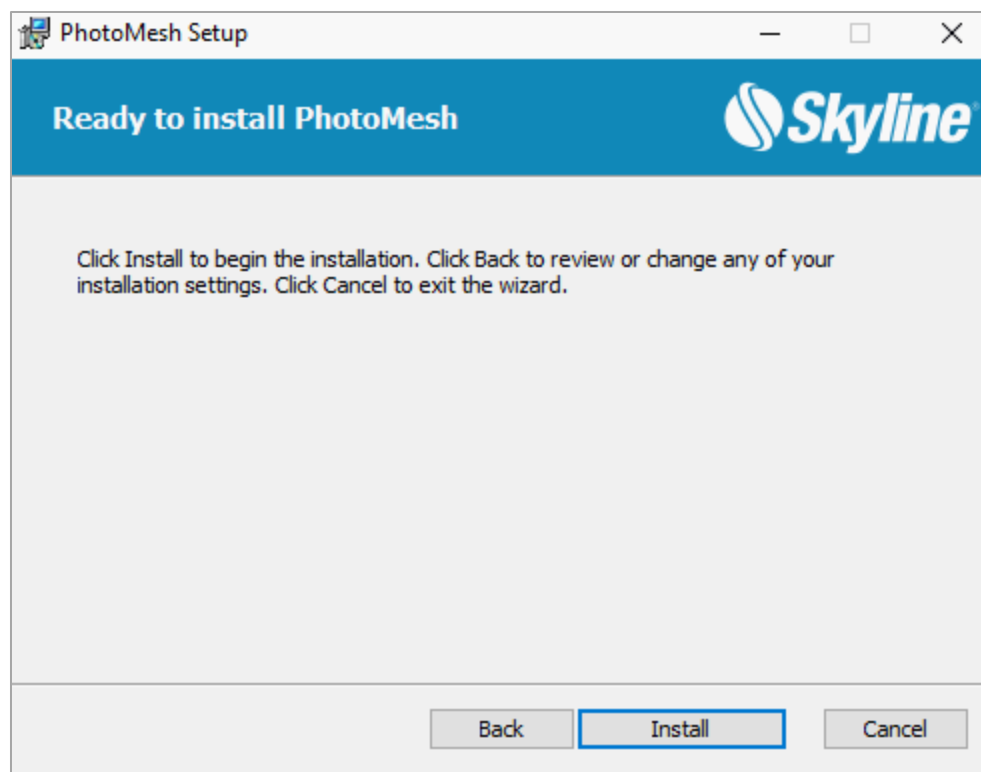
Ready to Install PhotoMesh Dialog Box

7. Click **Install**. A dialog box is displayed when the installation process is complete.
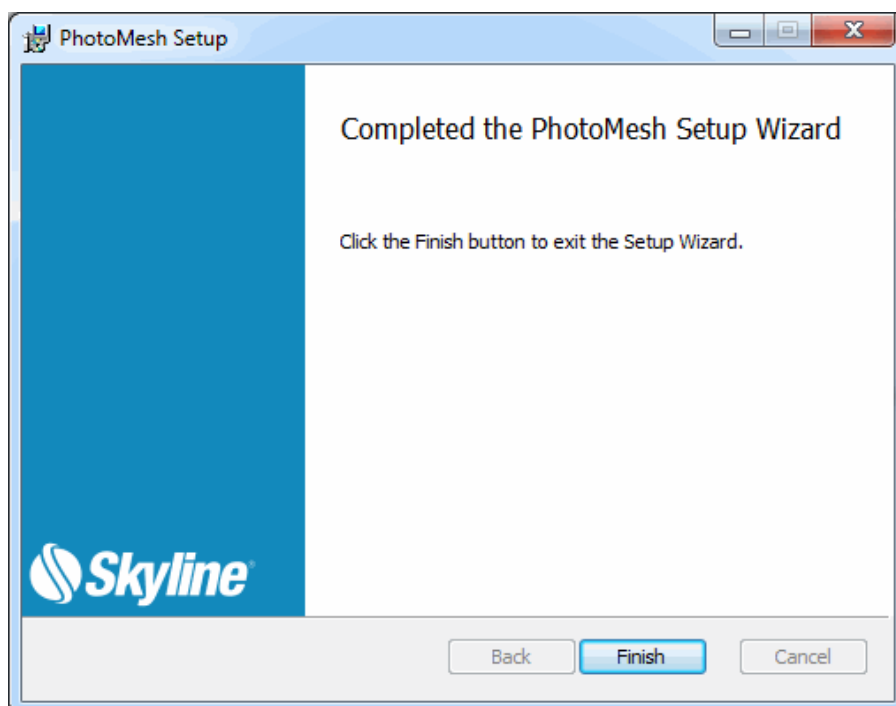
PhotoMesh Setup Complete

8. Click **Finish** to exit the Setup Wizard.

Manual Installation of the HASP Driver

If your software was delivered with a HASP license key, you can manually install the required HASP driver.

To install the HASP driver:

1. Download the HASP driver zip file: Sentinel_LDK_Run-time_setup.
2. Unzip the file to a temporary directory, and run the HASPUserSetup.exe file.
3. Follow the installation wizard instructions to complete the installation.

Configuring PhotoMesh to Use the Skyline Floating License Server (SFLS)

Each machine that is sharing the floating license for PhotoMesh must be configured with the address of the Floating License Server.

Note: The Skyline Floating License should be installed on the server before configuring it on each of the machines that is sharing the license.

To configure PhotoMesh to the SFLS address:

1. Open the License Manager. **See** "License Manager" in this chapter for information.
2. If the Skyline Floating License received from Skyline was not used in the installation (i.e., placed next to the **PhotoMesh.msi** file in the installation folder), from the **File** menu, select **Load License File**, and browse to the license file.
3. From the **File** menu, select **Set Floating License Server** to set the floating license server URL, e.g., http://<server_name>/SFLS.

Uninstalling PhotoMesh

The Uninstall feature lets you quickly remove all folders and files that you installed to your hard disk during the installation procedure.

To uninstall PhotoMesh (including PhotoMesh fuser):

1. Exit the PhotoMesh and PhotoMesh fuser applications.
2. From the Windows **Start** menu, click **All Programs**, then click the **Skyline PhotoMesh** folder, and select **Uninstall**. The Windows Installer dialog box is displayed.
3. Click **Yes** to uninstall.

Installing PhotoMesh Fusers

PhotoMesh fuser can be installed as a standalone installation on any client computer on the same network. The number of fusers that can run simultaneously is determined according to the PhotoMesh license. **See** "PhotoMesh License Options" in the "Getting Started" chapter. **See** the "Fusers" chapter for information on installing fusers for a fusers pool and for general fuser information.

To install a PhotoMesh fuser:

1. Log on as a user with administration privileges.

2. On your computer, browse to the PhotoMesh Fuser.msi file and double-click. The Welcome dialog box is displayed.
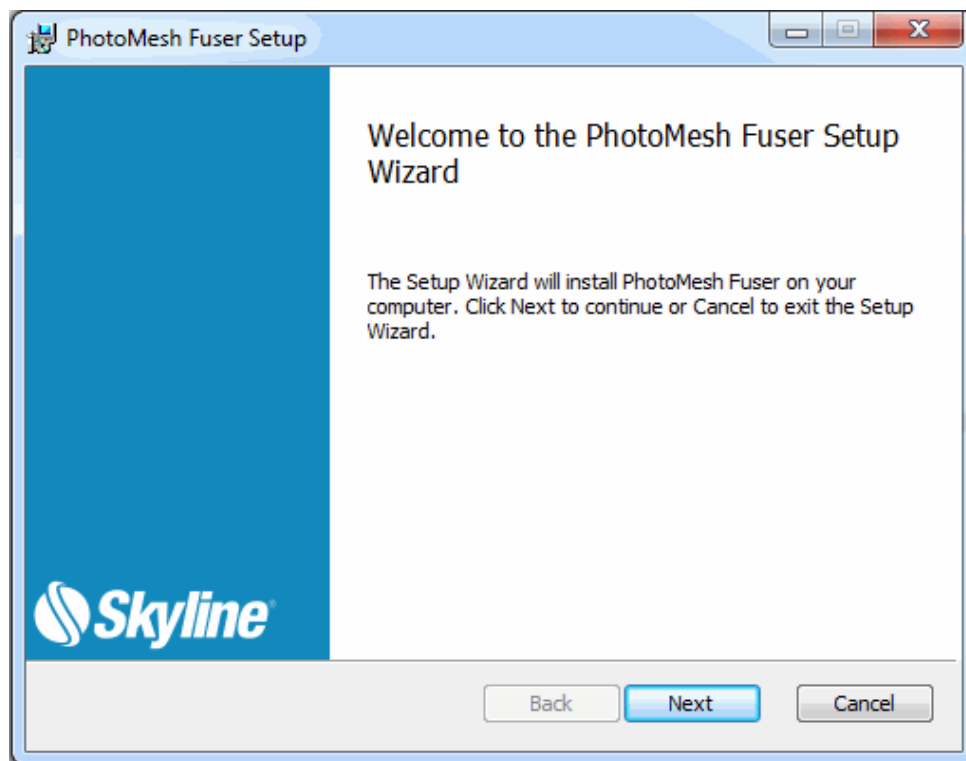
PhotoMesh Fuser Welcome Dialog Box

3. Click **Next** to view the Software License Agreement dialog box. If you agree to the licensing terms, click **Yes** to continue the installation. The Installing PhotoMesh Fuser dialog box is displayed.
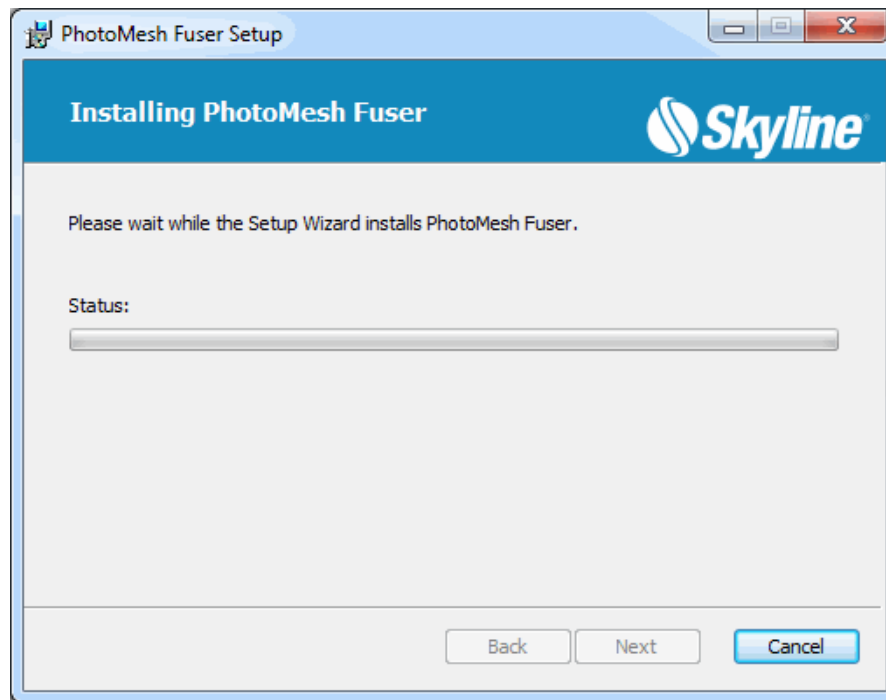
Installing PhotoMesh Fuser Dialog Box

4. Click **Next**. The Completed PhotoMesh Fuser Setup dialog box is displayed.
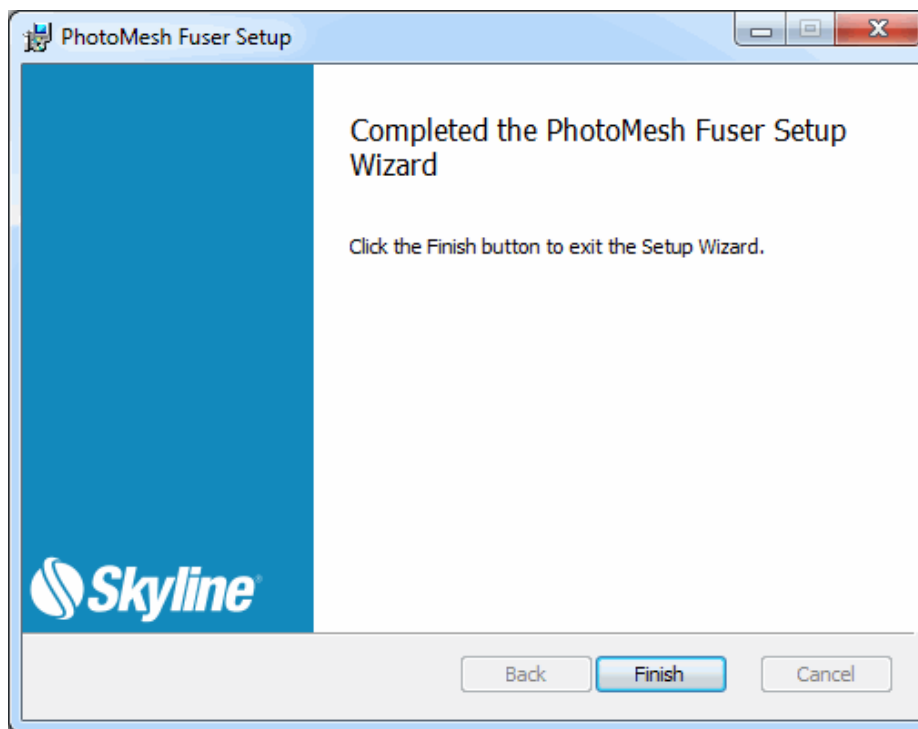
Completed Fuser Setup Dialog Box

5. Click **Finish**.

21

Installing Linux Fusers

To install a Linux fuser:

See: "Customizing the Instance" in the "Getting Started with AWS" chapter for more information.

1. Make sure you have the latest graphics drivers for your graphics card and X Window System configured.

    Note:　　To verify that your graphics card is configured properly, type the following in the terminal window: "glxinfo -B" and see if the right graphics card info is displayed.

2. Install and configure Wine by doing the following:

    a. Copy the installation package received from Skyline, "*wineinst.tar*", to your Linux machines, under "*/home/ubuntu/*".

    b. In terminal window, type the following:

        i.   tar -xvf wineinst.tar

        ii.   sh fuser.sh

        Note:　 This process can take a few minutes to complete.

3. Run PhotoMeshFuser:

    a. In terminal window, mount all the relevant shared drives on the Linux machine. (Working folder, Project, Input data)
        Example:
        mount -t cifs //<IP>/<SharedDrive> /mnt/mount1 -o username=<username>,password=<password>,vers=2.0,uid=ubuntu

    b. In terminal window, mount and hard link Wine DOS devices:
        Example:
        mkdir ~/.wine/dosdevices/unc
        mkdir ~/.wine/dosdevices/unc/<IP>
        ln -s /mnt/mount1 ~/.wine/dosdevices/unc/<IP>/<SharedDrive>

    c. In terminal window, navigate to the PhotoMesh fuser folder and run PhotoMeshFuser.exe with Wine. The fuser files can be on your Linux machine or on a shared drive.
        Example:
        wine photomeshfuser.exe

Uninstalling PhotoMesh Fusers

The uninstall feature lets you quickly remove all folders and files that were installed on the hard disk during the installation procedure.

To uninstall a standalone PhotoMesh fuser:

1. Exit PhotoMesh application.

2. From the Windows **Start** menu, click **All Programs**, then click the **Skyline PhotoMesh Fuser** folder, and select **Uninstall PhotoMesh Fuser**.

3. The Windows Installer dialog box is displayed. Click **Yes** to uninstall.

Starting PhotoMesh

Start PhotoMesh by doing one of the following:

▪   Click **Start** > **Skyline PhotoMesh**> **PhotoMesh**.

▪   Click the **PhotoMesh** icon on your desktop.

- Double-click any .PhotoMeshXML file on your file system.

CHAPTER 3   NEW FEATURES IN RELEASE 7.8

What's New in 7.8

Quality Improvements to Support Higher Quality Close Range Projects

With PhotoMesh 7.8, you can build ultra-realistic, close range 3D models such as towers, bridges, factories and more. The latest PhotoMesh features several enhancements for working with close range, sub-centimeter data collections:

- Modifications of aerotriangulation process to support data collection of close-range objects such as towers and antennas.

- Automatic identification and cleaning of sky background

- Better sharpening and modeling of thin elements such as poles and pipes to produce higher quality, detailed point clouds and mesh models

- Noise reduction around edges of complex models

- Improved model quality across all levels of resolution pyramid

Higher Quality Orthophoto

Quality-optimized image compression produces cleaner and higher quality orthophoto output. New preset offers the following advantages:

- Higher quality texture resolution:
    - Higher compression rate for the photos
    - Higher compression rate for the final ortho
    - Improved quality for ortho texturing

- Sharper building edges

- Reduced noise

New Option to Use Reference Project to Improve Project Alignment

This mechanism automatically geo-references a new project based on the aerotriangulation results of a previously built project, to ensure both projects are completely aligned.

- This tool can be particularly useful for multiple flights over the same area. Ground control points, or high accuracy GPS locations (RTK, PPK) can be used for the first flight and then the Reference Project mechanism can be used to accurately register the other flights, even without GCPs or accurate location information.

- The Reference Project mechanism can ensure perfect horizontal and vertical alignment for visualization and difference comparison tools.

- The Reference Project can also be used with partially overlapping projects. For best success, the same sensor and flight pattern should be used for both projects although the Reference Project mechanism can support different sensors and patterns

Export Undistorted Photos

New option to save the undistorted photos created in the AT process when exporting AT results. During the photo preparation steps, PhotoMesh calculates the photos' distortion parameters (that result from most cameras lens properties) and uses these parameters to undistort the photos so that feature matching and other AT sub-steps can be performed properly.

Performance Improvements

- Close range project memory usage optimized for 16GB machines

- Optimized production time for close range projects

- Overall performance improvements accelerate production

Quality Report API

PhotoMesh provides a REST API for PM Production to automate the production phases, e.g., load photo files or list, set AT and reconstruction areas, begin and manage a build, and create new build versions. PhotoMesh 7.8 features the addition of a new method to the API for generation of a report in JSON format with information about the general project, photo collections, AT results, control points, reference projects and reconstruction results.
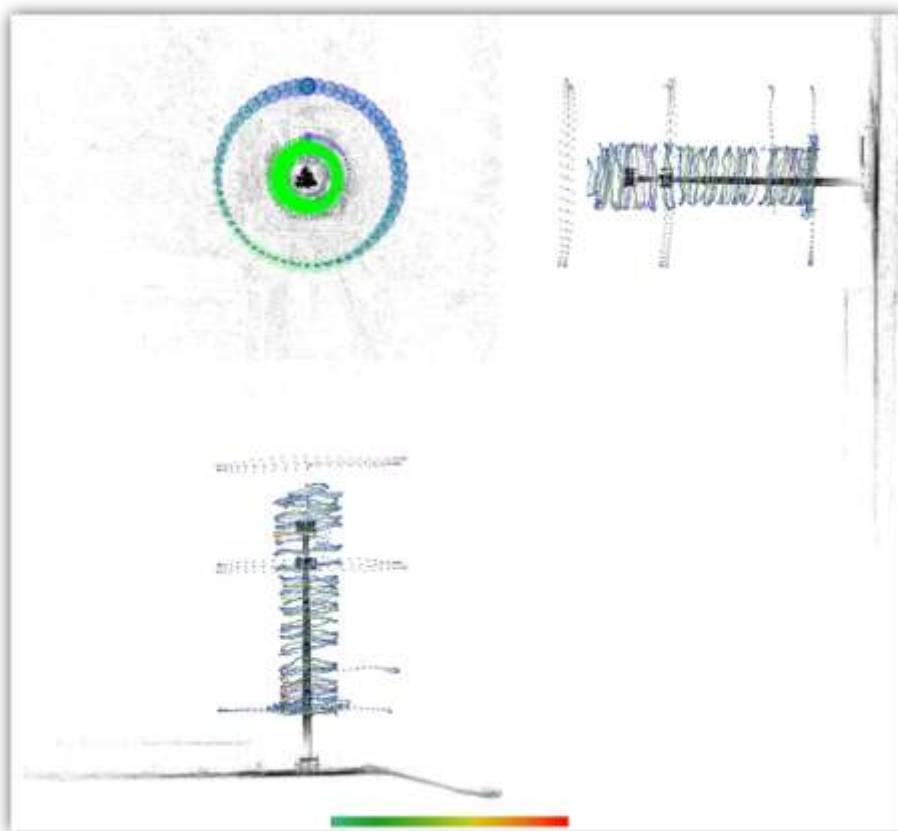
Usability Improvements

- Close range navigation - Improved 3D navigation to support close range and vertical objects

- Relative path – Project can be copied or moved to a new folder or drive location after any build step. It can then be resumed from any step from its new location
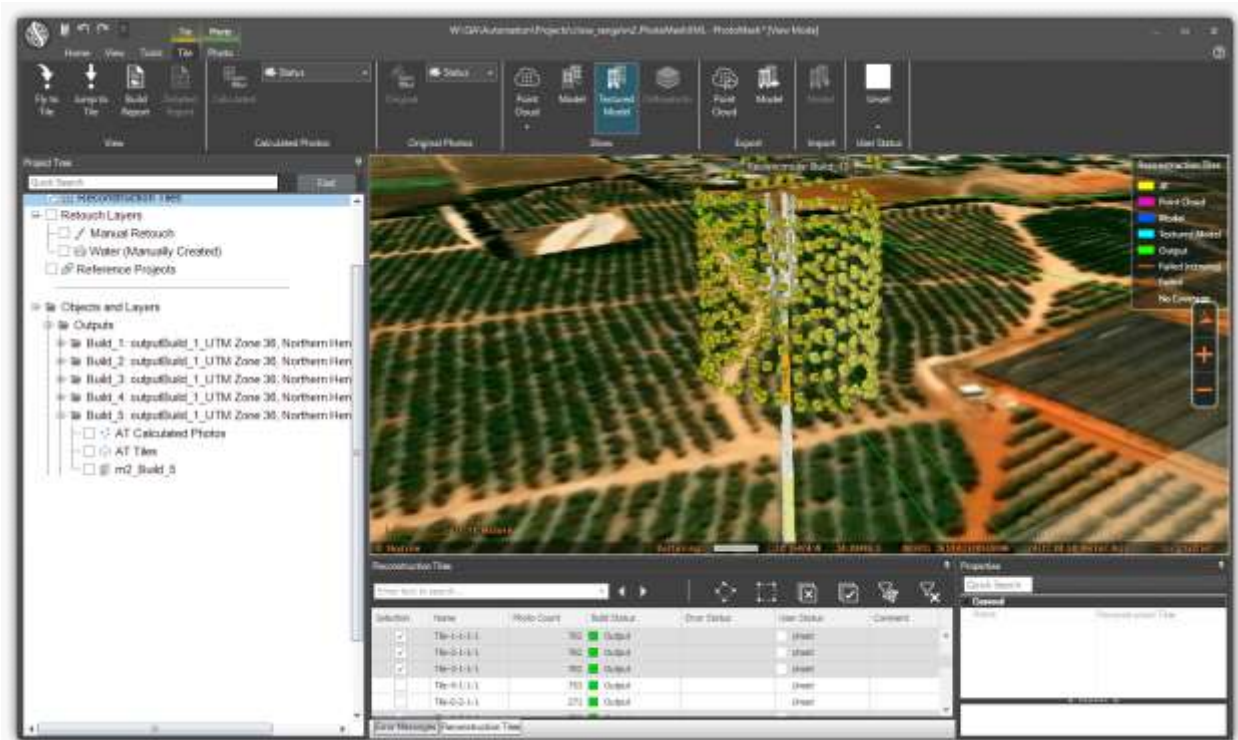
AT/Reconstruction Area Buffer

New options make it easier to set the appropriate area for aerotriangulation, especially in close-range projects. The area of the project on which PhotoMesh should perform aerotriangulation can now be calculated automatically by PhotoMesh based on all the project's photos plus a buffer in meters that is either based on the collections' maximum effective range or a set value that you define.
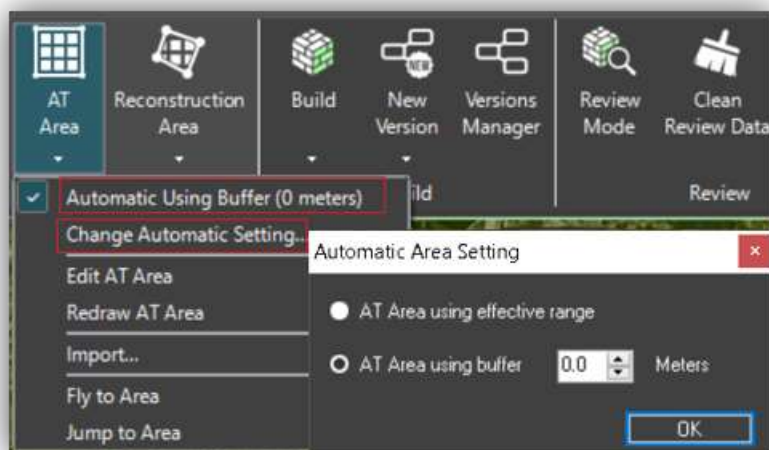
AWS Improvements: Fuser Throttling and Tagging

- Throttling – New Launch Rate setting lets you control the maximum number of fuser instances launched per minute, e.g. if Maximum Instances was set to 200, and the Launch Rate to 50, the Build Manager will spread the launch of the fusers over 4 minutes. This prevents a potential bottleneck that could be caused by a large number of fusers attempting to read the build data at the same time.

- Tagging – New user-defined tags added to fuser properties to help you easily identify a fuser instance in the Amazon EC2 console after it is launched.

Bug Fixes and Stability Improvements

- Resolved issue causing Lidar bounding box to float above point cloud data

- Resolved issue causing orthophoto generation to fail when tile size exceeded 50,000 pixels.

- Resolved issue causing build to fail in projects with more than 4,000 Lidars

- Fixed bug that caused Lidars to disappear from project after changing their coordinate system and then reopening the project

- Fixed bug that resulted in control point editor sometimes opening showing an "Overflow error"

- Corrected issue that caused a build to fail when generating orthophoto output with 'Mesh Vertical Bias' preset

- Failed photos are now included in the calculated lists of the Control Point Editor, with their name marked in red and a tooltip to inform the user that they were not calculated in the AT process.

- Resolved bug relating to control point sampling error variance

- Fixed bug affecting display of table in CSV/TXT Importer

- Fixed bug affecting display of orthophoto on white background

- Corrected bug causing white dots to display on PhotoMesh window when using the shortcut keyboard combination to open a tile's folder

New Features in 7.7

Quality Enhancements

Better handling of close-range projects:

- Photo matching step prioritizes features close to the camera in the bundle adjustment to improve aerotriangulation results in close-range projects (Implemented using "AT foreground object at close range" preset).

- Sharpens thin objects such as poles and wires by cleaning point cloud noise around their edges (Implemented using "Reconstruction ground-based LIDAR and projects with tall structures" preset).

- Cleans sky background from photos looking at the horizon (Implemented using "Reconstruction sky background clean" preset).

- Improved build results in low overlap, nadir projects – Fills in ground gaps in areas with low coverage (e.g., to sharpen building outlines and fill holes in the ground) by incorporating calculated ground surface (Implemented using "Reconstruction fill in ground" and "Reconstruction fill in ground and add walls" presets. Can be used together with the "Reconstruction mesh vertical bias" preset).

- Improved color balancing – Enhances contrast of project photos using a Contrast Limited Adaptive Histogram Equalization (CLAHE) algorithm (Implemented using the "DataPrep local_brightness histogram correction for images (CLAHE)" preset. Other preset options include " DataPrep brightness

histogram correction for images" and "DataPrep color histogram correction for images").

- Optimized mesh and texture quality:
    - Sharper mesh edges and more triangles in areas of complex topology.
    - Fewer triangles in flat areas, which results in cleaner geometry and reduced file size and processing time.
    - Optimized mesh triangle count for each of the model's levels of detail (LOD) produces a more accurate mesh in different viewing distances.
- Better support for multi-band projects:
    - Provides the ability to select one or more bands for orthophoto texturing and any three bands for mesh model processing.
    - Improves color balancing for multi-spectral bands (NIR, thermal) by minimizing color artifacts.
    - Optimizes texturing algorithm performance by restricting texture generation to only bands that are in use.

- Improved DSM quality – Smooths mesh model triangle artifacts when generating DSM raster output.

- Improved control point use in aerotriangulation algorithm – Ground control and tie points are now assigned a stronger weight when performing aerotriangulation, to increase geographical accuracy in projects with unreliable inputted camera positions.

- Improved Lidar textures – Improvements to Lidar color-based textures by applying a color smoothing algorithm.

Expanded Input and Output Support

PhotoMesh 7.7 offers expanded input and output options, including all the following:

Outputs:

- New DSM to LAS tool exports the Digital Surface Model (DSM) output as colored point cloud model in LAS or LAZ format.

- OBJ and DAE output models are now saved with single texture file per tile.

- Enhanced options for orthophoto and Digital Surface Model (DSM) outputs:
    - Ability to define maximum tile size in pixels.
    - Ability to define grid origin.
    - Ability to define tile overlap in pixels.
    - Default coordinate system for all raster outputs changed to the local UTM zone in project area.

- Aerotriangulation and reconstruction tile polygons saved as part of a project's outputs.

- Raster output (orthophotos, DSM & DTM) TIFFs can be saved as LZW lossless compression or as jpg compression.

Inputs

- Fully support photo orientation in yaw/pitch/roll format.

- Support reading photos directly from Amazon S3 buckets and behind HTTP servers (*not supported by the Gigapixel mechanism).

- CSV/Text loader - Photo and photo collection information (e.g., sensor width, focal length, and aerotriangulation data) can now be loaded from a CSV file.

- Phase One – Read photos in IIQ RAW file format that were created by Phase One sensors.

Lidar Improvements

PhotoMesh 7.7 features enhancements to Lidar display and functionality. Lidar data can be integrated with project photos to supplement project data and increase model accuracy.

- Point cloud styling - Set point size, color and visibility based on point cloud attribute information.

- Manage Lidar sources in a sortable, multi-edit list.

- Display the Lidar's trajectory in the 3D Window.

- Lidar files can now be loaded even after aerotriangulation is performed. If reconstruction had already started, all reconstruction tiles that intersect with the new Lidar's bounding box are rebuilt automatically in the next project build.

- Change Lidar coordinate system and automatically reconvert and import these Lidar files in the next project build.
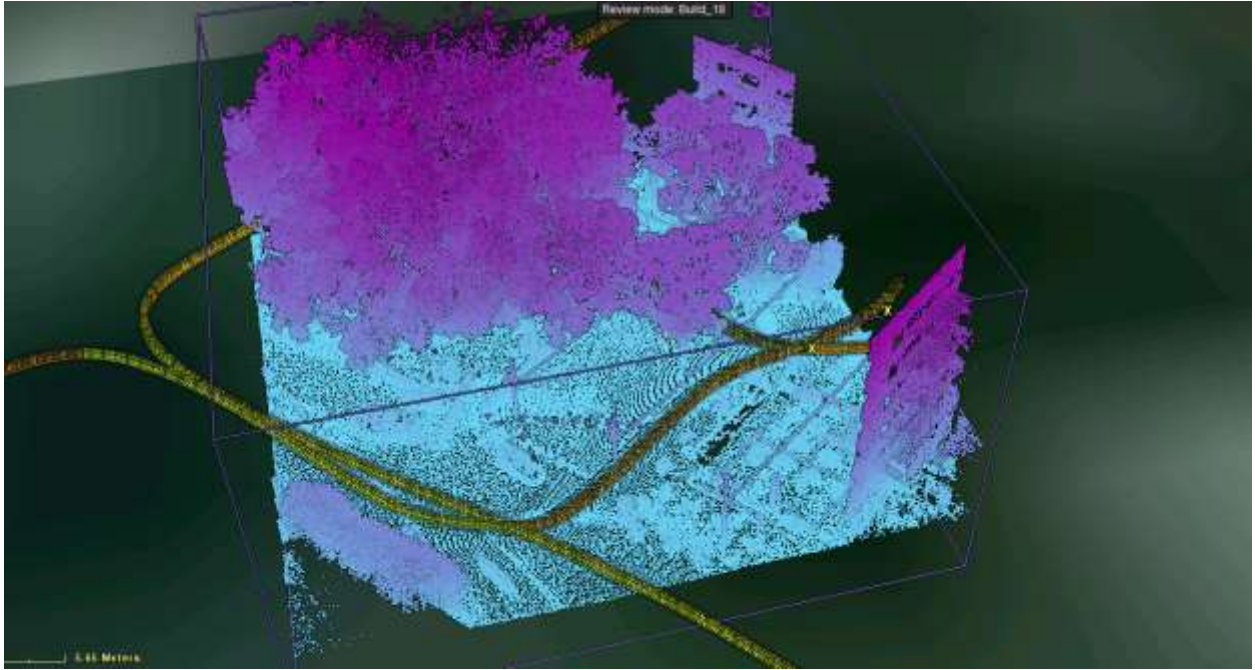
Preset Mechanism

PhotoMesh's build-preset mechanism has been enhanced to allow saving of only specific build settings as a preset, loading of multiple presets, and more. The preset mechanism makes it easy to automatically set the required parameters to generate the optimum output for the project's data as well as to share user-defined build settings between projects.

- Presets now include only changes from the default settings.

- Users can control which step settings are saved (AT/ Reconstruction/ Output/ CS), so only changes to these steps are saved and loaded.

- Multiple presets can be selected for a build, each of which defines settings for a different subset of the build steps. If selected preset files have conflicting properties or flags, the value from the last file on the list is applied.

- New set of presets for different use cases (close range, optimization for special output formats and more).

- GUI improvements include detailed descriptions of the selected presets, selection control to multi-select presets for a project, and powerful management control to load, edit and delete presets.
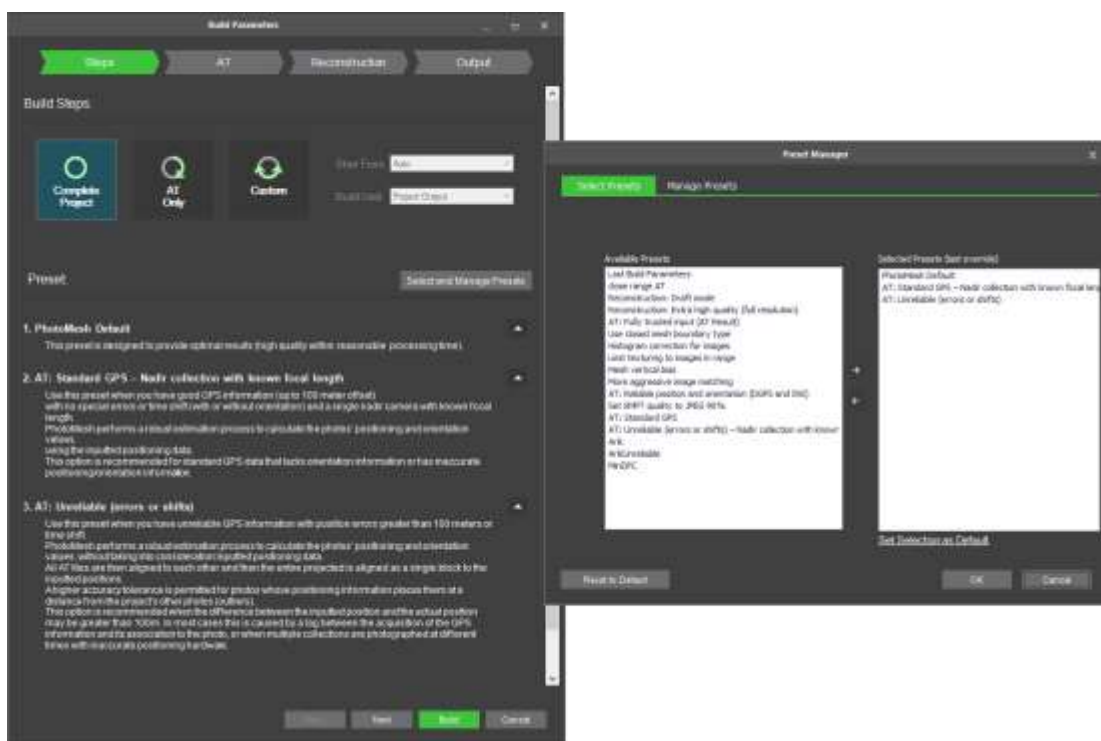
Manual Retouch

Mesh imperfections, such as bumps, irregular surfaces, or floating artifacts, can be flattened, filled, and cleaned up directly in PhotoMesh using the Manual Retouch tool. This tool performs smart selection, taking your rough marking of floating artifacts, wires, or surfaces for flattening, analyzing the pixels, and determining the object it thinks you want within the rough selection. You can then further modify the

selection polygon and adjust the applied buffers. The output of the Manual Retouch tool is a polygonal feature layer with attribute information that defines what retouch action should be performed in each area: clean, flatten, and retexture. When rebuilding the project, PhotoMesh uses the manual retouch layer to add a set of constraints to the reconstruction algorithms.

Project Queue

Improvements to the Project Queue automation mechanism. The Project Queue is a REST API to create a project queue, and then activate, monitor, and manage the queue processing.

- New script step type (python,JS) to analyze and manages the outputs of the queue processing.

- New pause step type to pause a build so the user can inspect the result and add control/tie points and then resume the process.

- Ability to load multiple presets, and support full path name for the preset location.

- Ability to sort the queue.
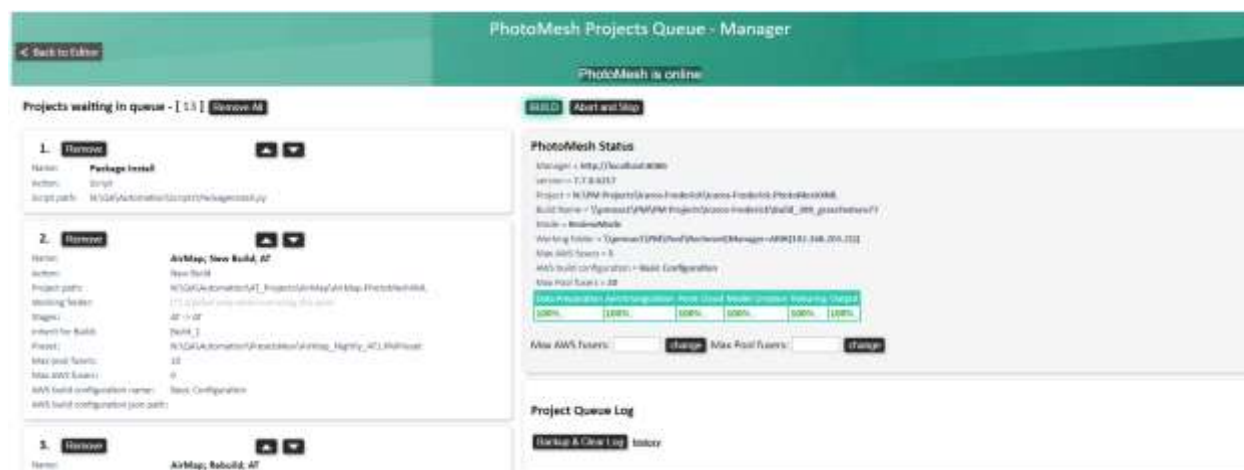
AWS and Cloud Automation

Improvements to the AWS fusers management mechanism to optimize your cloud resources:

- Increased options for defining fuser allocation allow you to select the most cost-effective fuser solution for your build processing needs and steps. You can now define the instance type, AWS zone, Spot/On-Demand allocation type and even use GPU machines only when needed.

- Support for instance startup script to dynamically allocate cloud resources needed for the build process (disk mount).

- Better management of your cloud costs and reporting by supporting AWS tags by your manager and fuser instances.
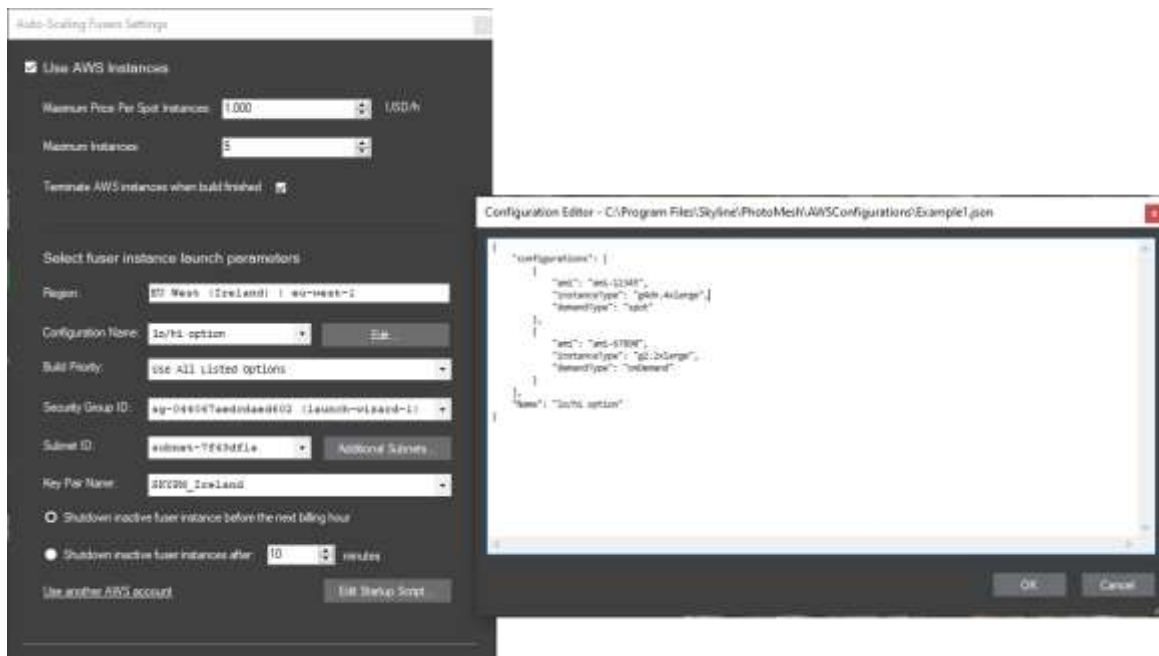
Usability Enhancements

New Item List

The Item List provides a powerful and efficient method of viewing information about and working with the items in a Project Tree group: photos, Lidar, reconstruction tiles, AT tiles, manual retouch layers, manually created water polygons and every imported water polygon layer. You can easily perform a textual or geographic search of the list to find and select the item you need, and then perform a wide range of actions on the item using an easy-to-access right-click menu or ribbon commands.

Selection	Collection	Name	Visible	Exclude	Project Original	X	Y	Altitude	Omega	Phi
	CAM3	15_2_285_C.tif	✓			-77.416882	39.416324	535.898937	6.400739	-36.38894
	CAM3	13_2_247_C.tif	✓			-77.414755	39.418086	535.498560	-1.023941	40.11156
	CAM3	15_2_284_C.tif	✓			-77.416879	39.415516	534.623305	5.452428	-34.35664
	CAM3	13_2_246_C.tif	✓			-77.414710	39.417290	537.020968	1.100079	41.51492
	CAM3	13_2_245_C.tif	✓			-77.414698	39.416528	538.517392	4.143940	43.33769
	CAM3	15_2_283_C.tif	✓			-77.416841	39.414760	533.113796	2.464684	-40.53048
	CAM3	13_2_243_C.tif	✓			-77.414675	39.414966	543.133684	4.388038	44.92065
	CAM3	13_2_242_C.tif	✓			-77.414644	39.414151	545.484226	0.148243	42.46198
	CAM3	13_2_241_C.tif	✓			-77.414649	39.413325	546.097707	4.529484	42.35891
	CAM3	15_2_280_C.tif	✓			-77.416645	39.412399	532.392554	-1.267799	-45.34305

Enhanced Selection Tools

PM 7.7's new point, polygon, and rectangle selection commands provide expanded options for selecting items in 3D Window by geographic location

3D Tile Selection Modes

New 3D tile selection modes allow you to view and inspect your project's 3D mesh layer output while selecting the underlying AT and reconstruction tiles.

Tiles User Status Mechanism

Using the new tile's user status mechanism, user can inspect, and tag AT and reconstructions tiles using a set of colored flags to indicate what additional operations still need to be performed on the tile: rebuild, external retouch, editing, or control points.
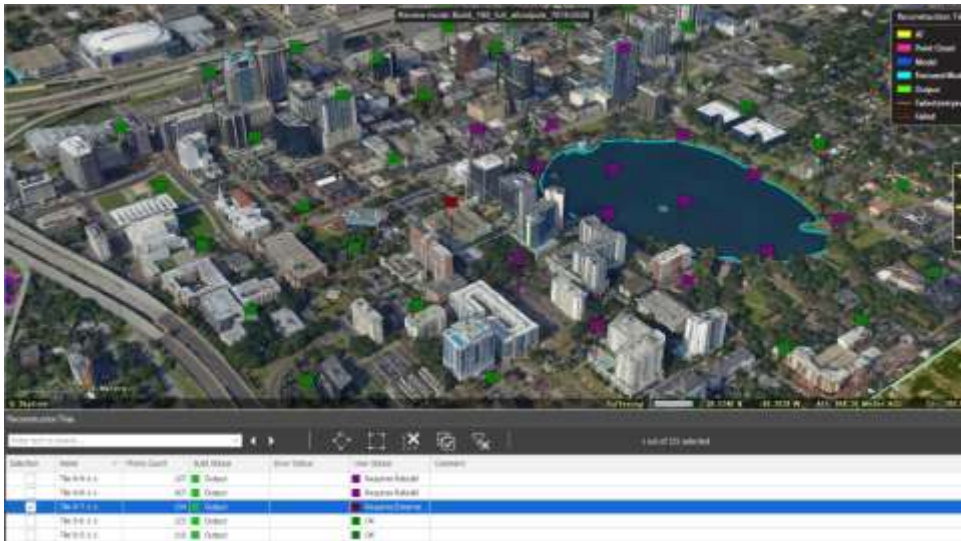
Cursor Selection Modes

New "List" selection mode limits your selection to the item type of the List currently open, making it easier to select the precise item you want in the 3D Window, e.g., if you want to select a tile when the Tile List is open, clicking on a camera icon that is somewhat concealing your tile, will still select the tile below it. You can easily toggle between List selection and Global selection, which selects whatever PhotoMesh item you point to in the 3D Window.

Depth Perception Mode

Highlight depth changes in the 3D model to make it easier to recognize the model's contours and shape. This can be particularly useful for an untextured model or point cloud without colors.
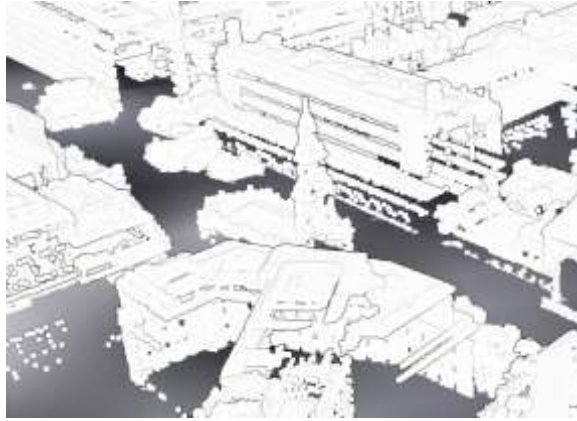
Drag and Drop

New drag and drop capability let you easily open a new project or load photos, Lidar, control points, and video files.

Import AT Area Polygon

The AT area can now be set either by drawing a polygon or by importing an AT area polygon.

New PhotoMesh Start Page

New Start page makes it easy to access recent projects, training materials, videos, demo projects and more.
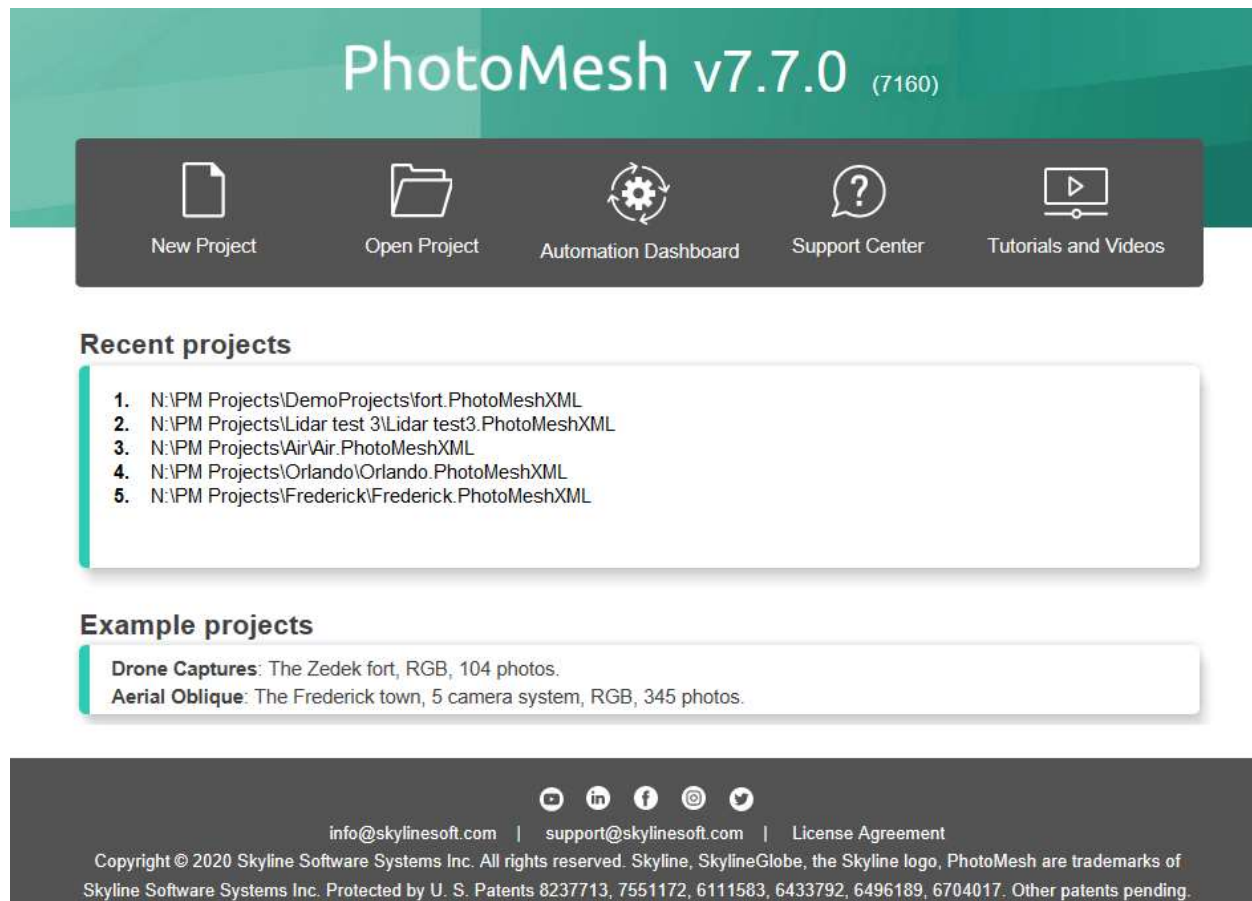
New Automation Dashboard

The Automation Dashboard consolidates all your PhotoMesh automation capabilities and documentation in one, easy-to-access place:

- Fusers Pool – Monitor and manage your fusers pool. The fusers pool is used to share fusers between several PhotoMesh master computers in your organization, using a distributed management approach.

- Amazon Web Services (AWS) Instances – Monitor and manage your AWS instances. PhotoMesh can work with Amazon Web Services to scale project production up and down based on resource needs. AWS fuser instances can be dynamically launched whenever there are unassigned pending fuser tasks, to further accelerate model creation.

- PhotoMesh API – View interactive documentation of the REST API for PhotoMesh Production. This API can be used to automate the production phases, e.g., load photo files or list, set AT and reconstruction areas, begin and manage a build, and create new build versions.

- Project Queue API – View technical and interactive documentation of the REST API for project queues and open the Project Queue Manager. The Queue API can be used to create a project queue and then activate, monitor, and manage queue processing.
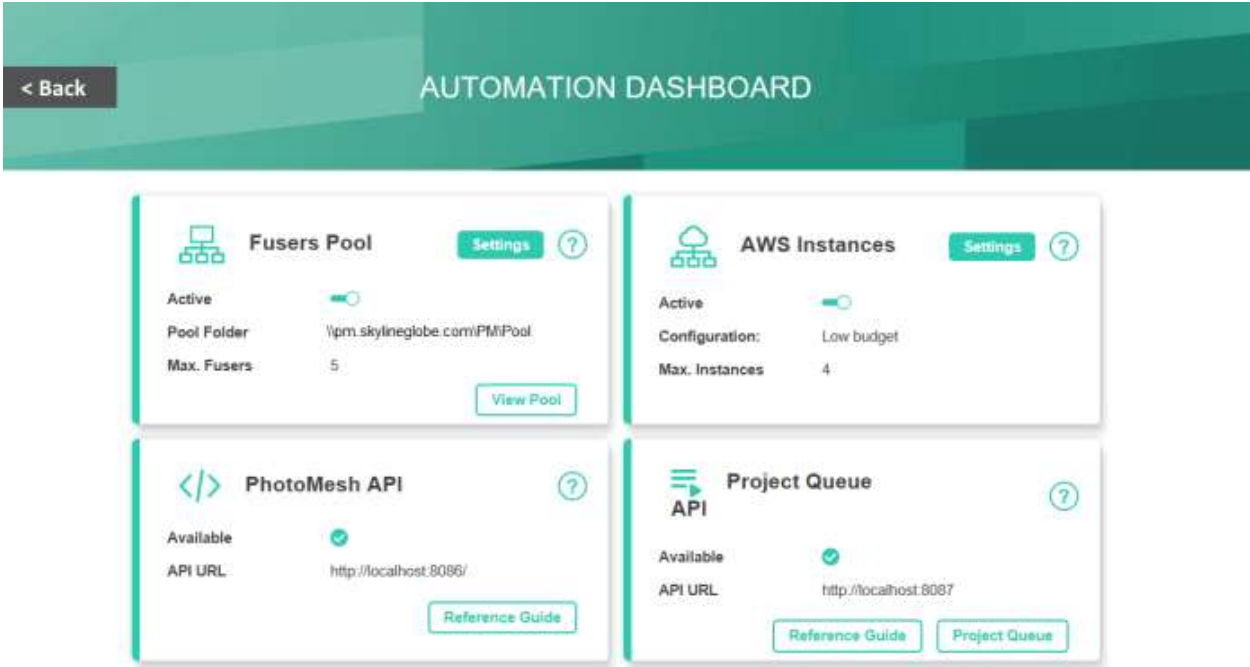
CHAPTER 4   THE PHOTOMESH WORKFLOW

About PhotoMesh Workflow

PhotoMesh converts standard 2D photographs into high-resolution, textured, 3D mesh models in a streamlined, step-by-step process:

- [Step 1: Imagery Acquisition](#)

- [Step 2: Creating a PhotoMesh Project](#)

- [Step 3: Loading Photos and Other Input Sources and Preparing the Project](#)

- [Step 4: Managing Local, Pool, and AWS Fusers](#)

- [Step 5: Building the Model](#)

- [Step 6: Managing and Reviewing the Build](#)

Step 1: Imagery Acquisition

The first step to creating a 3D model is imagery acquisition. All visible surfaces of the intended object or scene must be photographed from multiple overlapping viewpoints.

The accuracy of the resulting 3D mesh model is affected by several factors including: input image resolution and quality, lens distortion, overlapping ratio, directionality, and lighting. While PhotoMesh optimizes the accuracy of the 3D mesh model by selecting the best photos for an area, each factor can affect the overall output or cause varying levels of accuracy and quality in different areas of a project.

See the "[Imagery Acquisition](#)" chapter for more information.

Ground resolution = 5cm per pixel – Orlando USA

Ground resolution = 1 cm per pixel – Tel Aviv (drone)

Ground resolution = 3 cm per pixel - Frederick

Ground resolution = 7cm per pixel – Enfield England (Midas)

Step 2: Creating a PhotoMesh Project

Before beginning to work in PhotoMesh, you must create a new project or open an existing project. **See** "Creating and Opening a PhotoMesh Project" in the "Project Management" chapter for more information.

Step 3: Loading Photos and Other Input Sources and Preparing the Project

Photos and other input sources are loaded into a project using any of the following methods:

- Load photo files from disk (JPG, TIF, IIQ).

- Load an Excel/XML file with a list of photos, their file paths, all photo information (e.g., XYZ, OPK/YPR) and photo collection and camera information (e.g., focal length, sensor width, pixel size).

- Load a CSV file with a list of photos, their file paths and XYZ and OPK/YPR photo information.

- Load a PhotoMesh project. When a PhotoMesh project (.PhotoMeshXML) is loaded into a second project, only the project's photos and photo and collection parameters are loaded.

- Load a video from which PhotoMesh extracts individual image frames.

- Import photos of different data types (e.g., Inpho, Bingo, Stellacore) into the project with their metadata (e.g., sensor width, focal length, and aerotriangulation data). When AT results are imported, the AT step of the build process can be skipped (by selecting "Fully trusted input" in the Camera Position Accuracy AT parameter).

- Load Lidar data in .las, .laz or .e57 formats into a PhotoMesh project, integrating it with project photos to supplement project data and increase model accuracy.

See "Loading Photos" and "Importing Photos with Metadata" in the "Photo Management" chapter, and "Loading Lidar Data" in the "Lidar Management" chapter for information.

After photos have been loaded, do any of the following to prepare the project for the build:

- Modify photo and photo collection properties

- Verify that the camera symbols generate the expected cluster form

- Project photos on the terrain to see if they fit the terrain imagery or to better understand their position

- Show Lidars on the terrain to see if they fit the terrain imagery or to better understand their position

- Associate a Lidar with its trajectory data that provides information about the scanner's position while capturing the point cloud

- Improve the accuracy of the model's position by creating ground control points that associate real-world X, Y, Z coordinates with corresponding locations (in pixels) in photos

- Establish a correspondence between project photos and improve aerotriangulation results by creating tie points in the same physical point in three or more photos

- View photos in the photo viewer

- Import or draw water body polygons to generate a flat surface with correct texturing over user-defined water bodies

- Display the coverage map to identify which areas of the project have sufficient coverage and which require more

- Create a subset AT area in which to validate camera parameters and build settings

- Set a reconstruction area

See "Modifying Photo and Photo Collection Properties" in the "Photo Management" chapter, "Working with Water Body Polygons" in the "Reviewing your Build" chapter and the "Lidar Management", "Preparing the Project", and "Control Points" chapters for information.

Step 4: Managing Local, Pool, and AWS Fusers

Fusers allow you to share the demanding workload involved in processing the different build steps (data preparation, AT, point cloud, mesh model, texture, 3DML/other outputs) between several computers on the same local network. The master computer utilizes the computing power of the network's client computers by connecting to PhotoMesh fusers on the client machines.

Fusers can be shared between several PhotoMesh Build Managers in your organization, enabling each machine running PhotoMesh to use the combined resources of all the machines in the pool. PhotoMesh can also work with Amazon Web Services (AWS), adding and terminating virtual fuser machines, to scale a project up and down based on resource needs.

See "Installing PhotoMesh Fusers" and "Uninstalling PhotoMesh Fusers" in the "Getting Started" chapter for information on installing and starting the fusers. **See** "Fuser Management (on the Master computer)" and "Fuser Auto-Scaling" in the "Fusers" chapter for information on managing the fusers, and the "Getting Started with AWS" chapter for information about AWS fusers.

Step 5: Building the Model

After all photos are added to the project and all properties are set as required, you can launch PhotoMesh Build Manager to begin the build process. The build process provides flexibility both with regard to the area of the project being built (entire or selected area) and the build steps that are to be performed. To achieve optimal results and avoid unnecessary rebuilding, it is recommended to initially perform only the aerotriangulation step ("AT Only"), and then review the results, to ensure a precise AT which includes all project's photos that intersect with the AT area, before proceeding with the rest of the build. When an AT Only build is repeated following project adjustments, PhotoMesh determines precisely which tiles and which specific AT sub-processes were affected by the modification of properties or control points, enabling a much lighter and faster AT process.

The build process involves the following steps:

- **Data Preparation**
    - Conversion of photos to PhotoMesh's internal format (SMPT)
    - Conversion of Lidar to PhotoMesh's internal format (CPT)
    - Color correction of photos

- **Aerotriangulation**

- **Reconstruction**
    - Point Cloud Creation
    - Model Creation
    - Model Texturing

- **Project Output** - Select all required output formats: 3DML (for loading directly into TerraExplorer and streaming with SkylineGlobe Server), DAE, OBJ, 3D Tiles, I3S/SLPK, OSGB, Orthophoto, DSM, DTM/DEM, and LAS (Point cloud public file format). **See** "Setting Output Formats" in the "Building" chapter for information on the different file formats.

See the "Building" chapter for information.

Step 6: Managing and Reviewing the Build

Throughout the build process, you can monitor build progress from PhotoMesh Build Manager and view the build data already created in the PhotoMesh 3D Window. As soon as a build step is completed for a tile, its build data is viewable in the 3D Window, e.g. if aerotriangulation was performed, PhotoMesh can display the calculated camera positions in the 3D Window, and if reconstruction tiles were calculated, they can be displayed in the 3D Window, with each tile colorized according to the build step completed. You can also view point cloud, 3D model, and texture data for one or more tiles once this build step for the selected tile is completed. Build reports are available, as well, with statistical and graphical information about the build process. **See** "Reviewing Aerotriangulation (AT) and Reconstruction Tiles" in the "Reviewing Your Build" chapter for information.

After a build is complete, water bodies can be defined by drawing or importing a water body polygon, and mesh imperfections, such as bumps, irregular surfaces, or floating artifacts, can be marked for flattening, filling, and clean up using the Manual Retouch tool. **See** "Working with Water Body Polygons" and "Marking a Mesh Layer for Retouching" in the "Reviewing Your Build" chapter for information.

After reviewing build data, you can easily rebuild any of the following: the entire project, error tiles, or selected tiles. **See** "Monitoring a Build" in the "Building" chapter and the "Reviewing Your Build" chapter for information. Alternatively, you can create a new build version with different project steps and different

build parameters. You have the option to create a completely new build version or one which copies the AT information from the previous build, thereby accelerating the build process. **See** "Creating a New Build Version" and "Creating a New Build Version with a Copy of the Current Aerotriangulation Results" in the "Building" chapter for information.

Before rebuilding, 3D models can be exported when necessary to external software, such as Autodesk® MeshMixer, to perform certain edits, e.g. to remove a particular building that is being torn down, flatten surfaces and correct certain imperfections such as bumps or irregular surfaces, or edit textures, and then reimported to PhotoMesh. **See** the "Importing and Exporting Models" chapter for more information.

CHAPTER 5   BASIC CONCEPTS

Overview

This chapter addresses the following topics:

- Start Page

- Application Layout

- Error/Message Log

- PhotoMesh Ribbon

- Setting the Display of the 3D Window

- Saving a Project

- Undoing/Redoing an Action

- Selecting Objects in the 3D Window

- Dragging and Dropping Files in PhotoMesh

- Setting PhotoMesh Options

- Coordinate System Advanced Capabilities

Start Page

The Start Page is the first page you see after opening PhotoMesh from the **Program Files** menu or from the desktop icon. The Start Page also appears after you close your current project.

When you start PhotoMesh by selecting a .PhotoMeshXML file or link, PhotoMesh opens the project directly and does not show the Start Page.
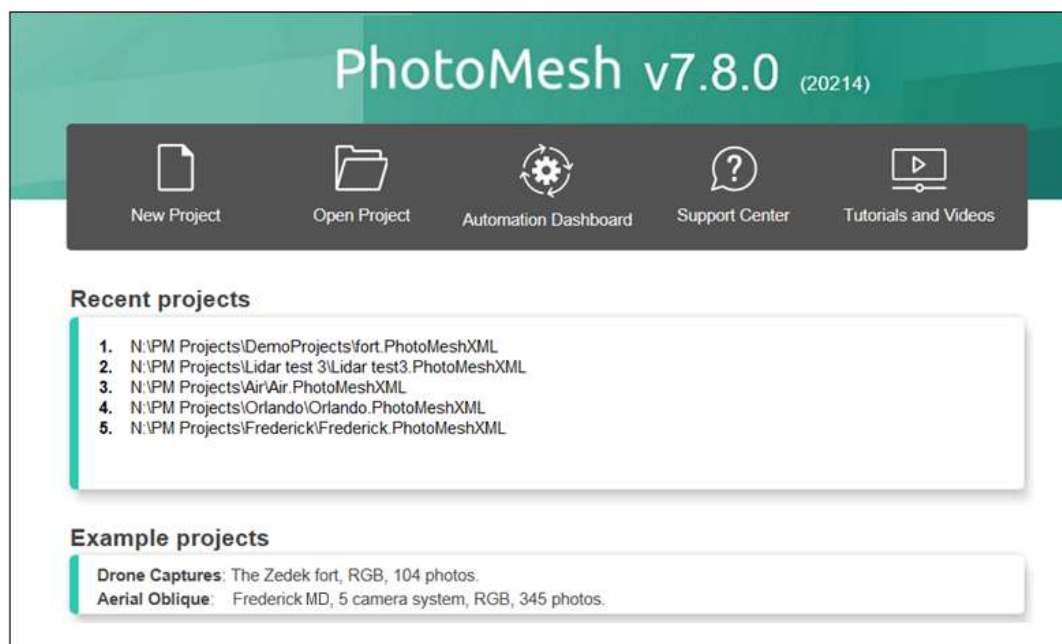
Start page options:

- **New Project** – Opens a dialog box in which you can create a new project.

- **Open Project** – Opens a dialog box in which you can browse to a project file.

- **Automation Dashboard** – Opens PhotoMesh's automation center, from where you can do all the following:

  - [Fusers Pool](#) – Monitor and manage fusers in your fusers pool.
  - [AWS Instances](#) – Monitor and manage your AWS instances.
  - [PhotoMesh API](#) – View interactive documentation of the REST API for PM Production.
  - [Project Queue API](#) – View technical and interactive documentation of the REST API for project queues and open the Project Queue Manager.

- **Support Center** – Opens Skyline's [knowledge base](#), a centralized repository of information about PhotoMesh.

- **Videos of major PhotoMesh Features** – Opens [Skyline's YouTube channel](#) with videos demonstrating PhotoMesh key features.

- **Recent Projects** – Opens one of your recent PhotoMesh projects.

- **Example Projects** – Download one of the ready-to-build example projects in PhotoMesh, and follow the instructions to get started building.

- **Contact Business and Sales** – Opens the Contact Skyline page on Skyline's website.

- **Technical Support** – Opens the Skyline Technical Support page on Skyline's website.

- **License Agreement** – Displays the PhotoMesh software license agreement on the main window.

Automation Dashboard
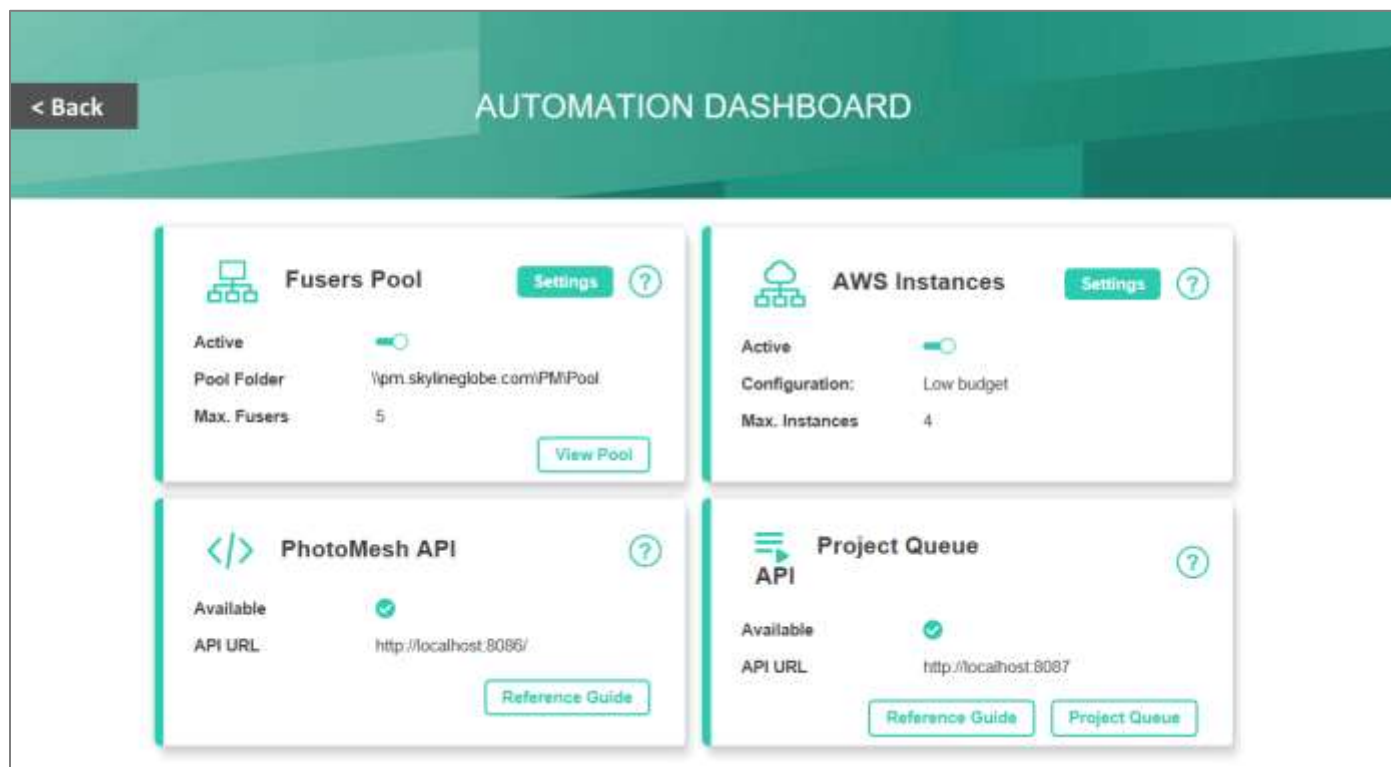
From PhotoMesh's Start page, you can access PhotoMesh's Automation Dashboard, from which you can control all the following:

- Fusers Pool – Monitor and manage fusers in your fusers pool:
  - Change pool settings: Active / Not active, and maximum number of fusers. Click **Settings** to open the **Auto Scaling Fusers Settings** dialog to change the pool status, pool folder, or maximum fusers. **See** "Using a Fusers Pool" in the "Fusers" chapter for more information.
  - Manage the fusers pool. Click **View Pool** to open the Pool Manager with information on all the fusers in the pool. **See** "Managing a Fusers Pool" in the "Fusers" chapter for more information
  - Click **Help** ⑦ to open the relevant section in Skyline's online knowledge base.

- AWS Instances – Monitor and manage your AWS instances:
  - Change AWS fuser settings: Active / Not active, and maximum number of instances. Click **Settings** to open the **Auto Scaling Fusers Settings** dialog to use/stop using AWS Instances, change maximum instances or other AWS settings. **See** "Using Amazon Web Services (AWS) Fusers" in the "Fusers" chapter for more information.
  - Click **Help** ⑦ to open the relevant section in Skyline's online knowledge base.

- PhotoMesh API – View interactive documentation of the REST API for PM Production. **See** "PhotoMesh API" in the "PhotoMesh Automation" chapter for more information. You can do all the following:
  - Click **Reference Guide** to view interactive documentation of the REST API for PM Production.

- Click **Help** ⑦ to open the relevant section in Skyline's online knowledge base.

- Project Queue API – View technical and interactive documentation of the REST API for project queues and open the Project Queue Manager. **See** "Queue API" in the "PhotoMesh Automation" chapter for more information. You can do all the following:

  - Click **Reference Guide** to view technical and interactive documentation of the REST API for project queues.
  - Click **Project Queue** to open the Project Queue Manager.
  - Click **Help** ⑦ to open the relevant section in Skyline's online knowledge base.

Application Layout

The PhotoMesh display includes five primary panels:

Panel	Description
3D Window	The main working area for various actions including preparing, monitoring, and reviewing the build.
Project Tree	Provides access to the items in the project. **See** "Using the Project Tree and Item List" chapter for further information. The Project Tree has the following groups: ▪ **Photos** – Includes all the photo collections loaded into the project. Information about the each of the photos in each collection is available from the Photos List. ▪ **Lidars** – All the Lidars in the project are listed in the Lidars List. ▪ **Control Points** – Includes all control points in the project. ▪ **AT Tiles** – All the AT tiles created in the current build (only visible in Review mode) are listed in the AT Tiles List. ▪ **Reconstruction Tiles** – All the reconstruction tiles created in the current build (only visible in Review mode) are listed in the Reconstruction Tiles List. ▪ **Retouch Layers** – Includes a Manual Retouch layer, a Water Polygons (Manually Created) group, and all the Water Polygons (Imported) layers. 　▪ **Manual Retouch** – All the manual retouch polygons are listed in the Manual Retouch List. 　▪ **Water (Manually Created)** – All water polygons are listed in the Water (Manually Created) List. 　▪ **Water (Imported)** – For each imported water polygon layer, all water polygons are listed in the respective Water (Imported) List. ▪ **Objects and Layers** – Includes all terrain objects: locations, imagery layers, elevations layers, 3DML and modify terrain/hole on terrain objects and some of the project outputs: 3DML, AT Calculated Photos, AT Tiles, DSM, DTM, and Orthophoto. These outputs are added to the Project Tree when a build is finished, and can be opened in TerraExplorer directly from the Project Tree's shortcut menu. The Project Tree's shortcut menu also provides an option to access to any of the project's outputs by opening the location of the project's output folder.
List	Displays a list of all the items in the currently selected Project Tree group, and information about each item. The Item List is displayed for the following groups:

	photos, Lidars, reconstruction tiles, AT tiles, manual retouch layers, manually created water polygons and every imported water polygon layer.
Properties	Displays data relating to the currently selected photo collection, photo, control point, or water polygon layer.
	Note: Lidar point cloud and terrain object properties are not displayed in the property sheet panel. You can view Lidar point cloud properties by right-clicking the Lidar in the Lidar List, and selecting **Style Point Cloud**. You can view terrain object properties by clicking **Edit** ⋯ in the Properties field.
	See "Modifying Photo and Photo Collection Properties" in the "Photo Management" chapter, "Editing Control Points" in the "Control Points" chapter, and "Setting Water Body Polygon Properties" in the "Reviewing your Build" chapter for information.
Error/Message Log	Records and displays warnings, messages, and errors to help you troubleshoot any PhotoMesh issue.
	See "Error/Message Log" in this chapter for more information.

You can create the application layout that is most convenient for you by repositioning or auto-hiding any of the panels, besides the 3D Window, on the screen.

Floating/Docked Windows

To reposition a panel:

1. Click the panel's title bar and drag the panel from its current location. A guide diamond with four arrows appears.
2. If you want the panel to float in front of the other application panels, drag it to the desired location.
3. If you want to dock the panel, do the following:

   a. Drag the panel towards the location where you want to dock it, and then rest the cursor on the guide diamond arrow that best corresponds to the desired docking location. A shaded blue outline of the panel appears in the area where the panel will dock.

   b. Release the mouse button to dock the panel in the area indicated.

Auto-Hide

When a panel is in auto-hide, a tab appears on an edge of the screen that controls the panel's display. When the cursor rests on the tab, the panel displays, and when it is moved away from the tab and the displayed panel, the panel disappears.

To auto-hide a panel, click the pushpin icon 📌 on the panel's title bar.

To view an auto-hidden panel:

▪ Move the cursor over the panel's tab.

   Note: If you want to hide the panel again, move the cursor off the panel.

To restore a panel from auto-hide, click the pushpin icon 📌 on the panel's title bar.

Error/Message Log

The PhotoMesh log records and displays warnings, messages, and errors to help you troubleshoot any PhotoMesh issue. Logs can be exported and sent to Skyline support:

▪ **Errors** – Alerts users of critical problems.

- **Warnings** – Alerts users of conditions they should be aware of, which might cause problems in the future.

- **Messages** – Records various activities.

Viewing the Log

To view the log:
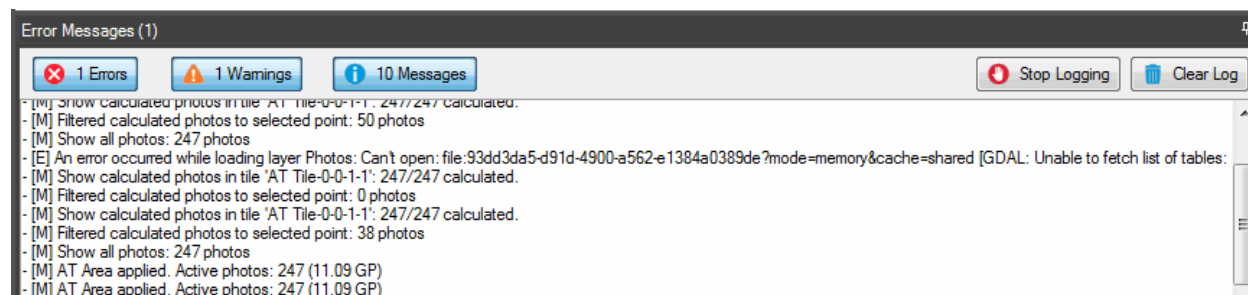
Error, Warning, and Message Log

- Click the **Errors**, **Warnings**, and **Messages** buttons to toggle the display of their respective messages on and off.

Clearing the Log

To clear the log:

- At the top of the log, click **Clear Log**.

Stop Logging

To stop logging:

- At the top of the log, click **Stop Logging**.

Auto-Hiding the Log

To auto-hide the log:

- Click the pushpin icon 📌 on the panel's title bar.

To view the auto-hidden log:

- Move the cursor over the panel's tab.

    Note: If you want to hide the panel again, move the cursor off the panel.

To restore the log from auto-hide:

- Click the pushpin icon 🔲 on the panel's title bar.

PhotoMesh Ribbon

The following commands are available from the **PhotoMesh** button 🌐 located at the upper left hand corner of the application:

Commands	Shortcut Key	Activity
New	CTRL+N	Creates a new PhotoMesh project. **See** "Creating and Opening a PhotoMesh Project" in this chapter for information.
Open	CTRL+O	Opens an existing PhotoMesh project. **See** "Creating and Opening a PhotoMesh Project" in the "Project Management" chapter for information.
Close		Closes the open PhotoMesh project.
Save	CTRL+S	Saves the current project. **See** "Save Project" in this chapter for information.
Resources		Opens resources that display license information, basic application information, or help files. **See** "PhotoMesh Resources" in the "Introduction" chapter for information.

Home Tab

The Home tab contains commands for setting up your project (setting build settings, adding photos to a project, setting photo display on the terrain, and setting the aerotriangulation and reconstruction areas), building the 3D model, and reviewing your project.

Home Tab

Group	Description
Select	**See** "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter.
Add	**See** "Loading Photos" in the "Photo Management" chapter, "Loading Lidar Data" in the "Lidar Management" chapter, "Setting the Terrain Database" in the "Project Management" chapter, "Working with Water Body Polygons" in the "Reviewing your Build" chapter, "Importing a Model" in the "Importing and Exporting Models" chapter and the "Control Points" chapter.
Photos	**See** "Displaying Camera Symbols" and "Showing the Coverage Map" in the "Preparing the Project" chapter.
Area	**See** "Setting the AT Area" and "Setting the Reconstruction Area" in the "Preparing the Project" chapter.
Build	**See** the "Building" chapter and "Managing Build Versions" in the "Reviewing Your Build" chapter.
Review	**See** the "Reviewing Your Build" chapter.

Tiles	**See** "Viewing Aerotriangulation (AT) Tiles" and "Viewing Reconstruction Tiles" in the "Reviewing Your Build" chapter.

View Tab

The View contains commands for changing your units and view in the 3D Window.

View Tab

Group	Description
Photos	**See** "Displaying Camera Symbols" and "Selecting Camera Symbol Size and Color" in the "Preparing the Project" chapter.
Environment	**See** "Setting the Display of the 3D Window" in this chapter.
Units	**See** "Units" in this chapter.
Mesh Layer	**See** "3DML Display Style Options" in the "Reviewing Your Build" chapter.
Reconstruction Tiles	**See** "Setting a Reconstruction Tile's Selection Style" and "Displaying User Status Flags and Legend" in the "Reviewing Your Build" chapter.

Tools Tab

The Tools tab provides a set of TerraExplorer analysis tools that help you validate photo parameters. It also contains terrain objects and layers that can be added to a PhotoMesh project and a command for connecting to SkylineGlobe Server.
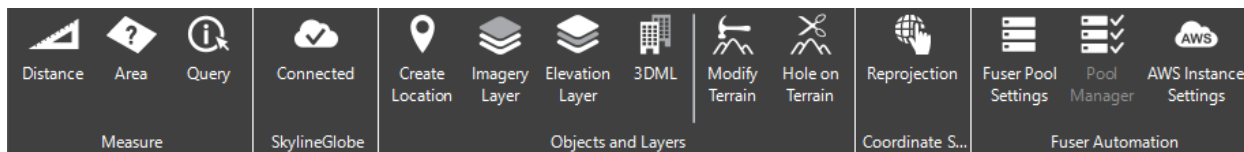
Tools Tab

Group	Description
Measure	**See** the "Analysis Tools" chapter.
SkylineGlobe	**See** "Connecting to a SkylineGlobe Server" in the "Working with SkylineGlobe Server" chapter.
Objects and Layers	**See** the "Adding Terrain Objects and Layers" chapter.
Coordinate System	**See** "" in the "Basic Concepts" chapter.
Fuser Automation	**See** "Fuser Auto-Scaling" in the "Fusers" chapter and the "PhotoMesh Automation" chapter.

Photo Tab

The Photo (contextual) tab contains tools for viewing the project's photos. This tab is displayed when a photo is selected in the photo list.
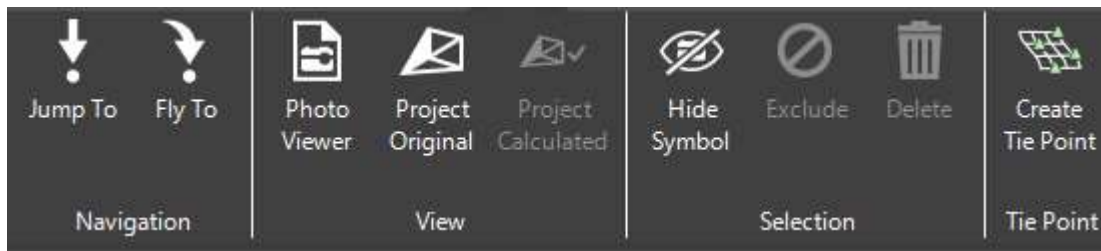
Photo Tab

Group	Description
Navigation	**See** "Navigating to a Photo" in the "Preparing the Project" chapter.
View	**See** "Viewing Photos in the Photo Viewer" and "Projecting a Photo on the Terrain" in the "Preparing the Project" chapter.
Selection	**See** "Displaying Camera Symbols" in the "Preparing the Project" chapter and "Excluding a Photo", and "Deleting a Photo" in the "Photo Management" chapter.
Tie Point	See "Creating Tie Points" in the "Control Points" chapter.

Lidar Tab

The Lidar (contextual) tab is displayed when a Lidar file is selected and it provides tools for viewing the Lidar data.
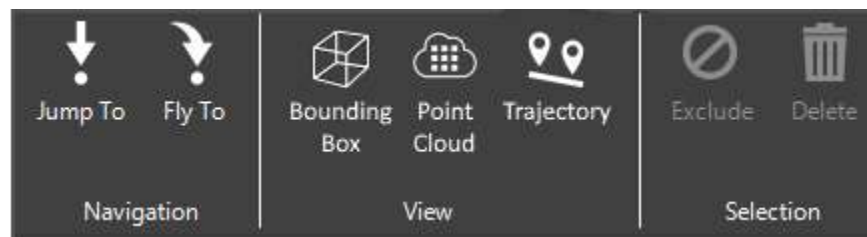
Lidar Tab

Group	Description
Navigation	**See** "Navigating to a Lidar" in the "Lidar Management" chapter.
View	**See** "Showing a Lidar Point Cloud on the Terrain" in the "Lidar Management" chapter.
Selection	**See** "Showing a Lidar Point Cloud on the Terrain", "Excluding a Lidar", and "Deleting a Lidar" in the "Lidar Management" chapter.

Tile Tab

The Tile (contextual) tab is displayed when either an AT or reconstruction tile is selected and it provides tools for displaying and exporting the different steps of the build output. The commands included in the tab depend on whether an AT or reconstruction tile is selected.

Reconstruction Tile Tab

AT Tile Tab

Group	Description
View	**See** "Navigating to a Tile", "Viewing a Tile's Build Report", and "Viewing an Aerotriangulation Tile's Detailed Report" in the "Reviewing Your Build" chapter.
Calculated Photos	**See "**Displaying Calculated Camera Positions for Tiles" in the "Reviewing Your Build" chapter.
Original Photos	**See "**Displaying Original Camera Positions for Tiles" in the "Reviewing Your Build" chapter.
User Status	**See** "Selecting Tiles" and "Viewing Tile Information in the Tile List" in the "Reviewing Your Build" chapter for information.
Sparse Point Cloud	**See** "Viewing an Aerotriangulation Tile's Sparse Point Cloud" in the "Reviewing Your Build" chapter. This group only displays when an AT tile is selected.
Show	**See** "Displaying Reconstruction Tile Results" in the "Reviewing Your Build" chapter. This group only displays when a reconstruction tile is selected.
Import	**See** "Importing a Model" in the "Importing and Exporting Models" chapter. This group only displays when a reconstruction tile is selected.
Export	**See** "Exporting a Point Cloud" and "Exporting a Model" in the "Importing and Exporting Models" chapter. This group only displays when a reconstruction tile is selected.

Control Point Tab

The Control Point (contextual) tab provides commands for working with control points and marking the position of control points in photos.
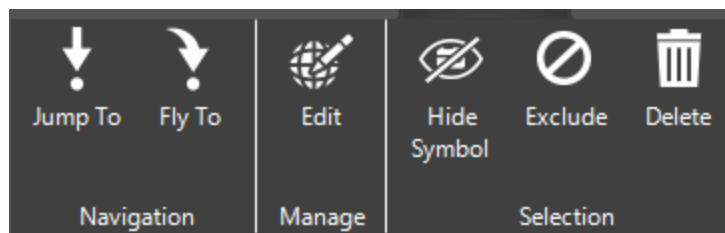
Control Point Tab

Group	Description
Navigation	**See** "Navigating to a Control Point" in the "Control Points" chapter.
Manage	**See** "Editing Control Points" and "Marking Control Points in Photos" in the "Control Points" chapter.
Selection	**See** "Hiding Control Point Symbols", "Excluding Control Points", and "Deleting Control Points" in the "Control Points" chapter.

Shortcut Keys

Most commands can be activated through a shortcut key combination (KeyTip) that appears on the Ribbon when you press the ALT key.

Note: Some alternate shortcut keys appear in this documentation in parentheses in the section where the command is explained.

To use KeyTips:

1. Press **ALT**. The KeyTips are displayed over each main tab and over each command on the Quick Access toolbar.
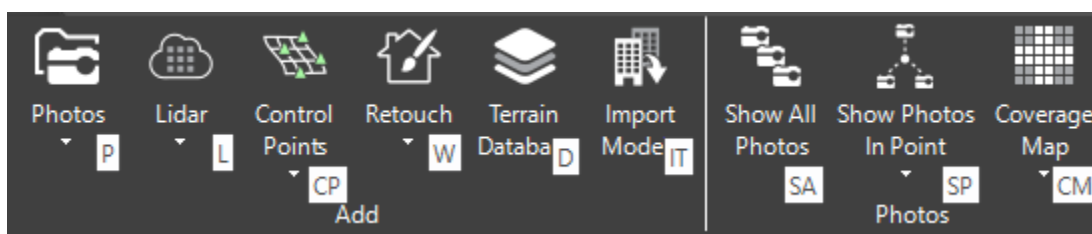
PhotoMesh KeyTips

2. Press the letter or number shown in the KeyTip over the tab/command that you want to use.
3. If you pressed a letter over one of the tabs, that tab is displayed showing KeyTips over the enabled commands. Press the letter of the required command.

Note: Pressing ALT a second time hides the KeyTips.

Quick Access Toolbar

The Quick Access toolbar is a customizable toolbar that contains a set of commands that are independent of the tab that is currently displayed.

Commands can be added or deleted from the Quick Access toolbar, according to each user's needs and convenience.

Adding/Deleting Commands from the Quick Access Toolbar

The Quick Access toolbar can be customized in two ways:

- By right-clicking the required commands on each tab.

  Note: This method is recommended for addition/deletion of just a few commands.

- By opening a dialog box in which you select the commands to include in the Quick Access toolbar.

  Note: This method is recommended for extensive modification of the Quick Access toolbar.

Adding/Deleting Commands by Right-Clicking

To add or delete a command:

1. If you want to add a command to the Quick Access toolbar, do the following:

   a. On the required tab, right-click the command you want added to the Quick Access toolbar.

   b. From the shortcut menu, select **Add to Quick Access Toolbar**. The command is added to the Quick Access toolbar.

2. If you want to delete a command from the Quick Access toolbar, do the following:

   a. On the **Quick Access** toolbar, right-click the command you want to delete.

   b. From the shortcut menu, select **Remove from Quick Access Toolbar**. The command is deleted from the Quick Access toolbar.

Adding/Deleting Commands Using the Customize Dialog Box

To add or delete commands from the Customize dialog box:

1. Right-click anywhere on the Ribbon.

2. From the shortcut menu, select **Customize Quick Access Toolbar**. The Customize dialog box is displayed.

3. From the **Choose commands from** drop-down list, select the Ribbon group whose commands you want to add or remove from the Quick Access toolbar.
   The Commands section displays all of the commands that are available on the selected option. The section on the right side of the dialog box displays all the commands that are presently included on your Quick Access toolbar.

4. If you want to add a command, select it in the **Commands** list and click **Add**.

5. If you want to remove a command, select it in the section on the right side of the dialog box, and click **Remove**.

6. If you want to change the order in which the commands are displayed on the Quick Access toolbar, in the section on the right side of the dialog box, drag the command to the desired position.

   Note: The default commands included in the Quick Access toolbar are: Save, Undo and Redo.

7. Click **OK**.

Setting the Display of the 3D Window

You can control the view, layout, and overlay of the 3D Window. This includes the display of several controls and indicators to orient you, options for creating a more realistic 3D World environment, and options for displaying the 3D model:

- [Navigation Controls](#)

- <u>Status Bar</u>

- <u>Date and Time Slider</u>

- <u>Sun</u>

- <u>Underground Mode</u>

- <u>Units</u>

- <u>Hide Terrain</u>

- <u>Mesh Model Display Style</u>

- <u>Highlight Depth Changes</u>

Navigation Controls

The navigation controls and virtual joystick enable you to zoom in or out or turn the plane to face north.

Navigation Controls

To navigate in the 3D Window using the navigation control:

- Click the navigation controls to turn North or zoom in or out of the terrain.

Navigation Control	Description
◎	Turn the plane to face north.
➕	Zoom in.
➖	Zoom out.

Status Bar

The status bar displays:

- The longitude, latitude, altitude, and direction of the plane/camera. Clicking on the status bar opens the Jump to Location dialog box. See the "Navigating in the 3D World" chapter in the TerraExplorer manual for further information.

- The current terrain buffering level in a graphical format. The display quality indicator represents the current terrain resolution for the area shown in the 3D Window, compared to the estimated maximum

resolution available in the terrain database. The graphic bar indicates values from 0%-100% of quality.

Status Bar

Sun

You can turn the sunlight source on or off. When turned on, the sun is used as the project's light source and affects the lighting on the terrain and on building objects.

To toggle the sunlight source on/off:

▪ On the **View** tab, in the **Environment** group, click **Sun**.

Date and Time Slider

The Date and Time Slider enables you to modify the current date and time so you can view each area of terrain in the date and time that you require. If the sun is used as the project's light source, the displayed light on the terrain corresponds to the sunlight there is on the terrain at the hour and date indicated by the blue marker on the slider.

To use the Date and Time slider:

1. On the **View** tab, in the **Environment** group, click **Time Slider**. The Date and Time slider is displayed on the upper left corner of the 3D Window.
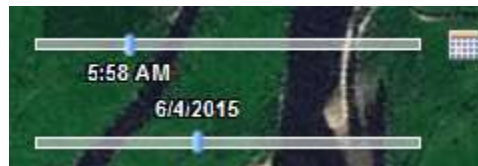
Date and Time Slider

2. Click the **Date and Time Settings** 🔲 icon to the right of the slider to select a mode. The Date and Time Setting dialog box is displayed.
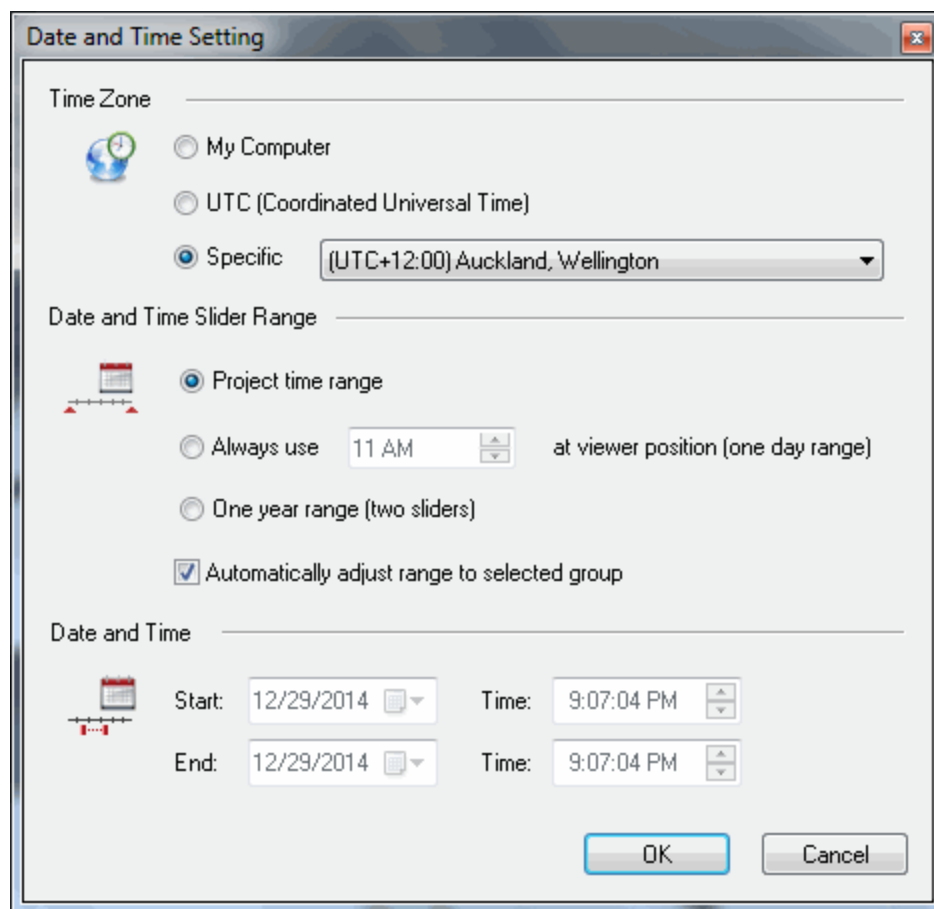
Date and Time Setting Dialog Box

3. Select one of the following time zones. The time zone you select determines which time zone is used for the time you select in the Date and Time Setting section in the same dialog box.

- **My computer** – Your local time zone.
- **UTC** (coordinated Universal Time) – UTC time (GMT + 0:00).
- **Specific** – The time zone you select from the drop-down list.

4. Control the interval of time for which you can view objects and features with timespans by selecting a date and time range for the date/time slider:

- Project time range

The slider's time range corresponds to the project's time range. Date and hour adjustments are both made on the same slider.

- Always use <> at viewer position

The slider's time range is 24 hours.
This option also requires you to select a specific time that will be the default current time (until the time is changed on the slider).

- One year range (two sliders)

The time slider's range is 24 hours and the date slider's range is one year.

5. If you want the date and time sliders ranges to automatically readjust to correspond to the timespan of the selected group, select the **Automatically adjust range to selected group** check box.

Note: This option does not apply in PhotoMesh.

6. Set the project's **current** time and date or time range.

Note: The current date and time can only be set when Project time range is selected above and there are objects or groups in the project for which timespans were set.

- If you want to set a specific current time and date, set the **same** Start and End Date and Time. The blue marker and yellow arrow are displayed together on the slider .

- If you want to set a current time **range** so that all objects and features whose timespans fall anywhere in the range display, set a Start Date and Time, and then set an End Date and Time. The blue marker on the slider indicates the Start Date/Time and the yellow arrow indicates the End Date/Time .

Note: Start and end date and time must be within the project range.

Underground Mode

The underground navigation mode allows you to explore the subsurface of the terrain. You can navigate under the terrain's surface and through buildings. A subsurface grid navigation aid appears when you navigate below the terrain, allowing you to navigate the same way as above ground.

To fly in underground mode:

- On the **View** tab, in the **Environment** group, click **Underground Mode**. You can freely go through the terrain surface in any direction.

Units

You can change the units used throughout PhotoMesh. Changing the unit settings substantially changes the display of your project: inputs and outputs and property sheets are all affected.

To change units:

- On the **View** tab, in the **Units** group, select the coordinate units and altitude:

Parameter	Description
Lat-Long Coordinates	
Decimal	Presents coordinate values in a decimal degrees format. This option is only available when the terrain is defined in the Lat/Long coordinate system.
Degrees; Minutes; Seconds	Presents coordinate values in a degrees, minutes and seconds format. This option is only available when the terrain is defined in the Lat/Long coordinate system.
Altitude Units	
Above Ground Level (AGL)	Displays altitude values as meters or feet above ground level.
Absolute Values	Displays altitude values as meters or feet above the terrain database vertical datum base ellipsoid.

Hide Terrain

You can hide the terrain texture (the terrain appears as a black image).

To toggle the display of the terrain on/off:

- On the **View** tab, in the **Environment** group, click **Hide Terrain**.

Display Style

There are several options for displaying the mesh model once it is built. The mesh model can include a wireframe and be textured or solid color. to view the interior of buildings or expose obscured sections of the 3D View.

To select a display style:

- On the **View** tab, in the **Mesh Layer** group, click **Display Style**, and select a style:
    - Texture
    - Texture + wireframe
    - Solid color
    - Solid color + wireframe

Depth Perception

You can highlight depth changes in the 3D model to make it easier to recognize the model's contours and shape. This can be particularly useful for an untextured model or point cloud without colors.

To highlight depth changes:

- On the **View** tab, toggle on **Depth Perception**.

Saving a Project

You can save a project and its system information.

Save Project

To save a project:

- Click the **PhotoMesh** button, and then click **Save** (Ctrl + S).

Undoing/Redoing an Action

You can undo and redo up to 100 actions in PhotoMesh.

To undo a single action:

▪ On the Quick Access Toolbar, click Undo [icon] (<CTRL> Z).

To undo multiple actions:
1. Click the arrow next to **Undo.**
2. Select the actions you want to undo, and then click the list. All the selected actions are undone.

To redo the last action that you undid:

▪ On the **Quick Access Toolbar**, click **Redo** [icon] (<CTRL> Y).

To redo multiple actions that you undid:
1. Click the arrow next to **Redo**.
2. Select the actions that you want to redo, and click the list. All the selected actions are redone.

Selecting Objects in the 3D Window

Two selection modes are available for selecting objects in the 3D Window: Global Selection and List Selection. Use Global Selection to select whichever PhotoMesh item you point to in the 3D Window. Use List Selection to limit your selection to only the item type of the list currently open, e.g., if the tile list is open, and you click in the 3D Window on a camera icon, the tile below it is selected.

To change the selection mode:

▪ On the **Home** tab, in the **Select** group, toggle on **Global Selection** or **List Selection**.

Dragging and Dropping Files in PhotoMesh

PhotoMesh's drag and drop capability supports project files, all photo and Lidar formats, control point lists, and video files. A single or multiple files can be dragged and dropped anywhere on the 3D Window.

File Type	How Loaded
.photmeshxml (project already open)	Open project file
.jpg .jpeg .jp2 .j2k jpc .tif .tiff, folder	Load photo or all photos in folder
.las .laz e57, folder	Load Lidar or all Lidar in folder
.gcp	Import control point data.
.asf, .wmv, .mp4, .avi, .fly, .mpg, .ts, .mov (only a single file at a time)	Load video.

Setting PhotoMesh Options

Default system settings for PhotoMesh are set from the Options dialog box.

To modify PhotoMesh Options:
1. Click the **PhotoMesh** button, and click **Options (F9)**.
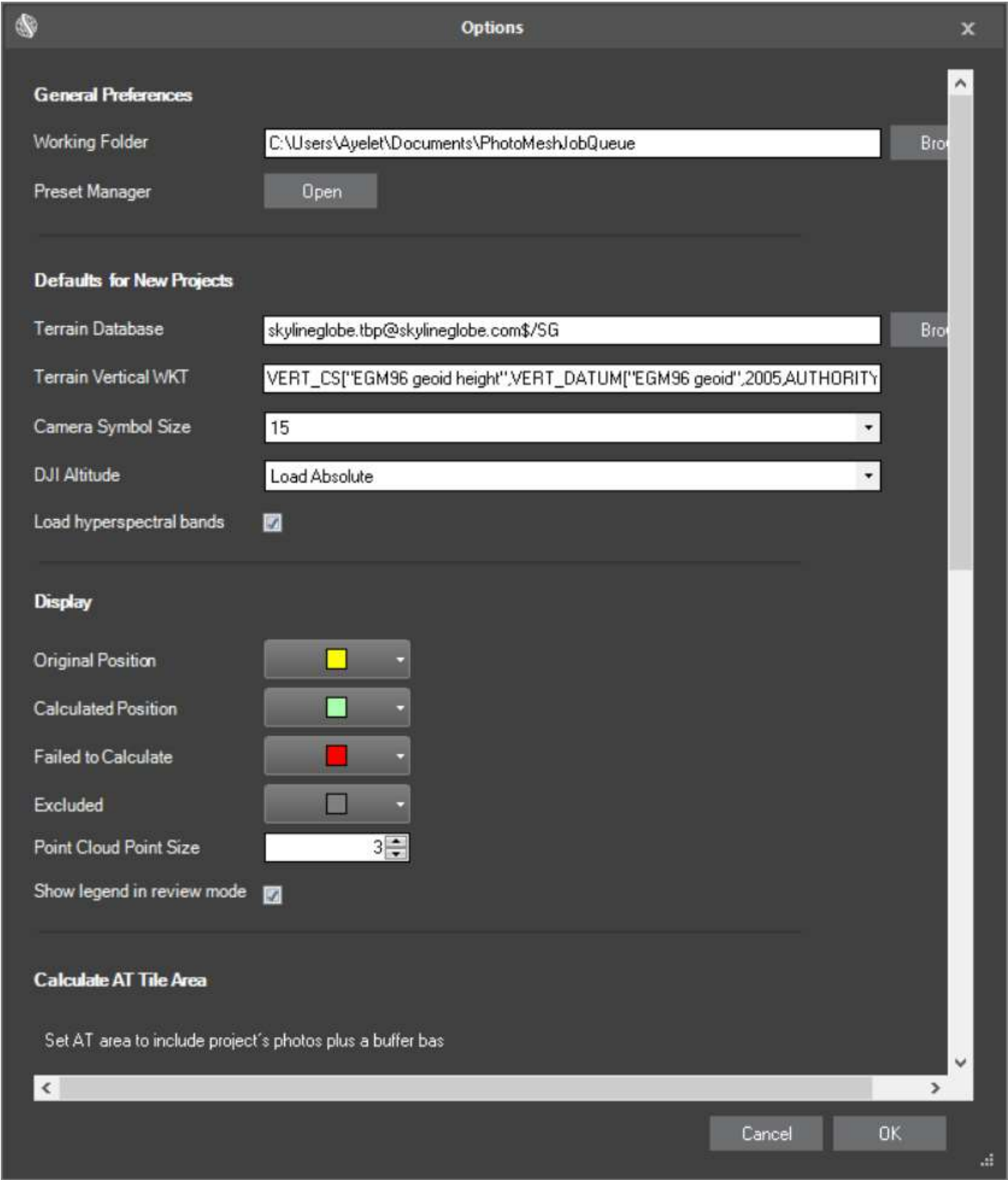
Options Dialog Box

2. Modify the following settings as required:

Parameter	Description
General Preferences	

Working Folder	The working folder that PhotoMesh Build Manager (on the master computer) and PhotoMesh fusers use to communicate: to deposit and collect pending tasks, and share information about fuser availability, status and progress. The working folder should be set to a network folder with read-write access permissions, for both the master and client computers. **See** "Setting the Working Folder (on the Master Computer)" in the "Fusers" chapter for information. Click **Browse** to browse to a different working folder, and then click **Select Folder**.
Preset Manager	Click **Open** to open the Preset Manager to load or manage preset files. **See** "Working with Presets" in the "Building" chapter for information.
Defaults for New Projects	
Terrain Database	Default MPT (local or remote) or .FLY to use as the base project when creating new project. Click **Browse** to browse to a different terrain database, and then click **Open**. **Note**: If a project is open, changes to the terrain database will take effect only after saving the project file and reopening it.
Terrain Vertical WKT	Well-Known Text of vertical datum of terrain database. **Note**: You can use the Coordinate System Generator to generate a WKT string defining the required coordinate system. Only enter the Vert_CS entity of the WKT in this field. **See** "Coordinate System" in the "Basic Concepts" chapter for information.
Camera Symbol Size	Select the size of the camera symbols that display for each photo.
DJI Altitude	Select the DJI altitude information to read from the metadata encoded in the photo files: ▪ **Load Absolute** – Load the absolute altitude values. ▪ **Load Relative** – Load the relative altitude values, and define altitude offset based on the terrain elevation at the position of the first photo in each collection.
Load Hyperspectral Bands	Select the check box to load hyperspectral bands for photos.
Display	
Original Position	The camera symbol's color for photos in the original position. Click **Change** to select a different color, and then select the color.
Calculated Position	The camera symbol's color for photos in the calculated position. Click **Change** to select a different color, and then select the color.
Failed to Calculate	The camera symbol's color for photos whose position PhotoMesh was unable to calculate. Click **Change** to select a different color, and then select the color.
Excluded	The camera symbol's color for photos manually excluded by the user or outside the project crop area. Click **Change** to select a different color, and then select the color.
Point Cloud Point Size	Enter the maximum size, in pixels, to display each point.

Show legend in review mode	Select the check box to display AT Tile and Reconstruction Tile legends in the 3D Window when in review mode, which indicate what color is associated with each AT and Build status.
Calculate AT Tile Area	
Set AT area to include	PhotoMesh can automatically calculate the AT area based on all the project's photos plus a buffer. The automatic AT area's buffer can be based on the collections' largest effective range, i.e., the maximum distance from the camera for which photo information is considered valid for aerotriangulation, or on a buffer you set.
	Select one of the following options:
	• **Photo collections' effective range** - Calculate the AT area based on the positions of all the project's photos plus a buffer that is based on the collections' largest effective range.
	• **Use buffer of** - Calculate the AT area based on the positions of all the project's photos plus the buffer that you set.
Tile Bottom Buffer	When splitting photos into AT tiles, photos whose frustum intersects with a box that defines the AT tile are included in the tile. This box's base is defined by the AT area's minimum terrain elevation minus the Tile Bottom Buffer value (in meters).
Tile Top Buffer	When splitting photos into AT tiles, photos whose frustum intersects with a box that defines the AT tile are included in the tile. This box's top is defined by the AT area's maximum terrain elevation plus the Tile Top Buffer value (in meters).
Maximum Photos Per Tile	Maximum number of photos that can be included in an AT tile, even if the Do Not Split If Sum Exceed or Do Not Split If Child Exceed properties (below) are met.
	Note: When a tile is split, some photos will be included in both sub-tiles, based on frustum area.
Do Not Split If Sum Exceed	In cases where splitting a tile into sub-tiles, based on Maximum Photos Per Tile, results in a significant number of photos that are included in both tiles, it may not be desirable to split the tile. This property sets the maximum percent the sum of the photos in the sub-tiles can exceed the photos in the original tile. If the percent increase exceeds this value, the tile will not be split into sub-tiles even though Maximum Photos Per Tile was exceeded.
Do Not Split If Child Exceed	In cases where splitting a tile into sub-tiles, based on Maximum Photos Per Tile, results in sub-tiles that are nearly as large as the original tile, i.e., a significant number of photos were included in both tiles, it may not be desirable to split the tile. This property sets the maximum percentage (of the original tile) for either sub-tile after the split. If either of the sub-tiles exceeds this value, the tile will not be split into sub-tiles even though Maximum Photos Per Tile was exceeded.
Photo Frustum Buffer	When splitting photos into AT tiles, photos whose frustum volume intersects with a box that defines the AT tile are included in the tile. This property lets you increase or decrease the frustum volume by a set percent.
Fully Trusted Max GP	Maximum number of gigapixels per AT tile for fully trusted input data when aerotriangulation is not performed. **See** "Importing

	Photos with Metadata" in the "Photo Management" chapter for more information on fully trusted data.
User Status	
AT Tile	Select the name and color to represent each of the **user** status options in the AT Tiles List. The user status indicates what actions still need to be performed on the tile, e.g., rebuild or control points. **See** "Selecting Tiles" and "Viewing Tile Information in the Tile List" in the "Reviewing Your Build" chapter for information.
Reconstruction Tile	Select the color to represent each of the **user** status options in the Reconstruction Tiles List. The user status indicates what actions still need to be performed on the tile, e.g., rebuild, editing or external retouch. **See** "Selecting Tiles" and "Viewing Tile Information in the Tile List" in the "Reviewing Your Build" chapter for information.

Coordinate System

A coordinate system is a reference system used to represent the locations of project data within a common coordinate framework. A coordinate system can be projected (X, Y), geographic (latitude-longitude), or geocentric (X, Y, Z). Each coordinate system is associated with its Well-Known Text (WKT) description, a simple structured, text-based format for spatial reference system information that is easy to store and share between systems (**See** http://www.geoapi.org/2.0/javadoc/org/opengis/referencing/doc-files/WKT.html). A coordinate system can optionally include a vertical datum that defines the origin for height values.

When you add input data to a project, PhotoMesh reprojects the various inputs to the AT's project coordinate system and eventually to the output's coordinate system. The inputs and other project information (e.g., calculated photos and output previews) are also reprojected to the 3D Window's Lat-Long WGS84 coordinate system (with vertical datum if provided). If an input source does not include coordinate system information or PhotoMesh is unable to read this information, you must supply it so that PhotoMesh can reproject the source. Usually, the data supplier provides the information about the coordinate system used for each source.

The Coordinate System dialog box is used to define the coordinate system for all project data.

When data is in Lat-Long WGS84 or ECEF coordinate system and no other coordinate system was used for photos and ground control points, etc., the default system used is the Universal Transverse Mercator (UTM) grid system with the UTM zone based on where the center of the data falls.

To set the coordinate system:

1. Open the **Coordinate System** dialog box.
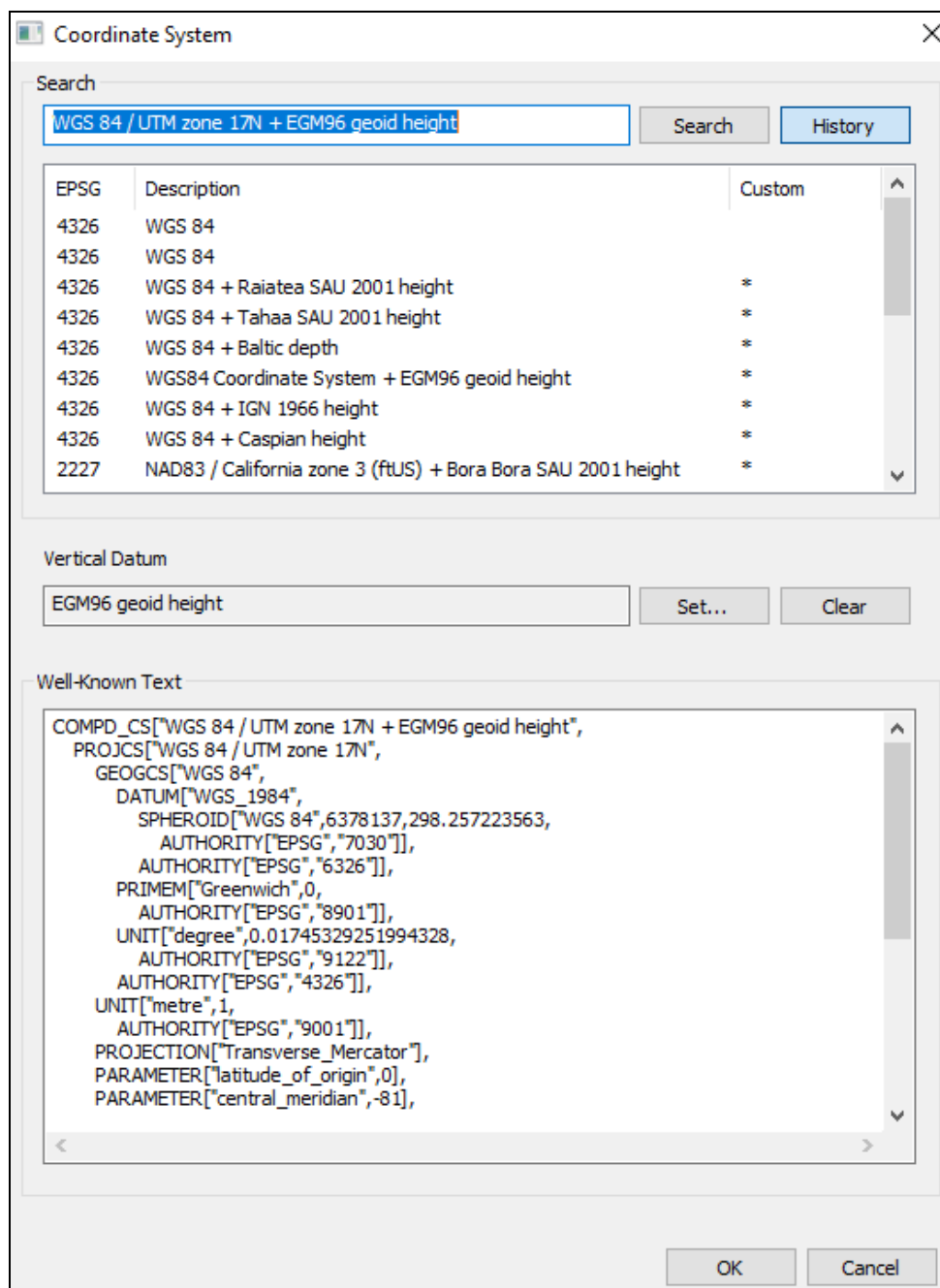
Coordinate System Dialog Box

2. Enter the coordinate system information using one of the following methods:

- Click **History** and select one of the recently used coordinate systems. The last 20 coordinate systems used are saved in the History for future use.

- In the **Search** field, type the EPSG code or a phrase from the Coordinate system Description, as indicated by the supplier, and click **Search**. Then select the required result from the search results displayed directly below.

- In the **Well-Known Text** section, type the WKT. The Well-Known Text representation of spatial reference systems, regulated by the Open Geospatial Consortium (OGC), provides a standard textual representation for coordinate system information. You can edit an existing WKT.

    Note: A custom coordinate system is listed with all the other coordinate systems, with an asterisk in the Custom column.

3. If your coordinate system includes vertical datum, click **Set** and select a vertical datum from a list of supported datums.

    Note: If the WKT of your coordinate system references a particular GTX file that was not bundled with the PhotoMesh installation, you can add the GTX by downloading it and placing it in the correct directory. **See** "Adding Vertical Datum (GTX) Files" in this chapter for information.

4. Click **OK**.

The following advanced coordinate system capabilities are available:

- Generate WKT strings for coordinate systems

- Add missing vertical datum grid shift files (GTX)

Coordinate System Projection Tool

The Projection Tool projects data from one coordinate system to another:

- Generate a WKT string that defines a coordinate system (optionally, with vertical datum) for inclusion in Excel/XML photo list files, control point files, and trajectory files.

- Convert coordinate, OPK, and YPR values from one coordinate system to another.

- Convert the rotation format from a rotation matrix to OPK/YPR. The rotation matrix transforms coordinates in the specified coordinate system into camera coordinates using: a 3x3 matrix with <Mij> tags, where Mij denotes the coefficient of the (i+1) row and the (j+1) column of the rotation matrix.

- Convert the rotation format from OPK/YPR to a rotation matrix

To generate a WKT string for a coordinate system:

1. On the **Tools** tab, in the **Coordinate System** group, click **Reprojection**. The Projection Tool dialog box is displayed.

2. Click **Open List**, and select a coordinate system.

3. Click **Set** and select a vertical datum from a list of supported datums.

4. Click **OK**. The generated WKT string can be copied into your files as required.

To convert from one coordinate system to another:

1. On the **Tools** tab, in the **Coordinate System** group, click **Reprojection**. The Projection Tool dialog box is displayed.

2. On the left side, do the following:

    a. Click **Open List**, and select a coordinate system.

    b. Click **Set** and select a vertical datum from a list of supported datums.

    c. Click **OK**.

    d. Enter the XYZ values, and optionally, the OPK or YPR values.

3. On the right side, do the following:

    a. Click **Open List**, and select the coordinate system you want to project to.

    b. Click **Set** and select a vertical datum from a list of supported datums.

    c. Click **OK**.

4. Click **Project** [→]. The XYZ, OPK, and YPR fields on the right side are populated with the projected values.

To convert a rotation matrix to OPK and YPR:

1. On the **Tools** tab, in the **Coordinate System** group, click **Reprojection**. The Projection Tool dialog box is displayed.

2. On either of the sides, do the following:

    a. Click **Open List**, and select a coordinate system.

    b. Click **Set** and select a vertical datum from a list of supported datums.

    c. Click **OK**.

    d. Enter the XYZ and OPK or YPR values.

3. Click **Convert** [↓]:

To convert OPK and YPR to a rotation matrix:

1. On the **Tools** tab, in the **Coordinate System** group, click **Reprojection**. The Projection Tool dialog box is displayed.

2. Enter a rotation matrix.

3. Click **Convert** [↑]:

Adding Vertical Datum (GTX) Files

PhotoMesh uses vertical datum grid shift files (GTX) for shifting and transforming between some coordinate systems that have different datums. Sometimes the Well-Known Text (WKT) of one of the coordinate systems in your project references a particular GTX file that was not bundled with your installation, or the reference itself is missing from the WKT. The steps below describe how to add the missing GTX information in both of these cases.

To add missing GTX information:

1. If a referenced GTX file was not bundled with the installation, download and unzip the required GTX file. Common GTX files are available from the following sites:

    ▪ https://github.com/OSGeo/proj-datumgrid/tree/master/

    ▪ https://vdatum.noaa.gov/download.php

    ▪ https://raw.githubusercontent.com/OSGeo/proj.4/gh-pages/proj.pdf

    ▪ https://proj.org/resource_files.html#transformation-grids

2. For computers with full PhotoMesh installation (**See** "Installing PhotoMesh" in the "Getting Started" chapter), copy the GTX file into the following subdirectories:

    ▪ The **\PhotoMesh\proj** subdirectory of PhotoMesh's installation directory.

    ▪ **The \PhotoMesh\Fuser\proj** subdirectory of PhotoMesh's installation directory.

3. For computers with only fuser installation (**See** "[Installing PhotoMesh Fusers](#)" in the "Getting Started" chapter), copy the GTX file into the *PhotoMesh\proj* subdirectory of PhotoMesh **Fuser's** installation directory.

4. If a reference to the GTX file is missing in the WKT, add the following reference within the WKT's VERT_DATUM object that is under the VERT_CS object:

```
EXTENSION["PROJ4_GRIDS","[Name of GTX File]"
```

E.g.,

```
COMPD_CS["RGF93 / Lambert-93 + NGF-IGN69 height",
    PROJCS["RGF93 / Lambert-93",
    ...,
    AUTHORITY["EPSG","2154"]],
VERT_CS["NGF-IGN69 height",
    VERT_DATUM["Nivellement General de la France - IGN69",2005,
        EXTENSION["PROJ4_GRIDS","RAF18.gtx"],
    AUTHORITY["EPSG","5119"]],
    UNIT["metre",1,AUTHORITY["EPSG","9001"]],
    AXIS["Up",UP],AUTHORITY["EPSG","5720"]]
]
```

CHAPTER 6   WORKING WITH SKYLINEGLOBE SERVER

About Working with SkylineGlobe Server

SkylineGlobe Server (SGS) is a private cloud solution that provides a comprehensive set of web services for publishing, storing, managing and streaming 3D spatial data. Remote PhotoMesh clients can connect to a SkylineGlobe Server to load imagery, elevation and 3DML layers into their projects. All data on your server is cataloged and referenced so clients can easily find the specific published layer required.
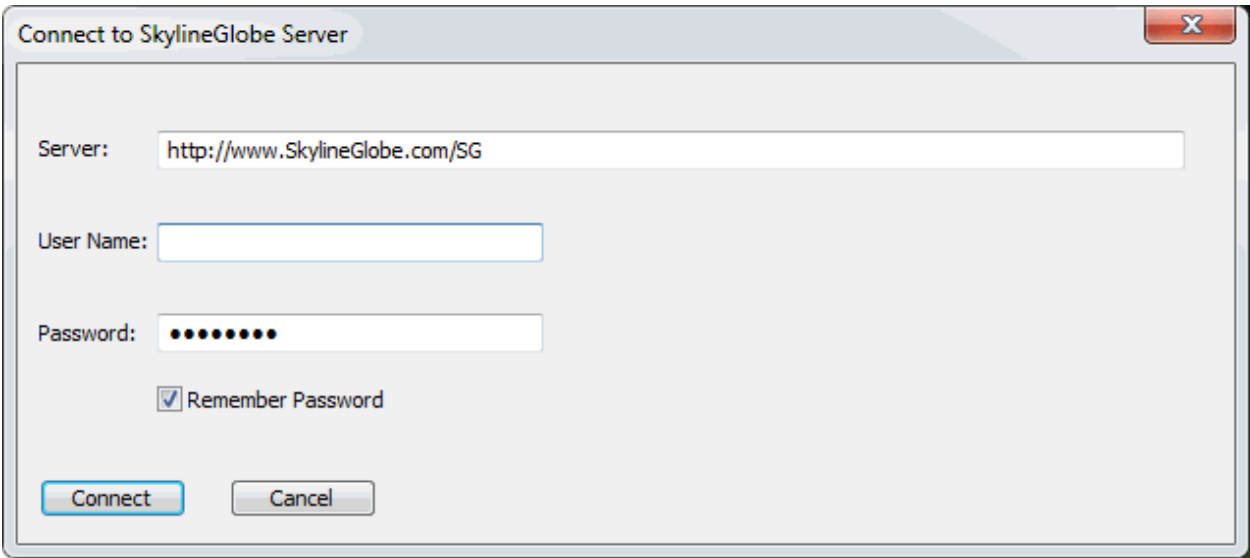
For more information about SkylineGlobe Server, see the SkylineGlobe Server User Guide.

Connecting to a SkylineGlobe Server

To connect to a SkylineGlobe Server:

1. On the **Tools** tab, in the **SkylineGlobe** group, click **Disconnected.** The SkylineGlobe Server dialog box is displayed.

   Note: If you are already connected to a server and want to connect to a different one, on the **Tools** tab, in the **SkylineGlobe** group, click **Connected**. In the Connect to SkylineGlobe Server dialog box that is displayed, first click **Disconnect**, and then enter the new server connection information.

Connect to SkylineGlobe Server Dialog Box

2. Enter the connection information and click **Connect**:

Parameter	Activity
Server	The Server Host Name, which is the URL link of the server.
User Name	User name to access the server (mandatory).
Password	Enter the password.
Remember Password	Select this option to save the password. The password will be saved and will be displayed using asterisks the next time the SkylineGlobe Server dialog box is displayed.

The values that you enter for Server and User Name are saved. The next time you load this window, this information is automatically filled in.

Loading SkylineGlobe Layers

To add a layer from a SkylineGlobe server:

1. On the **Tools** tab, in the **Objects and Layers** group, click the required layer type (Imagery Layer, Elevation Layer, or 3DML). The Browse dialog box is displayed.

2. In the **From Server** section, select **SkylineGlobe Server** as the **Server Type**, and click **Select**. The SkylineGlobe Layers dialog box is displayed, showing the selected layer type.
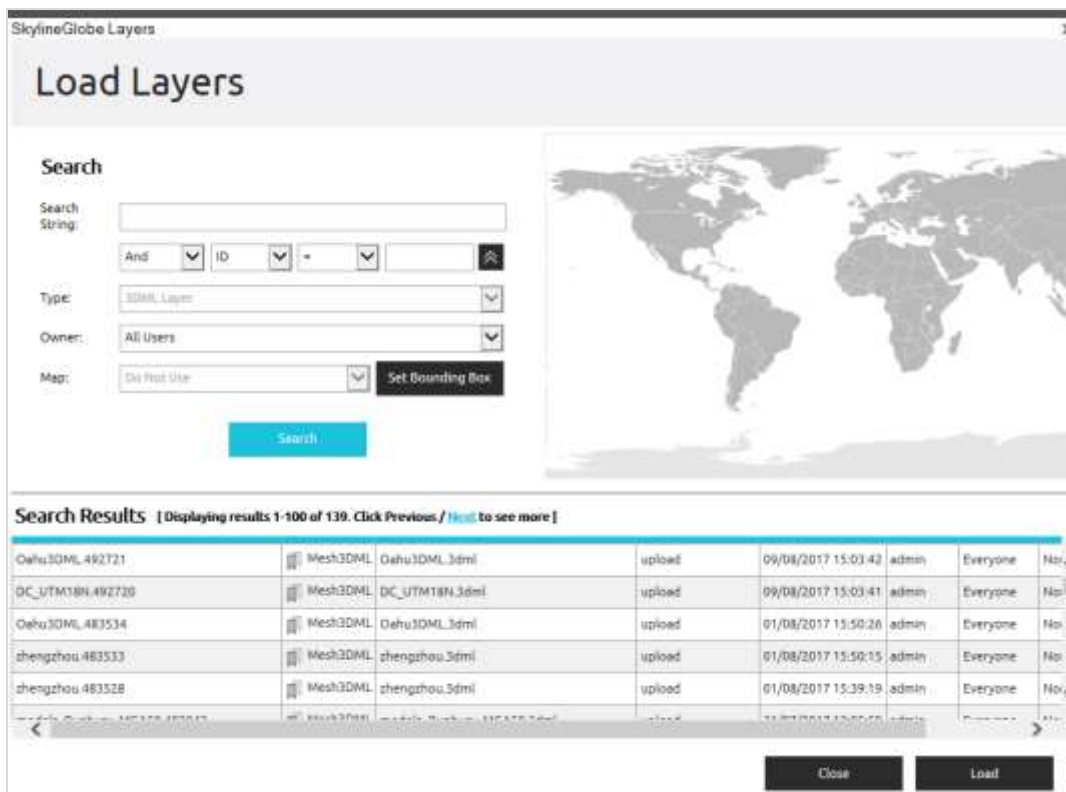
SkylineGlobe Layers Dialog Box

3. If you aren't already connected to the required SkylineGlobe Server, the Connect to SkylineGlobe Server dialog box is first displayed to enable you to connect. **See** "Connecting to a SkylineGlobe Server" in this chapter for information. After connecting, the SkylineGlobe Layers dialog box is displayed showing all layer types.

4. Search for the required layer. **See** "Searching for SkylineGlobe Layers" in the "Working with SkylineGlobe Server" chapter.

5. Select the layer you want to load, and click **Load Layer**. Use CTRL-click or SHIFT-click to multi-select layers.

Searching for SkylineGlobe Layers

To search for a layer in the SkylineGlobe Layers dialog box:

1. If you want to search for layers whose property information contains a specific string (e.g., ID, Name, Data Source), do the following:

    a. In the Search String field, type a search string or enter an attribute, condition, and value and click **Insert String** ⌃. The search expression is displayed in the Search string field.

    b. If you want to include additional strings in your search expression, select a connector: And ▾ and OR or, and repeat step a for the next condition.

2. If you want to search according to geographic location, do the following:

    c. Click **Set Bounding Box**, and select the required region by drawing a rectangle on the map on the right side of the page around the area to which the search is confined

    d. From the **Map** field, select one of the following:
    - **Do Not Use** – To ignore the bounding box.
    - **Contained** – If you want to search only for layers whose entire geometry fall within the bounding box.
    - **Intersect** – If you want to search for all layers whose geometry intersect at any point with the bounding box.

    e. If you want to clear the bounding box, click **Clear**.

3. Click **Search**. Your search results are displayed in the search results list at the bottom of the page.

CHAPTER 7   IMAGERY ACQUISITION

About Imagery Acquisition

The first step in the PhotoMesh process entails photographing all visible surfaces of the intended object or scene from multiple overlapping viewpoints. PhotoMesh supports standard image formats and large frame sensors.

The accuracy of the resulting 3D mesh model is affected by several factors including: input photo resolution and quality, lens distortion, overlapping ratio, directionality, and lighting. While PhotoMesh optimizes the accuracy of the 3D mesh model by selecting the best photos for an area, each factor can affect the overall output or cause varying levels of accuracy and quality in different areas of a project.

In most projects with good quality input, the resolution of the 3D model has been found to be approximately 1-2 times the ground sample distance of the source imagery.

Input Format and Resolution

Input Format/Data	
Image Format	Standard image formats (JPG, TIF, IIQ, etc.).
Positioning Information	XY coordinates and altitude of each photo.
Orientation Information	Omega, Phi, Kappa values (Recommended for large scale projects).
Camera	
Large Frame Sensors (100+ Mega Pixel)	Supported.
Focal Length	Photos with different focal lengths are supported.
Digital Zoom	No.

Directionality

Since PhotoMesh models can be viewed from any angle, it is important to capture nearly all visible surfaces, including walls and facades. Any object in the area of interest should be photographed multiple times from the top and at least four other directions.

Overlap

To create an accurate 3D model, each point in the area of interest should be visible in multiple photos. A >66% overlap between consecutive photos is recommended. Each point in the area of interest should be visible by, at least, 3 cameras (min 10° offset). PhotoMesh can combine photos with different ground resolutions.

79

For **structured** aerial photography

- Overlap in direction of flight (forward overlap) – 75 - 90% (recommended - 80%)

- Overlap between neighboring rows (side overlap) – 70 - 90% (recommended - 80%)
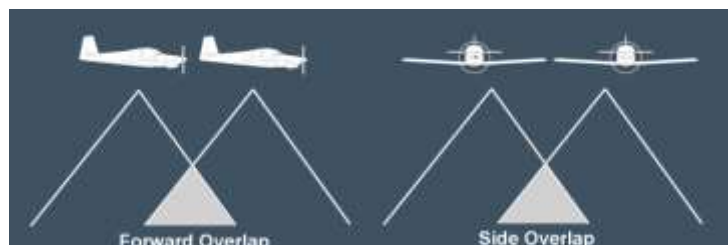
Structured Aerial Photography Recommendations

The following guidelines are recommended for optimal model accuracy and texture quality:

- **Capture time** – Avoid long shadows and sun reflections

- **Capture date** – Capture all photos in the same season

- **Collection system** – Use stable platforms with minimal smearing

Data Collection Recommendations for Photos

This section outlines Skyline's recommendations for optimal 3D PhotoMesh production from collected data:

- [Camera and lens combinations (hardware preferences)](#)

- [Camera settings and image format](#)

- [Flight planning and execution](#)

- [Mobile photography](#)

Camera and Lens Combinations (Hardware Preferences)

The following factors in sensor and lens selection are critical for 3D model geometry and quality:

- A high quality lens, with minimal distortion.

- Fixed focal length - to avoid alternating focal lengths in photos which are meant to be processed in the same dataset.

- For a standard grid collection pattern, it is important to have a lens with a field of view (FOV) at least 90 degrees wide in the long axis (60 degrees in the short axis, ~108 degrees in diagonal).

- A FOV < 120 degrees in the long axis, since these lenses tend to be fish-eye and too distorted for high-quality aerotriangulation.

- A lens which can maintain sharpness in the edges of the
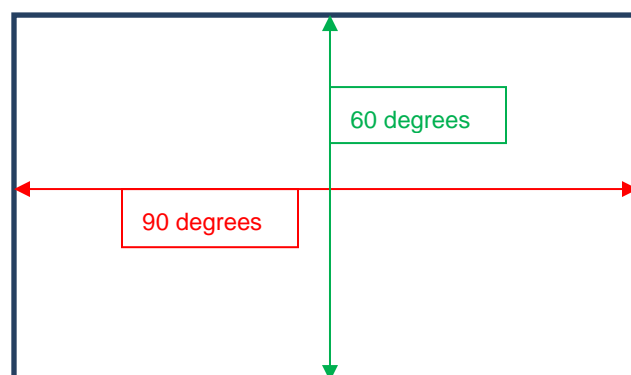
image - for proper aerotriangulation and texturing of facades in the 3D model.

- A sensor which can collect images rapidly - to maintain high overlap collection in low altitude flights while achieving proper overlap.

Camera Settings and Image Format

The following camera settings are recommended:

- Disable any "auto-rotate" options, to ensure that all of the images maintain a uniform orientation during the flight.

- Avoid slow shutter speed (generally < 1/1000) since it will cause significant motion blur, resulting in photos that cannot be used in the PhotoMesh production process. Instead of allowing a shutter speed drop to 1/640 or 1/500, it is preferable to open up the aperture value as wide as possible, even at risk of slight softness at the corners of the image. It is also possible to increase the ISO value in high quality cameras, and still maintain non-grainy images with high dynamic range in bright and dark areas.

- Disable any settings which might affect the relative location of the lens to the sensor, such as anti-shock or vibration features, since this will create a random principal point for the camera model. Vibration or motion in the UAV stabilization platform should be addressed with the camera and lens combination as a static object.

- Avoid auto-focus or automatic zoom. Focus the lens (usually to infinity) and lock it in that position.

- Use full resolution format, so that it is not down-sampled from the sensor pixels.

- Collect the photos in full uncompressed format. If JPEG format is necessary, use minimum compression to avoid artifacts or missing information in the images.

Flight Planning and Execution

The following flight patterns and execution are recommended:

- For single building or small area collection, use a circular flight pattern, with the target area in the center of the image from all perspectives.

- For large scale collection, use a grid pattern which allows for efficient area collection with the goal of creating 3D products.

- For nadir cameras with a field of view of approximately 60 degrees, tilt the camera 10-15 degrees forward to achieve the necessary 40-45 degree oblique perspective in all directions. The resulting field of view profile looks as follows:

-30 degrees   Nadir   30 degrees off nadir

Standard Nadir Flight

-15 degrees   Nadir   45 degrees off nadir

15 degrees   tilt forward

15 Degree Forward-Tilt
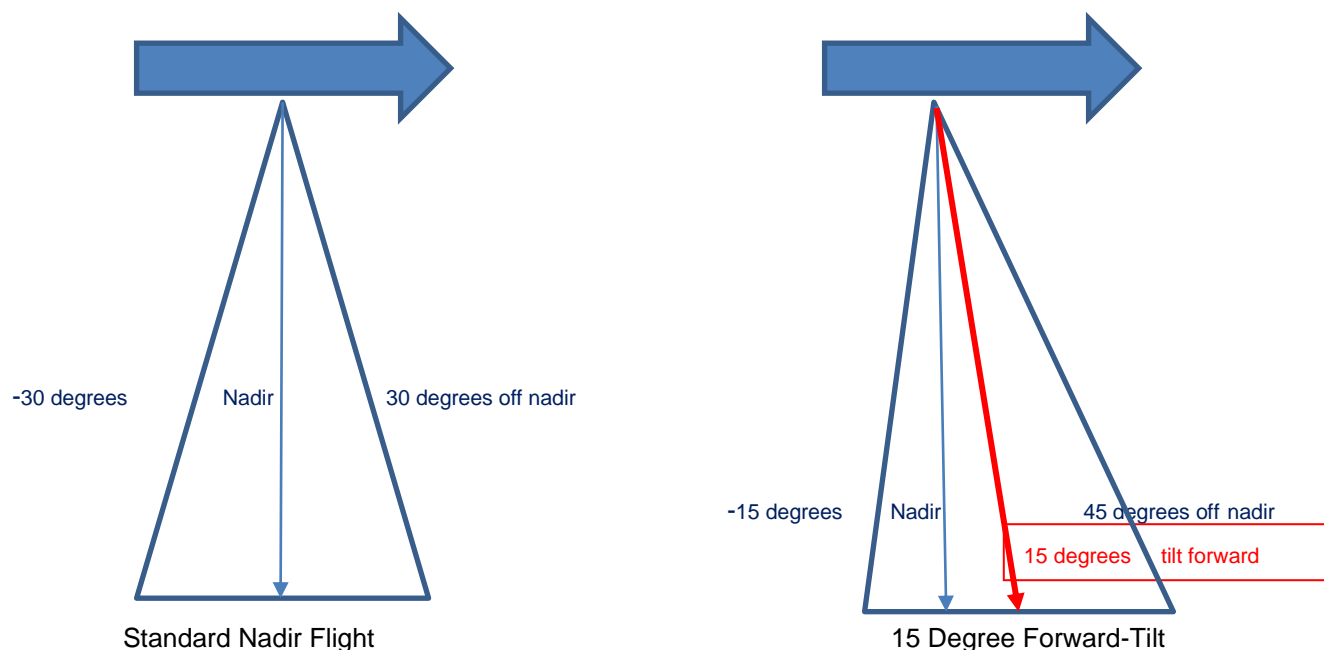
- Maintain a frontal (vertical) overlap of 75 - 90% (recommended - 80%).
- Maintain a side (lateral) overlap of 70 - 90% (recommended - 80%).

Mobile Photography

The following perspective and overlap are recommended for optimum reconstruction results:

- **Perspective** – Capture required objects and facades at a perpendicular angle and a 30 degree oblique variance (60 and 120 degrees) to maximize triangulation quality, while minimizing perspective distortion. In close-range data collection, any perspective which captures the object or façade at more than a 45 degree oblique will most likely be at a considerably lower resolution to provide effective input for the reconstruction.

- **Sky** – Limit the sky component in the photo to no more than 30%. This can be achieved by setting the tripod to a **higher** altitude when possible, as well as by slightly angling the camera down (instead of pointing the camera directly at the horizon, 90 degrees from the ground, it can be angled 70-80 degrees from the ground).

- **Overlap** – Maintain at least an 80% overlap, so that each object will appear in at least five photos. This provides PhotoMesh with the data to fill in holes in planes and facades that are partially obstructed by objects (such as trees and street furniture) which are closer to the camera.

Data Collection Recommendations for Full Motion Video (FMV)

This section outlines Skyline's recommendations for optimal 3D PhotoMesh reconstruction from full motion video. The recommendations below are for oblique data. It is generally recommended to use nadir video datasets only for production of orthophotos or point clouds, and not for full 3D Mesh models.

Video Camera – Hardware and Settings

- Use video setting most suitable for sharp video frames in motion.

- Use gyroscope or accelerometer-based gimbal or other 3-axis stabilization.

Flight Planning and Execution

- Slow flight to prevent motion blur.

- Minimum field of view - 45 degrees (optimal: 60-100 degrees), depending on the lens quality.

- Minimum lateral (side) overlap - 40% (optimal: 60-80%).

- Minimum overlap between consecutive photos - 60%. This can be achieved by a combination of flight speed and frame rate.

- It is recommended to use consistent zoom, especially in random flight patterns.

- For large area collection, use a criss-cross grid pattern.

- For single building or small area collection, video the target feature from different altitudes, and different pointing angles for complete coverage.

CHAPTER 8   PROJECT MANAGEMENT

Creating and Opening a PhotoMesh Project

When you create a new project, PhotoMesh creates an output folder with the same name as the project file for the project file and output files.

To start a new project:

1. Click the **PhotoMesh** button ⊛, and then click **New** (CTRL+N). The New Project dialog box is displayed.
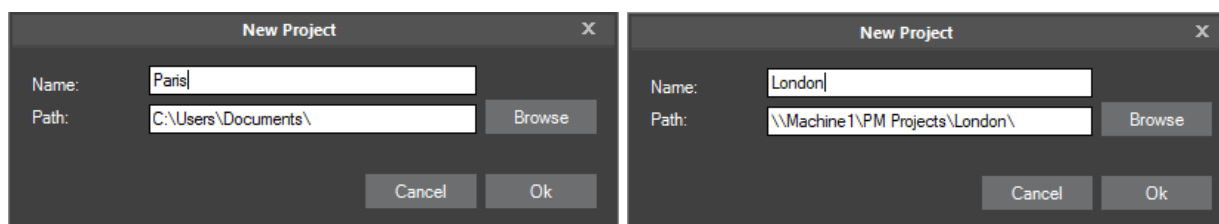
New Project on Local Drive / New Project on Network Drive

2. Browse to the required path for the project folder.

   Note: PhotoMesh can take advantage of computer clusters to accelerate database creation. **See** the "Fusers" chapter for more information. To enable this, both the project folder and the working folder should be set to network folders with read-write access permissions, for both the master and client computers. It is recommended to use UNC format (*[UNC Master] \folder\ProjectFolder, e.g.*: \\Machine1\c\PhotoMesh\ProjectFolder and *[UNC Master] \folder\WorkingFolder, e.g.*: \\Machine1\c\PhotoMesh7.0\WorkingFolder), though a mapped network drive can also be used as long as the same letter is mapped to the same share on all computers. The shared project folder and working folder must both be on a device that supports the number of concurrent network connections you require, e.g., Windows 10 limits the number of supported concurrent connections to 20.

3. In the **Name** field, type a project name, and then click **Ok**. PhotoMesh creates an output folder (with the same name as the project file) for the project file and output files.

4. If PhotoMesh cannot load the default base terrain (generally because you are not connected currently to the Internet), the Terrain Database dialog box is displayed. **See** "Setting the Terrain Database" in this chapter for more information.

To open an existing project:

1. Click the **PhotoMesh** button, and then click **Open**. The Open dialog box is displayed.

2. Browse to the required file and click **Open**.

3. If PhotoMesh cannot load the default base terrain (generally because you are not connected currently to the Internet), the Terrain Database dialog box is displayed. **See** "Setting the Terrain Database" in this chapter for more information.

Setting the Terrain Database

You can set the MPT or .FLY to use as the base terrain for your project.

To set the terrain database:

1. On the **Home** tab, in the **Add** group, click **Terrain Database**. The Terrain Database dialog box is displayed.
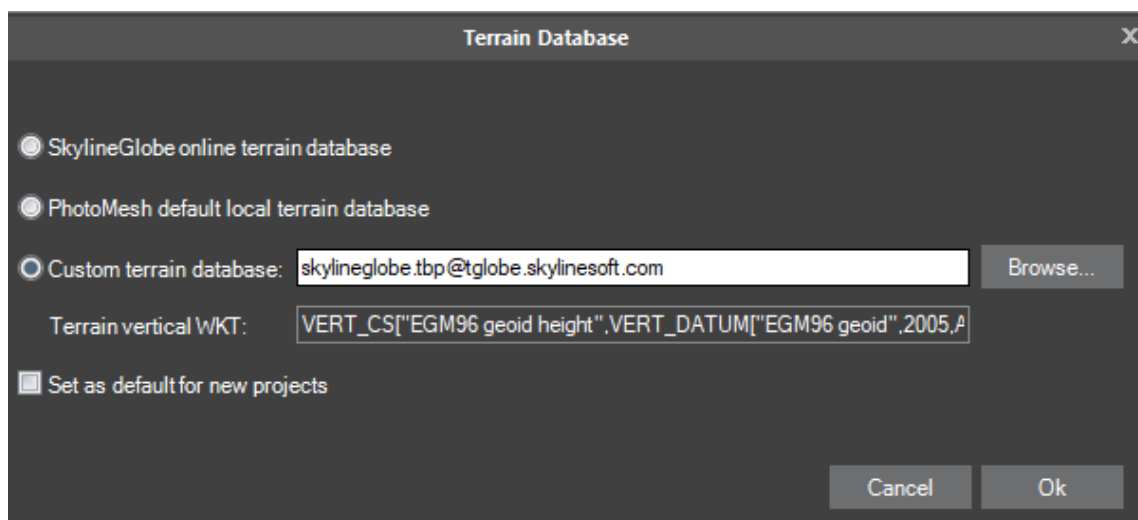
Terrain Database Dialog Box

2. Select one of the terrain database options:
   - SkylineGlobe online terrain database – To use the default terrain database (skylineglobe.tbp@tglobe.skylinesoft.com).
   - PhotoMesh default local terrain database – To use the base terrain file included with the PhotoMesh installation (Default_Local_Terrain.mpt).
   - Custom terrain database – To use a custom terrain database. Browse to the required file.
3. If you want to set your selected option as the default for new projects, select the check box. The default terrain database for new projects is listed in the PhotoMesh Options dialog box. **See** "Setting PhotoMesh Options" in the "Basic Concepts" chapter for more information.
4. Click **OK**.

Viewing Build Settings

The Build Settings dialog box is used to view project parameters, such as AT and reconstruction settings, output coordinate system, and output formats. Build settings can only be **edited** when beginning a build or rebuild process. **See** "Setting Build Parameters and Processing the Build" in the "Building" chapter for information.

To view Build Settings:

1. On the **Home** tab, in the **Build** group, click the arrow next to **Build**, and select **View Build Settings**. The Build Settings dialog box is displayed, showing the Steps tab.
2. Click the tabs to view the build parameters. **See** "Setting Build Parameters and Processing the Build" in the "Building" chapter for more information about these parameters.

Setting the Coordinate System

The Coordinate System Settings dialog box is used to set and view the project's internal coordinate system, in which aerotriangulation and reconstruction are performed. The coordinate system is also relevant for calculating the horizontal and vertical accuracy of the control points.

Note: When Review mode is on, you can only view the coordinate system.

To view Coordinate System Settings:

1. On the **Home** tab, in the **Build** group, click the arrow next to **Build**, and select **View Coordinate System Settings**. The Coordinate System Settings dialog box is displayed, showing the name of the project's coordinate system.

    Note: When Review mode is off, you can click the arrow to select a different coordinate system from the drop-down list. Select the **Custom** option to create a new custom coordinate system. **See** "Setting Output Formats (Coordinate System)" in this chapter and "Coordinate System" in the "Basic Concepts" chapter for more information.

2. Click **View** to view the full Well-Known Text (WKT) for the coordinate system.

CHAPTER 9   USING THE PROJECT TREE AND ITEM LIST

About the Project Tree

The Project Tree provides quick access to PhotoMesh items. The Project Tree lists all of the project's photo collections, AT tiles, reconstruction tiles, Lidars, control points, retouch layer, water body layers (manually created and imported), terrain objects, imagery and elevation layers and outputs.

Note: The number in brackets next to the name of each collection indicates the number of photos in this collection. The number in brackets next to the name of each control point indicates the number of photos in which this control point was marked.
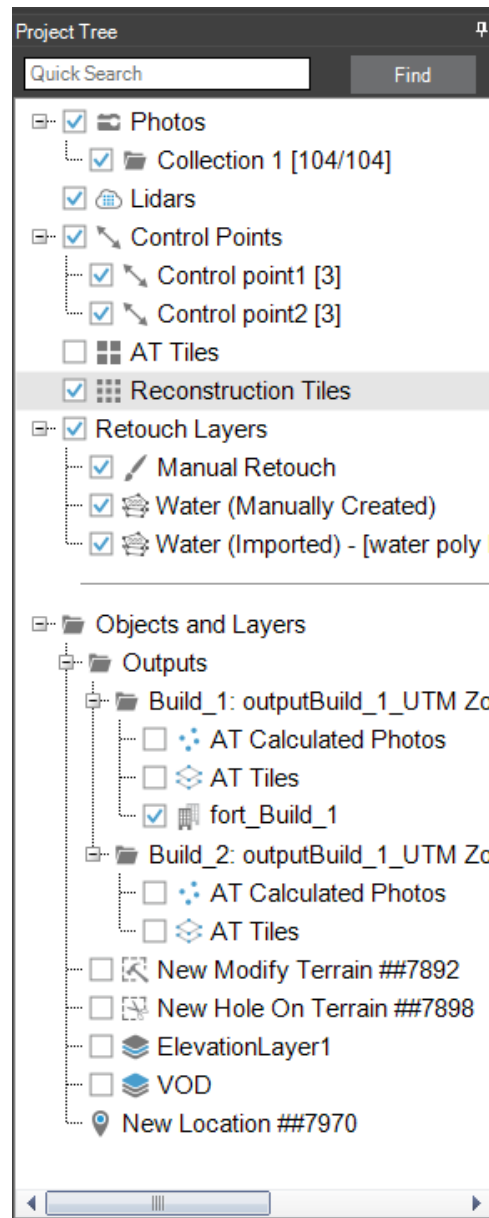
Project Tree

About the Item List

The Item List provides a list of all the items in the currently selected Project Tree group, and information about each item. An item list is displayed for each of the following groups: photos, Lidars, reconstruction tiles, AT tiles, manual retouch layers, manually created water polygons and every imported water polygon layer. The columns in each item list vary depending on the item type. The List panel is by default displayed under the 3D Window.

Photo List

Searching the Project Tree

You can search the Project Tree for any project element: collection group, photo, Lidar, control point, AT/reconstruction tile, retouch layer, water polygon, terrain object, imagery/elevation layer, or output.

To search:

- In the **Project Tree's Quick search** field, type a search term, and click **Find**. The first project element in the Project Tree with the specified text is selected.

To select an item:

- In the **Project Tree**, click the required item.

Showing and Hiding Items

Click on the item's show/hide icon ☑ to display or hide it in the 3D Window.

Selecting Items Using the Item List

You can search for and select items, e.g., photos, Lidars, and tiles, using the Item List. All selected items are selected both in the list (a check is displayed in the Selection column and their row is highlighted) and in the 3D Window.

To select items using an item list:

1. If you want to search for items according to geographic location, use any of the selection buttons:

88

	Selection Point - Select all list items that intersect with the point you draw. This option is only available for photos and Lidar.
	Selection Polygon - Select all list items that intersect with the polygon you draw.
	Selection Rectangle - Select all list items that intersect with the rectangle you draw.
	Select All - Select all list items.
	Clear Selection - Clear all selected list items.
	Clear Columns Filter – Clear any active filter that was applied. **See** "Filtering Data" in this chapter for information.
	Manual Retouch – Open Manual Retouch tool. This option is only available for manual retouch polygons. **See** "Marking a Mesh Layer for Retouching" in the "Reviewing Your Build" chapter for information.
	Filter by Reconstruction Area – Select all tiles in the current reconstruction area. This option is only available for reconstruction tiles. **See** "Setting the Reconstruction Area" in the "Preparing the Project" chapter for information on defining the area of the project on which PhotoMesh should perform the reconstruction steps of the build process.

2. If you want to search for group items whose property information contains a specific string (e.g., Name, Altitude or X/Y value), in the list's search field, type a search term, and press **ENTER**. The specified text is highlighted wherever it appears in the list. The number of search results found is indicated to the right of the search term entered. You can use the **Previous** ◀ and **Next** ▶ buttons to jump in the list from one search result to the next.

Filtering Data

To filter data:

1. Point to the column by which you want to filter, and click the **Filter** icon ▼. The Filter dialog box is displayed.
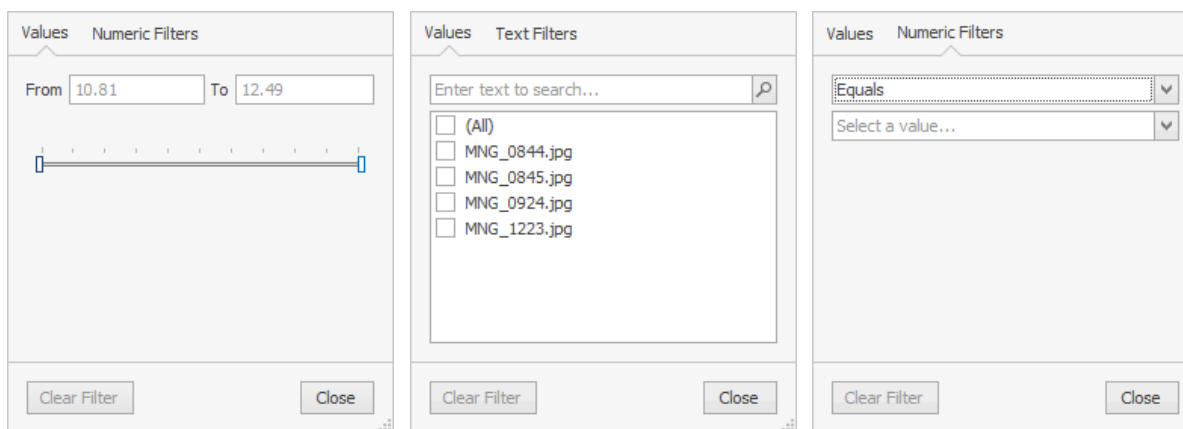
Numeric/Text Filters

2. Do any of the following:
    - To select a numeric/text filter, select the **Numeric Filters/Text Filters** tab. A filter is comprised of the (already) selected property, an operator, and a property value. Each property type has its corresponding operators. Select an **operator** from a list of operators, and a **property value**.
    - To filter by values, select the **Values** tab. For numeric values, enter the **From/To** values of the data you want to view. For text values, select the check boxes of the data you want to view. All other data will be filtered, i.e., temporarily hidden.
    - If you want to clear the filter, click **Clear Filter**.
3. Click **Close**. The applied filter(s) are listed at the top of the panel, e.g.,

    [visible] = False And [projected] = False

4. If you want to clear all filters that were applied, click **Clear Columns Filter** .

Filter Editor

You can use the Filter Editor to add and remove conditions. A search expression can consist of more the one condition. In this case, you need to insert a connector between the conditions to specify their relations. In addition to AND, you can use the OR, NOT and [] connectors to define the relations between conditions.

To access additional sorting commands, right-click any column header. Select any of the following commands:

Command	Description
Sort Ascending	Sort all data in the table by this column in ascending order.

Sort Descending	Sort all data in the table by this column in ascending order.
Group by This Column	Group the data by this column. **Note**: If you want to clear the grouping, right click above the column headers, and select Clear Grouping.
Show Group by Box	Display boxes, above the photo table, indicating how the data is grouped in the table. **Note**: If you want to clear the Group by boxes, right click above the column headers, and select Hide Group by Box.
Hide This Column	Hide the selected column
Column Chooser	Customize which columns show in the table. Drag and drop columns to and from the Customization dialog box.
Best Fit	Adjust the selected column's width for the best fit for its contents.
Best Fit (all columns)	Adjust the all the table's column widths for the overall best fit.
Filter Editor	Open the Filter Editor to edit the table filter.
Show Auto Filter Row	Display a row with a filter option for each of the columns.

Right-Click Shortcut Menus

Each project item type displays a unique menu of options when you right-click it in the Project Tree.

Photo Collection Menu

To open a Photo Collection's shortcut menu:

▪ In the **Project Tree**, right-click the required collection.
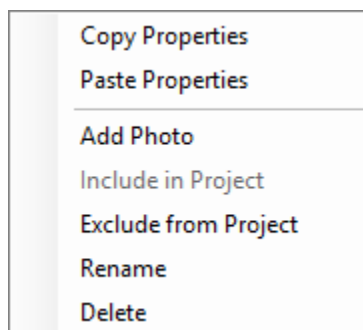
Collection Shortcut Menu

The Collection menu has the following options:

Note: Only the Copy Properties option is available when in Review mode.

91

Option	Description
Copy Properties	Copy the collection's properties.
Paste Properties	Paste the previously copied collection properties to this collection.
Add Photo	Add photo to collection.
Include in Project	Include collection in project.
Exclude from Project	Exclude collection from project.
Rename	Rename collection in Project Tree.
Delete	Delete the collection from the project.

Photo Menu

To open a Photo's shortcut menu:

- In the **Photos** list, right-click the required photo.
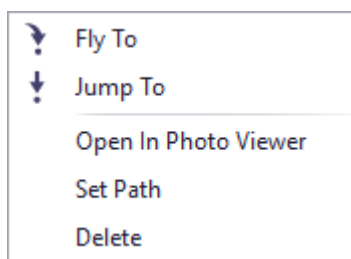
Photo Shortcut Menu

The Photo menu has the following options:

Option	Description
Fly To	Fly to the photo. **Note:** This option is only available when all position parameters (X, Y, and Altitude) were provided.
Jump To	Jump to the photo.
Open in Photo Viewer	Open the photo in Photo Viewer.
Set Path	Set the photo's directory. **Note:** This option is not available in Review mode.
Delete	Delete the photo from the project. **Note:** This option is not available in Review mode.

Tile (AT or Reconstruction) Menu

To open a Tile's shortcut menu:

- In the **Tile** list, right-click the required tile.
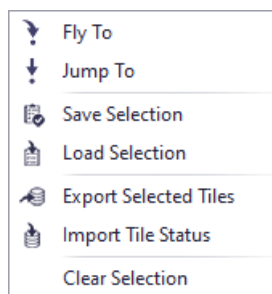
Tile Shortcut Menu

The Tile menu has the following options:

Option	Description
Fly To	Fly to the selected tile.
Jump To	Jump to the selected tile.
Save Selection	Save the selection information (which tiles were selected) to a .select text file.
Load Selection	Load a tile's .select text file with selection information (which tiles should be selected).
Export Selected Tiles	Export the selected tiles with all their information (e.g., status, photo count) to an SQLite file.
	Note: The exported tiles can be reviewed in TerraExplorer or other standard geospatial applications, and user status and comments added, and then reimported to PhotoMesh.
Import Tile Status	Reimport a tile from an SQLite file (generally after the tile was reviewed in TerraExplorer or another standard geospatial application, and user status and comments were added).
Clear Selection	Clear all selections.

Control Point Menu

To open the Control Point shortcut menu:

- In the **Project Tree**, right-click the **Control Points** group.
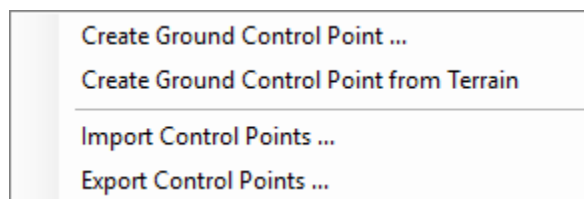
Control Point Shortcut Menu

The general Control Point menu has the following options:

Option	Description
Create Ground Control Point	Create a ground control point. Its position parameters can then be set from its property sheet.
Create Ground Control Point from Terrain	Create a ground control point from identifiable location on the terrain.
Import Control Points	Import control point data in a .gcp (text format) file.
Export Control Points	Export control point data to a .gcp (text format) file.

To open a specific control point's shortcut menu:

- In the **Project Tree**, right-click the required control point.
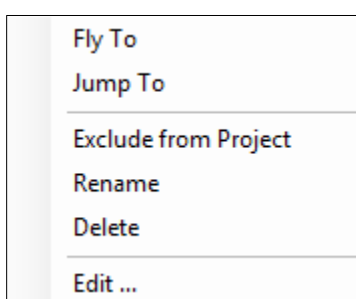
Specific Control Point's Shortcut Menu

The Control Point menu has the following options:

Option	Description
Fly To	Fly to the control point.
Jump To	Jump to the control point.
Exclude from project	Exclude the control point from the project.
Rename	Rename the control point in the Project Tree.
Delete	Delete the control point from the project.
Edit	Opens the Control Point Editor to mark control points in the photos. (**See** "Marking Control Points in Photos" in the "Control Points" chapter for information).

Objects and Layers Menu

To open an Objects and Layers shortcut menu:

- In the **Project Tree**, right-click the required terrain object or layer.
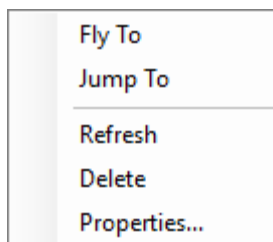
Objects and Layers Shortcut Menu

The Objects and Layers menu has the following options for objects and layers:

Option	Description
Fly To	Fly to the terrain object.
Jump To	Jump to the terrain object.
Refresh	Reloads the layer (This option is only displayed for layers).
Delete	Delete the terrain object from the project.
Properties	Open the terrain object's property sheet to edit or view its properties.

Outputs Menu

To open an Outputs shortcut menu:

▪ In the **Project Tree**, right-click the required build output.
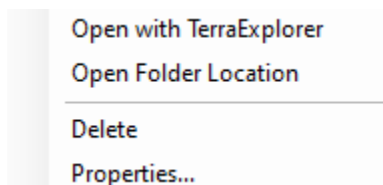
Outputs Shortcut Menu

The Outputs menu has the following options:

Option	Description
Open with TerraExplorer	Open the output in TerraExplorer (This option is only available for 3DML, AT Calculated Photos, AT Tiles, DSM, DTM, and Orthophoto).
Open Folder Location	Open the location of the project's output folder to view all outputs.
Export to LAS	Export DSM to LAS with Orthophoto colors (This option is only available for DSM).
Export to LAZ	Export DSM to LAZ with Orthophoto colors (This option is only available for DSM).
Delete	Delete the output from the project.
Properties	Open a property sheet with various properties for the output.

Water Polygon Menu

To open the water polygon layer general shortcut menu:

- In the **Project Tree**, right-click the **Retouch Layers** group.

Import Water Polygon Layer...
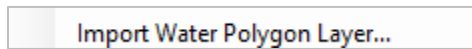
Retouch Layers General Shortcut Menu

The shortcut menu has the following options:

Option	Description
Import Water Polygon Layer	Import a water polygon from a shapefile or KML layer.

To open an individual water polygon layer's shortcut menu:

- In the **Project Tree**, right-click the required imported water polygon layer or the manually created water polygon layer.

Exclude from Project

Export Layer...

Delete
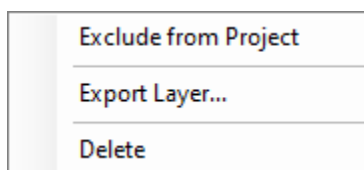
The water polygon layer shortcut menu has the following options:

Option	Description
Exclude from/Include in Project	Exclude/Include the polygon layer from the project.
Export Layer	Export the polygon layer to a shapefile.
Delete	Delete the polygon layer from the project. **Note:** This option is only available for an imported water polygon layer.

To open a specific water polygon's shortcut menu:

- In the **Project Tree**, select the water polygon layer, and in the **Polygon List**, right-click the required polygon.
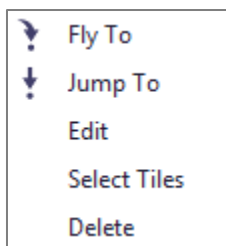
Water Polygon Shortcut Menu

The Water Polygon menu has the following options:

Option	Description
Fly To	Fly to the water polygon.
Jump To	Jump to the water polygon.
Edit	Edit the polygon's shape.
Select Tiles	Select the intersecting tiles.
Delete	Delete the water polygon from the project.

Retouch Layers Menu

To open a specific manual retouch operation's shortcut menu:

▪ In the **Project Tree**, select the **Manual Retouch** group, and then in the **Manual Retouch** list, right-click the required retouch polygon (i.e., row).
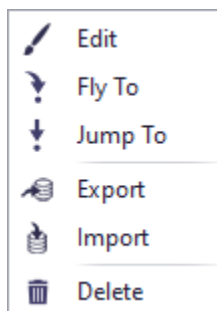
Manual Retouch Shortcut Menu

The Manual Retouch menu has the following options:

Option	Description
Edit	Edit the polygon.
Fly To	Fly to the polygon.
Jump To	Jump to the polygon.
Export	Export the retouch polygon layer with the selected retouch polygons to an SQLite file. The layer can be modified in external software and reimported to PhotoMesh.

97

Import	Reimport a retouch layer from an SQLite file (generally after modification in external software).
Delete	Delete the selected retouch polygons from the project.

Lidar Menu

To open a Lidar shortcut menu:

▪ In the **Project Tree**, select the **Lidars** group, and then in the **Lidars** list, right-click the required Lidar.
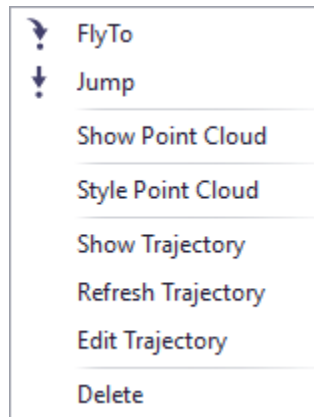
Lidar Shortcut Menu

The Lidar menu has the following options:

Option	Description
Fly To	Fly to the Lidar point cloud.
Jump	Jump to the Lidar point cloud.
Show Point Cloud	Show the Lidar point cloud on the terrain.
Style Point Cloud	Open the Lidar point cloud's property sheet to set point size, color, visibility, etc. **See** Lidar Point Cloud Properties Dialog Box" in the "Lidar Management" chapter.
Show Trajectory	Show the Lidar point cloud's trajectory. The trajectory is the path of the scanner's position while capturing the Lidar point cloud. **See** "Associating a Lidar Point Cloud with its Trajectory Data" in the "Lidar Management" chapter for information.
Refresh Trajectory	Refresh trajectory data.
Edit Trajectory	Open the trajectory data file in a text editor, so you can edit its data. **See** "Associating a Lidar Point Cloud with its Trajectory Data" in the "Lidar Management" chapter for information.
Delete	Delete the Lidar point cloud from the project.

Reference Project Menu

To open a reference project's collection shortcut menu:

▪ In the **Project Tree**, right-click the required reference project collection.
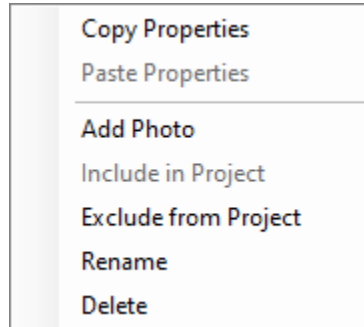
Reference Project Collection Shortcut Menu

The reference project collection menu has the following options:

Option	Description
Copy Properties	Copy the collection's properties.
Paste Properties	Paste the previously copied collection properties to this collection.
Add Photo	Add photo to collection.
Include in Project	Include collection in project.
Exclude from Project	Exclude collection from project.
Rename	Rename collection in Project Tree.
Delete	Delete the collection from the project.

Reference Photo Menu

To open a shortcut menu for a reference project's photos:

▪ In the **Reference Photos** list, right-click the required reference project photo.

Reference Photo Shortcut Menu

The reference photo menu has the following options:

Option	Description
Fly To	Fly to the photo. **Note:** This option is only available when all position parameters (X, Y, and Altitude) were provided.
Jump To	Jump to the photo.
Open in Photo Viewer	Open the photo in Photo Viewer.
Set Path	Set the photo's directory. **Note:** This option is not available in Review mode.
Delete	Delete the photo from the project. **Note:** This option is not available in Review mode.

Project Tree Symbols

Symbol		Meaning
Strikethrough font		Excluded from project. **Note:** If a collection is marked in strikethrough font, this indicates that all the collection's photos are excluded from the project.
?	orange question mark	▪ **On a collection** – Missing one or more of the camera properties that are used to set the focal length. (These properties vary depending on the Focal Length Normalization property setting set in the Collection property sheet. **See** "Collection Property Sheet" in the "Photo Management" chapter for information.) The missing property field is also marked in the Property Sheet with an orange question mark.
!	red exclamation mark	▪ **On a collection** – One or more photos is missing required properties. The missing property fields are also marked in the Property Sheet with a red exclamation mark. ▪ **On a control point** – The control point is marked in fewer than three photos. The missing property fields are also marked in the Property Sheet with a red exclamation mark. ▪ **On the Lidars group** – The coordinate system is missing from one of the Lidars. This Lidar is excluded from the project. **Note:** The missing property fields are also marked in the Property Sheet with a red exclamation mark.
[#calculate / #total]		Number of included photos/Total

CHAPTER 10 PHOTO MANAGEMENT

About Photo Management

This chapter deals with basic photo management tasks:

- [Loading of new photos in a project](#)

- [Excluding/removing a photo from the project](#)

- [Modifying photo and photo collection properties](#)

Photo Requirements

See "[About Imagery Acquisition](#)" in the "Imagery Acquisition" chapter for a list of requirements and recommendations for the photos from which the model will be generated.

Loading Photos

Load photos into your project using any of the following methods:

- Load photo files from disk.

- Load an Excel/XML file with a list of photos, their file paths, and all photo and photo collection information.

- Extract individual frames from video and save them as image files.

- Load a PhotoMesh project. When a PhotoMesh project (.PhotoMeshXML) is loaded into a second project, only the project's photos and photo and collection parameters are loaded.

Photos can only be loaded when Review mode is off. **See** "[About Reviewing Your Build](#)" in the "Reviewing Your Build" chapter for more information about Review mode.

Loading Photo Files from Disk

For optimal performance, photographs should be divided into one or more **collections**, homogeneous groups of photographs, all taken by the same physical camera with identical focal length and principal point.

To select photo files from your local disk:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select one of the following:
   - **Load Files** – To add selected files.
   - **Load Folder** – To add an entire folder.
2. Browse to the required folder/file(s) and click **Open**.
3. If you are adding a folder of photos, PhotoMesh automatically creates a group (collection) in the project tree and places all photos under the folder in this group. If you are adding a folder with sub-folders, a dialog box is displayed asking if you want to create a project sub-group for each sub-folder. Click **Yes** to create sub-groups for the sub-folders.

   Note: If a folder is added with photos that were taken by different cameras or with different focal lengths and principal points, PhotoMesh will automatically place the photos in different collections as long as the required metadata (e.g., Exif) is encoded in the photo files.

Defining Collection Properties with collectionProperties.xml

You can define collection properties and override those included in the metadata (e.g., Exif) encoded in photo files by creating a file named collectionProperties.xml with the required collection properties. This saves you the trouble of creating a full Excel or XML file that contains a list of all the photos. The collectionProperties.xml file is commonly used to add band information, focal length, or other collection level properties. Each collectionProperties.xml file can only contain data for a single collection, so a separate file must be created for each collection for which you want to provide supplementary information and placed in the corresponding collection folder.

There are several mandatory properties in the collectionProperties.xml file. If you want to use the values in the metadata encoded in the photo files, for these properties, set the required property in the collectionProperties.xml file equal to 0, e.g. "Width = 0".

A sample collectionProperties.xml follows below. Property descriptions are in the comments <! ….>.

```xml
<?xml version="1.0" encoding="UTF-8"?>
<BlocksExchange version="2.2">


<!-- Block definition -->
<Block>


<Collections>
<!-- A collection is a homogeneous group of photographs, all taken by the
same physical camera with identical focal length and principal point.-->
<Collection>
<!-- Name of the collection -->
<Name>NIR</Name>


<!-- (Mandatory) Width and height of the photos, in pixels. If you do not
want the collectionProperties.xml to override the width or height
information in the metadata encoded in the photo files, set their values
equal to 0. -->
<ImageDimensions>
<Width>0</Width>
<Height>0</Height>
</ImageDimensions>


<!-- Optional Parameters. Don't include if you do not want the
collectionProperties.xml to override the information in the metadata
encoded in the photo files. -->


<!-- (Optional) Bands in comma delimited string format, e.g.,
(Red,Green,Blue,NIR). Possible values are: Grayscale, Red, Green, Blue,
NIR, InfraRed, RedEdge, Alpha, Pan, SWIR_1, SWIR_2, TIR_1, TIR_2, Costal,
Custom_0, Custom_1, Custom_2, Custom_3, Custom_4, Custom_5, Custom_6,
Custom_7, Custom_8, Custom_9, Exclude (This band will be ignored.) -->
```

```
<Bands>Red,Green,Custom_1</Bands>


<!-- (Optional) Focal length in millimeters. -->
<FocalLength>80.0</FocalLength>


<!-- (Optional) Size of a physical pixel on the camera sensor in
millimeters. -->
<PixelSize>0.0052</PixelSize>


<!—(Optional) Origin and units of the principal point. The possible values
are: topleftpixels topleftmm centerpixels centermm -->
<PrincipalPointReferenceMethod>centerpixels</PrincipalPointReferenceMethod
>


<!— (Optional) PrincipalPoint x and y values, in the units set by
PrincipalPointReferenceMethod. -->
<PrincipalPoint>
    <x>7360</x>
    <y>4912</y>
</PrincipalPoint>


<!—Use yaw, pitch, roll values for orientation -->
</UseYPR>


<!-- (Optional) Offset along the Phi axis in degrees.-->
<PhiOffset>90</PhiOffset>


<!-- (Optional) Offset along the Kappa axis in degrees.-->
<KappaOffset>-180</KappaOffset>


<!-- (Optional) Offset along the Omega axis in degrees. -->
<OmegaOffset>0.0</OmegaOffset>


<!-- (Optional) Offset from the current altitude value in the coordinate
system units. If Lat-Long, use meters. -->
<AltitudeOffset>-10.0</AltitudeOffset>


<!-- (Optional) Flips the Phi axis.-->
<FlipPhi>false</FlipPhi>


<!-- (Optional) Flips the Kappa axis.-->
```

```
<FlipKappa>false</FlipKappa>


<!-- (Optional) Flips the Omega axis.-->
<FlipOmega>false</FlipOmega>


<!-- (Optional) Boolean that determines whether to include this collection
in the point cloud.-->
<UseInDpc>false</UseInDpc>


<!-- (Optional) Boolean that determines whether to include this collection
in the textured model.-->
<UseInModelTexture>true</UseInModelTexture>


<!-- (Optional) Boolean that determines whether to include this collection
in the orthophoto. -->
<UseInOrthoTexture>true</UseInOrthoTexture>


<!-- (Optional) Radial and tangential distortion coefficients calculated
using the Brown-Conrady model. See
https://en.wikipedia.org/wiki/Distortion_(optics)#Software_correction for
more information. -->


<Distortion>

   <!-- (Optional) Defines the 1st radial distortion coefficient. -->
   <K1>0.0</K1>
   <!-- (Optional) Defines the 2nd radial distortion coefficient. -->
   <K2>0.0</K2>
   <!-- (Optional) Defines the 3rd radial distortion coefficient. -->
   <K3>0.0</K3>
   <!-- (Optional) Defines the 1st tangential distortion coefficient. -->
   <P1>0.0</P1>
   <!-- (Optional) Defines the 2nd tangential distortion coefficient. -->
   <P2>0.0</P2>
</Distortion>


</Collection>
</Collections>
</Block>
</BlocksExchange>
```

To define collection properties using a collectionProperties.xml file:

1. Place the collectionProperties.xml file in the corresponding collection folder.
2. Load the collection folder. **See** "Loading Photo Files from Disk" in this chapter for information.

Loading a Photo List File

Instead of manually entering all photo and photo collection information (e.g., sensor width, focal length, and aerotriangulation data), you can load an Excel or ContextCapture XML file that references the photos and includes this data. You also have the option to load a CSV file that includes more limited photo information: only XYZ and OPK/YPR data. The photo list file can reference photos stored locally or on an HTTP or HTTPS server, or Amazon S3. The path in the photo list file can be relative or absolute, e.g., *s3://skyline-data/Frederick_xlsx/CF012316.tif* OR *http://www.skylineglobe.com/Photos_Lidars/Photos/tux-1771.jpg*

Note: Photos that are stored on HTTP or HTTPS servers, or Amazon S3 can **only** be loaded using a photo list file. In the photo list file, the photos' path should point directly to the photos' path on the server or S3. The photo list file itself, must be located on and loaded from a local or shared drive.

A sample Excel (PhotoMeshSample.xlsx) file is included in the installation folder, which can serve as a template for creating such a file as well as provide basic property definitions (in the Excel's notes). You can also see "Collection Property Sheet" and "Photo List" in this chapter for a description of the Excel/XML/CSV parameters.

The Excel file should include the following sheets:

- The first sheet named Collections should detail the group properties for each of the "collections", i.e., groups of photos. For optimal performance, each collection should include only photographs taken by the same physical camera with identical focal length and principal point. These group properties are the same as the properties that are listed in the Collection property sheet. **See** "Collection Property Sheet" in this chapter. Each row after the heading row (with all the property names) represents a different collection.

- A separate sheet for each of the collections, listing the properties of each photo in the collection. The name of each collection sheet must be identical to the collection name defined in the Name field of the Excel's Collections sheet. The photo properties are the same as the properties in the Photo property sheet. **See** "Photo List" in this chapter.

Excel File – Collection Property Sheet

Path	X	Y	Altitude	Omega	Phi	Kappa
C:\skyline\data\Frederick\\Images\13_2_239_A.jpg	-77.41471	39.4117622	546.20636	-1.57382	-40.4607	84.137
C:\skyline\data\Frederick\\Images\13_2_240_A.jpg	-77.41468	39.4125328	546.6153	2.948473	-44.11862	88.372
C:\skyline\data\Frederick\\Images\13_2_243_A.jpg	-77.41468	39.4149437	543.0982	-4.01135	-45.51225	84.531
C:\skyline\data\Frederick\\Images\13_2_244_A.jpg	-77.41471	39.4157143	540.58136	-1.84912	-38.56183	87.436
C:\skyline\data\Frederick\\Images\13_2_245_A.jpg	-77.4147	39.4165268	539.212646	-3.47045	-47.07546	84.857
C:\skyline\data\Frederick\\Images\14_2_259_A.jpg	-77.41569	39.4117279	497.463959	-3.54981	-44.754	87.436
C:\skyline\data\Frederick\\Images\14_2_260_A.jpg	-77.41571	39.4125481	496.3526	-4.06477	-44.60705	86.866
C:\skyline\data\Frederick\\Images\14_2_261_A.jpg	-77.41579	39.41332	469.36673	-0.52508	-50.41594	87.792
C:\skyline\data\Frederick\\Images\14_2_262_A.jpg	-77.41577	39.4141273	493.204834	-3.02948	-44.9635	87.1
C:\skyline\data\Frederick\\Images\14_2_263_A.jpg	-77.4157944	39.4149323	493.079224	-2.34766	-45.31397	87.602
C:\skyline\data\Frederick\\Images\14_2_264_A.jpg	-77.41583	39.4157372	492.870483	-2.31804	-45.10428	87.605

Excel File – Photo Property Sheet for Specific Collection

```
Collection,Path,Y,X,Z,yaw,pitch,roll
1,lower_left_DSC04405.JPG,-19.2812836,-48.4160783,862.72,187.22,-9.47,-2.57
1,lower_right_DSC04405.JPG,-19.2812836,-48.4160783,862.72,187.22,-9.47,-2.57
1,upper_left_DSC04405.JPG,-19.2812836,-48.4160783,862.72,187.22,-9.47,-2.57
1,upper_right_DSC04405.JPG,-19.2812836,-48.4160783,862.72,187.22,-9.47,-2.57
1,lower_left_DSC04406.JPG,-19.2814442,-48.4161162,863.24,186.84,-8.4,-2.38
1,lower_right_DSC04406.JPG,-19.2814442,-48.4161162,863.24,186.84,-8.4,-2.38
1,upper_left_DSC04406.JPG,-19.2814442,-48.4161162,863.24,186.84,-8.4,-2.38
1,upper_right_DSC04406.JPG,-19.2814442,-48.4161162,863.24,186.84,-8.4,-2.38
1,lower_left_DSC04407.JPG,-19.2815949,-48.4161432,863.61,188.29,-7.58,-1.93
1,lower_right_DSC04407.JPG,-19.2815949,-48.4161432,863.61,188.29,-7.58,-1.93
1,upper_left_DSC04407.JPG,-19.2815949,-48.4161432,863.61,188.29,-7.58,-1.93
1,upper_right_DSC04407.JPG,-19.2815949,-48.4161432,863.61,188.29,-7.58,-1.93
1,lower_left_DSC04408.JPG,-19.2817591,-48.4161703,863.88,189.58,-7.58,-0.51
1,lower_right_DSC04408.JPG,-19.2817591,-48.4161703,863.88,189.58,-7.58,-0.51
1,upper_left_DSC04408.JPG,-19.2817591,-48.4161703,863.88,189.58,-7.58,-0.51
1,upper_right_DSC04408.JPG,-19.2817591,-48.4161703,863.88,189.58,-7.58,-0.51
2,lower_left_DSC04409.JPG,-19.2819152,-48.4161977,864.35,189.86,-6.37,-0.62
2,lower_right_DSC04409.JPG,-19.2819152,-48.4161977,864.35,189.86,-6.37,-0.62
2,upper_left_DSC04409.JPG,-19.2819152,-48.4161977,864.35,189.86,-6.37,-0.62
2,upper_right_DSC04409.JPG,-19.2819152,-48.4161977,864.35,189.86,-6.37,-0.62
2,lower_left_DSC04410.JPG,-19.2820761,-48.4162257,864.59,191.08,-7.19,0.47
2,lower_right_DSC04410.JPG,-19.2820761,-48.4162257,864.59,191.08,-7.19,0.47
2,upper_left_DSC04410.JPG,-19.2820761,-48.4162257,864.59,191.08,-7.19,0.47
2,upper_right_DSC04410.JPG,-19.2820761,-48.4162257,864.59,191.08,-7.19,0.47
```

CSV File – Photo Properties for Multiple Collections

To add photo files using an Excel or XML file:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select **Load from List**. The Load a Photo List File dialog box is displayed.
2. Browse to the Excel or XML file. The photo files are loaded into the project.

To add photo files using a CSV file:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select **Load a Photo List File**. The Load a Photo List File dialog box is displayed.
2. Browse to the CSV file. The CSV/TXT Importer dialog box is displayed.

CSV/TXT Importer

3. Enter the row at which to start import, and select a **Delimiter**.

4. In the **Mapping Table**, select an **Orientation Method**.

5. Select the CSV column that corresponds to each of the photo properties.

6. Click **Import**.

Loading Photos from a Video

PhotoMesh can extract individual frames from video and save them as photo files. Supported video formats include: MPEG, AVI, MOV, and WMV.

To load photos from a video:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select **Load Video (FMV)**. The Load Video dialog box is displayed.

2. Browse to the video file, and click **Open**. The Load Video Editor is displayed.

Load Video Editor

3. Enter the following information:

Start	Start time from which video frame extraction should begin. Type in the time or drag the Start marker of the Video Cropping bar to the required position.
End	End time for video frame extraction. Type in the time or drag the End marker of the Video Cropping bar to the required position.
Frame Every	Sampling interval, i.e., every how many seconds a frame should be captured.
Focal Length Normalization	Camera properties used to set the focal length. Select one of the following options: ▪ Focal length and sensor width ▪ Focal length and pixel size ▪ Focal length 35
Focal Length	Focal length in millimeters.
Sensor Width	Size of the camera sensor in millimeters.
Pixel Size	Size of a physical pixel on the camera sensor in millimeters.
Zoom	How the video was captured: ▪ Fixed - Select if the video was captured with a fixed zoom. In this case, PhotoMesh will ensure that all intrinsic parameters for all the video frames are the same.

> ▪ Not Fixed - Select if the zoom level varied over the course of the capture, e.g. by using auto-zoom. In this case, PhotoMesh will calculate intrinsic parameters per frame.

4. Click **Load**. The video frames are loaded into the project as photo files into a single photo collection.

Loading a PhotoMesh Project

When a PhotoMesh project (.PhotoMeshXML) is loaded into a second project, only the project's photos and photo and collection parameters are loaded. Other project information (e.g. terrain objects, control points, and other project parameters) is not loaded.

To load a PhotoMesh project:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select select **Load a Photo List File**. The Load a Photo List File dialog box is displayed.
2. Browse to the required PhotoMesh project file. The project's photos and photo and collection parameters are loaded into the project.

Importing Photos with Metadata

Import photos of different data types into the project with their metadata (e.g., sensor width, focal length, and aerotriangulation data). When importing Inpho or Bingo results in a **Cartesian** coordinate system, such as ENU or ECEF, the AT step of the build process can be skipped (by selecting "Fully trusted input," in the [Camera Position Accuracy](#) AT setting). Imported AT data in a **projected** coordinate system such as UTM, however, may be unreliable either due to earth curvature or earth curvature corrections performed in the other system. This is particularly true for datasets that contain oblique photos or large frames, or cover a large geographic area. For data of this type, aerotriangulation should be re-performed in PhotoMesh, i.e., use Reliable instead of Fully trusted for the [Camera Position Accuracy](#) AT setting.

- [Importing Inpho Data](#)

- [Importing Bingo Data](#)

- [Importing Stellacore Data](#)

Importing Inpho Data

When importing Inpho results, the AT step of the build process can sometimes be skipped. **See** "[Importing Photos with Metadata](#)" in this chapter for more information.

To import Inpho Data:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select **Import Tool**. The Import Tool dialog box is displayed.

   Note: The fields displayed vary depending on the type of data selected.

Import Tool

2. Enter the following information:

Field	Description
Data Type	Select the data type.
Import Project File	Click **Browse.** Then in the Import Project File dialog box, browse to the required file, and click **Open**.
Image Folder	Click **Browse.** Then in the Import Image Folder dialog box, browse to the required folder, and click **Select Folder**.
Image Search Method	Select one of the following options: ▪ Recursive – To include image files in all subdirectories ▪ Top Directory Only – To include image files only from the top level directory

3. Click **Import**.

Importing Bingo Data

When importing Bingo results, the AT step of the build process can sometimes be skipped. **See** "Importing Photos with Metadata" in this chapter for more information.

To import Bingo Data:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select **Import Tool**. The Import Tool dialog box is displayed.

   Note: The fields displayed vary depending on the type of data selected.

Import Tool

2. Enter the following information:

Field	Description
Data Type	Select the data type.
Import Project File	Click **Browse.** Then in the Import Project File dialog box, browse to the required file, and click **Open**.
Image Folder	Click **Browse.** Then in the Import Image Folder dialog box, browse to the required folder, and click **Select Folder**.
Image Search Method	Select one of the following options: ▪ Recursive – To include image files in all subdirectories ▪ Top Directory Only – To include image files only from the top level directory
Angle Units	Select the data's angle units (Gons or Degrees).
Rotation Order	Select the rotation order (Phi-Omega-Kappa or Omega-Phi-Kappa).
WKT/ESPG	Click **Set** to set the data's coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information.
Camera Definition	For each camera in the BINGO project file, enter the following information: ▪ **Camera ID** – Camera name as it appears in the BINGO project file. This field is automatically filled in after the Import Project File is selected.

- **Columns** – Number of pixel columns in a photo taken with this camera (image width).

- **Rows** – Number of pixel rows in a photo taken with this camera (image height).

- **PixelSize** – Camera sensor's pixel size.

- **GridFile (Optional)**: File with camera distortion information that may be supplied by your data provider together with the BINGO file. This information can be applied during import to reduce image distortion. Click the button to open the Select Grid Calibration File dialog box. Then browse for the grid file.

3. Click **Import**.

Importing Stellacore Data

To import Stellacore Data:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Photos**, and select **Import Tool**. The Import Tool dialog box is displayed.

   Note: The fields displayed vary depending on the type of data selected.
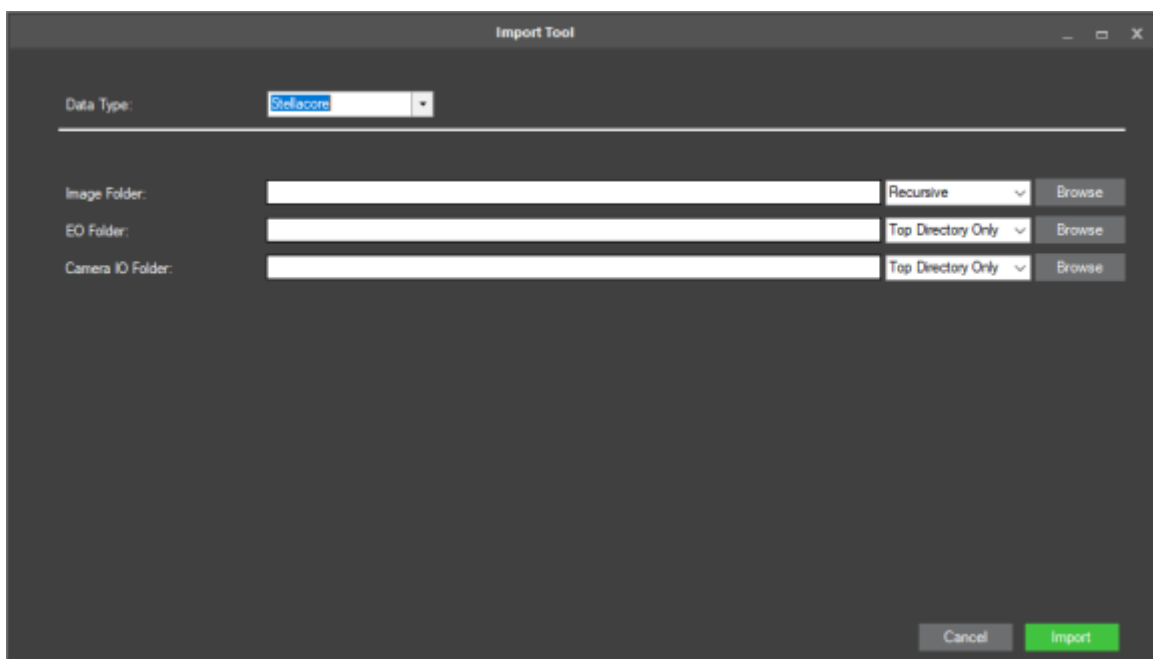
Import Tool

2. Enter the following information:

Field	Description
Data Type	Select the data type.
Image Folder	A folder containing the images. Click **Browse.** Then in the Select Image Folder dialog box, browse to the required folder, and click **Select Folder**.

112

	Select one of the following search options: ▪ Recursive – To include image files in all subdirectories ▪ Top Directory Only – To include image files only from the top level directory
EO Folder	A folder containing the external orientation information. Click **Browse.** Then in the Import EO Base Folder dialog box, browse to the required folder, and click **Select Folder**. Select one of the following search options: ▪ Recursive – To include files in all subdirectories ▪ Top Directory Only – To include files only from the top level directory
Camera IO Folder	A folder containing the camera's internal orientation information. Click **Browse.** Then in the Import Camera IO Base Folder dialog box, browse to the required folder, and click **Select Folder**. Select one of the following search options: ▪ Recursive – To include files in all subdirectories ▪ Top Directory Only – To include files only from the top level directory

3. Click **Import**.

Excluding a Photo

To exclude a photo from the project:

Note: Photos can only be excluded when Review mode is off. **See** "About Reviewing Your Build" in the "Reviewing Your Build" chapter for more information about Review mode.

1. In the **Project Tree**, select the required photo collection, and then in the **Photo List**, select the required photo.
2. On the **Photo** tab, in the **Selection** group, click **Exclude**. When a photo is excluded, the Exclude check box is selected in the photo list.

Deleting a Photo

To delete a photo from the project:

Note: Photos can only be deleted when Review mode is off. **See** "About Reviewing Your Build" in the "Reviewing Your Build" chapter for more information about Review mode.

Note: When a photo is **deleted** from a project, the photo is no longer available for inclusion within the same project.

1. In the **Project Tree**, select the required photo collection, and then in the Photo List, select the required photo.
2. On the **Photo** tab, in the **Selection** group, click **Delete**. The photo is deleted from the photo list.

Modifying Photo and Photo Collection Properties

Photo data can be viewed and modified in the photo collection property sheet and photo list. Information displayed in the property sheet refers to the collection that is currently selected in the collection list, while information in the photo list is specific to each photo.

Information about the photo collections and photos (e.g., coordinate system, sensor width, focal length, photo position information) is generally read from a loaded Excel or XML file that includes this information. These parameters can then be adjusted in the collection property sheet and photo list.

To display and modify photo collection properties:

1. In the **Project Tree**, select the collection, and click the **Properties** tab.
2. In the collection property sheet, modify the property value as required. **See** "Collection Property Sheet" in this chapter for more information.

To display and modify photo properties:

1. In the **Project Tree**, select the collection, and click the **Photo List** tab.
2. In the **Photos** list, select the required photos, and modify their properties as required. **See** "Photo List" in this chapter for information.

   Note: You can also search the list for the required photos. **See** "Selecting Items Using the Item List" in the "Basic Concepts" chapter for information.

Collection Property Sheet

Properties	📌
Quick Search	
⊿ Collection	
Name	CAM3
Calculate Camera Parameters	Entire Collection
Coordinate System	WGS 84 + EGM96 geoid height
Ignore Orientation	☐
Orientation Method	Omega Phi Kappa
⊿ Camera	
Focal Length Normalization	Focal length and sensor width
Sensor Width	35.9168
Pixel Size	0.0048800000000000007
Focal Length	84.9112465541587
Focal Length 35	
Principal Point Reference	Top left in pixels
Principal Point X	3640.0301108239541
Principal Point Y	2485.9500336083788
Width	7360
Height	4912
Gamma	1
Effective Range	
Calculated Effective Range	869.47053554680951
Crop Area	100 %
Use In Textured Model	☑
Use In Orthophoto	☑
Use In Point Cloud	☑
⊿ Bands	
⊿ Hyperspectral Bands	Red,Green,Blue
1	Red
2	Green
3	Blue
⊿ Camera Distortion	
K1	0.044001660034628193
K2	0.19618285711837311
K3	3.8859124931693629
P1	0
P2	0
⊿ Advanced Position	
Altitude Offset	
Omega Offset	
Phi Offset	
Kappa Offset	
Flip Omega	☐
Flip Phi	☐
Flip Kappa	☐
Rotate Yaw	
Rotate Pitch	
Rotate Roll	

Property	Description
Collection	
Name	Name of the collection in the Project Tree.
Calculate Camera Parameters	Method of calculating the camera's intrinsic parameters.
	▪ Entire collection – Calculate the AT so that the camera intrinsic properties are the same for all photos in the collection.
	▪ Per photo – Calculate the camera intrinsics individually for each photo in the collection.
	Note: Parameters should be calculated per photo when not all of a collection's photos were taken by the same physical camera with identical focal length and principal point.
Coordinate System	Coordinate system of the collection. Click Select 🔲 to set a different coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information.
Ignore Orientation	Ignore the orientation value.
Orientation Method	Select which system to use to set orientation: OPK (Omega, Phi, Kappa) or YPR (Yaw, Pitch, and Roll). This selection determines which fields are displayed in the Advanced Position section of the property sheet.
Camera	
Focal Length Normalization	Camera properties used to set the focal length.
	Select one of the following options:
	▪ Focal length and sensor width
	▪ Focal length and pixel size
	▪ Focal length 35
Sensor Width	Size of the camera sensor in millimeters.
Pixel Size	Size of a physical pixel on the camera sensor in millimeters.
Focal Length	Focal length in millimeters.
Focal Length 35	Equivalent focal length in millimeters for a 35 mm film camera.
Principal Point Reference	Origin and units of the principal point. Select either:
	▪ Top left in pixels – Positive x-direction is to the right and positive y-direction is downwards.
	▪ Top left in mm – Positive x-direction is to the right and positive y-direction is downwards.
	▪ Center in pixels – Positive x-direction is to the right and positive y-direction is upwards.
	▪ Center in mm – Positive x-direction is to the right and positive y-direction is upwards.

Principal Point X	Principal point X value in pixels or mm according to principal point reference.
Principal Point Y	Principal point Y value in pixels or mm according to principal point reference.
Width	Width of photo in pixels.
Height	Height of photo in pixels.
Gamma	Brightness of the photo's colors.
Effective Range	Maximum distance from the camera position for which photo information is considered valid for the aerotriangulation.
Calculated Effective Range	Whenever the Effective Range parameter value is not provided, PhotoMesh calculates the maximum distance from the camera for which photo information is considered valid for the aerotriangulation, and then displays and uses this value.
Crop Area	Percentage of the photo to use in the project. Blurry or obstructed areas on the edges of a photo will be cropped based on this property value.
Use in Textured Model	Select the check box to include this collection in the textured model.
Use in Orthophoto	Select the check box to include this collection in the orthophoto.
Use in Point Cloud	Select the check box to include this collection in the point cloud output.

Bands – Hyperspectral

Hyperspectral bands in the collection's photos. These bands can be used in texturing the 3D model and in generating the orthophoto. See "Setting Reconstruction Parameters" and "Setting Output Formats" in the "Building" chapter.

Note: The band information is retrieved from the collection information in the photo list Excel or XML, the collectionProperties XML, or from the first photo in the collection. **See** "Loading an Excel or ContextCapture XML Photo List File" and "Loading Photo Files from Disk" in the "Photo Management" chapter. You can manually set the value for each band in the property sheet.

1	Select the first band.
2	Select the second band.
3	Select the third band.

Camera Distortion (Optional)

Radial and tangential distortion calculated using the Brown-Conrady model. See https://en.wikipedia.org/wiki/Distortion_(optics)#Software_correction for more information.

K1	Defines the 1st radial distortion coefficient.
K2	Defines the 2nd radial distortion coefficient.
K3	Defines the 3rd radial distortion coefficient.
P1	Defines the 1st tangential distortion coefficient.
P2	Defines the 2nd tangential distortion coefficient.

Advanced Position (Either OPK or YPR offset and flip values will be displayed, depending on your selection above for Orientation Method)

117

Altitude Offset	Offset from the current altitude value.
Omega Offset	Offset along the Omega axis in degrees.
Phi Offset	Offset along the Phi axis in degrees.
Kappa Offset	Offset along the Kappa axis in degrees.
Flip Omega	Flips the Omega axis.
Flip Phi	Flips the Phi axis.
Flip Kappa	Flips the Kappa axis.
Yaw Offset	Offset along the Yaw axis in degrees.
Pitch Offset	Offset along the Pitch axis in degrees.
Roll Offset	Offset along the Roll axis in degrees.
Flip Yaw	Flips the Yaw axis.
Flip Pitch	Flips the Pitch axis.
Flip Roll	Flips the Roll axis.
Secondary Yaw Rotation	Additional rotation along the Yaw axis in degrees, that is performed in YPR order after the photos' OPK or YPR orientation and offsets were applied.
Secondary Pitch Rotation	Additional rotation along the Pitch axis in degrees, that is performed in YPR order after the photos' OPK or YPR orientation and offsets were applied.
Secondary Roll Rotation	Additional rotation along the Roll axis in degrees, that is performed in YPR order after the photos' OPK or YPR orientation and offsets were applied.

Photos List

In the Photos list, you can view and set photo properties, as well as set photo visibility and inclusion in the project. When calculated camera positions are displayed, you can also view calculated property information for a selected photo. **See** "About the Item List" in the "Using the Project Tree and Item List" chapter for information on working with the Photos list.

Photo List

To open the Photos list:

▪ In the **Project Tree**, select a **Photos** collection. The Photos list lists all the photo files in the collection, with the following information/options:

Property	Description
Selection	Select the photos on which to perform an action, e.g., create tie point, show/hide camera symbol, exclude, delete.
Collection	Name of the photo collection (Read-only).
Name	Name of the photo.
Visible	Select the check box to show the photo in the 3D Window.
Exclude	Select the check box to exclude the photo from the project. This option is not available in Review mode.
Project Original	Select the check box to project the photo on the terrain in its original camera position. A photo can be projected on the terrain to see if it fits the terrain imagery or to better understand the position of the photo. Only photos that have positioning (XY coordinates and altitude) and orientation (Omega, Phi, and Kappa) information can be projected. **Note:** Projecting multiple photos simultaneously can take several minutes.
X	East-west coordinate of the camera. (Required)
Y	North-south coordinate of the camera. (Required)
Altitude	Altitude of the camera in meters. (Required)
Omega	Omega Euler angle in degrees.
Phi	Phi Euler angle in degrees.
Kappa	Kappa Euler angle in degrees.

Yaw	Direction angle of the camera along the vertical axis relative to north.
Pitch	Tilt angle of the camera along its lateral axis relative to the horizon.
Roll	Roll angle of the camera along its longitudinal (front-to-back) axis.
Path	Full path of the photo.
K1	Defines the 1st radial distortion coefficient.
K2	Defines the 2nd radial distortion coefficient.
K3	Defines the 3rd radial distortion coefficient.
P1	Defines the 1st tangential distortion coefficient.
P2	Defines the 2nd tangential distortion coefficient.

Calculated Camera Position Information (Additional columns)

Note: If you want to view calculated information for a selected photo, click the **Calculated** button on the **Tile** tab**.**

Project Calculated	Select the check box to project the photo on the terrain in its calculated camera position.
Median Error	Median of the triangulation errors of all the features in the photo. The triangulation error is the distance, in pixels, for each of the features in the photo, between the automatically identified feature and its projected triangulation position.
X Calculated	Calculated east-west coordinate of the camera.
Y Calculated	Calculated north-south coordinate of the camera.
Altitude Calculated	Calculated altitude of the camera in meters.
Omega Calculated	Calculated Omega Euler angle in degrees.
Phi Calculated	Calculated Phi Euler angle in degrees.
Kappa Calculated	Calculated Kappa Euler angle in degrees.
K1 Calculated	Calculated 1st radial distortion coefficient.
K2 Calculated	Calculated 2nd radial distortion coefficient.
K3 Calculated	Calculated 3rd radial distortion coefficient.
P1 Calculated	Calculated 1st tangential distortion coefficient.
P2 Calculated	Calculated 2nd tangential distortion coefficient.

CHAPTER 11 LIDAR MANAGEMENT

About Lidar Management

Lidar (light imaging, detection and ranging) is a remote sensing method that uses pulsed laser light to measure terrain and objects. This data together with other information recorded by the sensor system is used to generate a densely spaced point cloud of highly accurate geo-referenced points.

Lidar data can be loaded into a PhotoMesh project and integrated with project photos to refine and enhance the 3D model, producing cleaner results and preventing artifacts, while increasing the level of accuracy of the complete PhotoMesh process:

- Lidar improves the precision of the mesh geometry, since Lidar has less noise-to-signal ratio than image-based correlation. This means smoother and cleaner roads, facades, and other planar surfaces.

- Lidar helps in areas that are very homogenous and don't have a lot of texture to produce image correlation points (wide roadways with no markings, white rooftops, repetitive vegetation like crops or grass, etc.).

- Lidar makes it easier to get into tight angles in urban areas, since Lidar only needs a single ray to reach the bottom of a building or inside an alley in order to have it measured, whereas with image correlation two images with enough angular disparity are required to reach a feature and to be able to correlate depth in that point.

- Lidar enhances street level data, providing points underneath vegetation and where trees get close to buildings.

- Lidar improves fine details of complicated features such as scaffolding, tops of towers, draw bridges, cranes, etc.

- Lidar provides a geographic control to achieve a high level of absolute accuracy in the final 3D mesh and additional geospatial products.

The Lidar data's proper alignment with the project's photos can be verified and corrected using ground control points and through projection of the photos on the Lidars. **See** "Projecting a Photo on the Terrain" in the "Preparing the Project" chapter and "Creating Ground Control Points from Identifiable Locations on the Terrain" in the "Control Points" chapter for more information.

This chapter deals with basic Lidar management tasks:

- Loading of Lidar data

- Modifying Lidar properties

- Associating a Lidar point cloud with its trajectory data

- Showing Lidar point cloud on terrain

- Navigating to a Lidar

- Excluding/deleting Lidar files from the project

Loading Lidar Data

Lidar data can be loaded into a PhotoMesh project and integrated with project photos to supplement project data and increase model accuracy. PhotoMesh supports Lidar data in .las, .laz or .e57 formats. To improve the reconstruction of the model, Lidar data can be loaded together with a trajectory file that provides information about the scanner's position while capturing the point cloud. Some .e57 files include internal trajectory information and therefore do not require an external one. Lidar are converted to CPT, PhotoMesh's internal format, during the build's data preparation step or by Showing Point Cloud. **See** "About Building" in the "Building" chapter and "Showing a Lidar Point Cloud on the Terrain" in this chapter for more information.

If Lidar files are loaded after aerotriangulation is performed and reconstruction has already started, all reconstruction tiles that intersect with the new Lidar's bounding box are restored to the point cloud step. This ensures that all necessary processing is performed when the Build Start From step is set to Auto. **See** "Setting Build Steps" in the "Building" chapter for more information.

To load Lidar files from your local disk:

1. On the **Home** tab, in the **Add** group, click the arrow next to **Lidar**, and select one of the following:

   ▪ **Load files** – To add selected files.

   ▪ **Load folder** – To add an entire folder. All supported Lidar files in the folder are loaded.

2. Browse to the required folder/file(s) and click **Open**.

3. Set the Lidar's coordinate system including its vertical datum.

   Note: If incorrect coordinate system information is entered, the *.CPT created for Lidar will need to be manually deleted from the project directory: PM_project\cpts\

4. Add a trajectory file and modify Lidar properties, as required, in the **Lidars** list. **See** "Modifying Lidar Properties" and "Associating a Lidar Point Cloud with its Trajectory Data" in this chapter for information.

Modifying Lidar Properties

Lidar data can be viewed and modified in the Lidars list and point cloud properties dialog box. This dialog box is accessed from a right-click menu as described below.

To display and modify Lidar point cloud properties:

1. In the **Project Tree** select the **Lidars** group, and then in the **Lidars** list, select the required Lidar.

2. For basic properties such as path and coordinate system, in the **Lidars** list, select the check boxes of the required Lidar, and modify their properties as required. **See** "Lidar List" in this chapter for more information.

   Note: You can also search the list for the required Lidar. **See** "Selecting Items Using the Item List" in the "Basic Concepts" chapter for information.

3. For point styling properties and other properties relating to how the point cloud is displayed, right-click the required Lidar, and select **Style Point Cloud**. The Point Cloud Properties dialog box is displayed. **See** "Lidar Point Cloud Properties Dialog Box" in this chapter for more information.

   Note: A Lidar must be converted to CPT, PhotoMesh's internal format before it can be styled. You can convert the Lidar by performing the build's data preparation step or by **Showing Point Cloud**. **See** "About Building" in the "Building" chapter and "Showing a Lidar Point Cloud on the Terrain" in this chapter for more information.

Lidar Point Cloud Properties Dialog Box

Property	Description
Style	
Point Size	Size, in pixels, of each of the Lidar point cloud's points. If you want to base the point size on attribute fields, click the **Field by Attribute** button ▦. In the Field by Attribute dialog that opens, select the attribute on which to base point size. Then for each attribute value, type the value and the corresponding point size.
Point Color	Color of the Lidar point cloud points. This can be set to either of the following: ▪ A set color value for all the points. To select a color, click **Edit** and select a color. ▪ Based on a specific attribute field - To base each point of the point cloud's color on a specific attribute field's value, click the **Field by Attribute** button ▦. In the Field by Attribute dialog that opens, select the attribute on which to base point color. Then for each attribute value, type the value and the corresponding point color, e.g., color each point of the point cloud according to the CPT's RGB attribute values or according to the CPT's intensity attribute values.
Point Visibility	Determines if the Lidar point cloud's points are visible. To set the visibility of points based on their attribute values, click the **Field by Attribute** button ▦. In the Field by Attribute dialog that opens, select the attribute on which to base visibility. Then for each attribute value, type the value and the corresponding visibility setting.
Intensity Blend	Determines how much the Lidar point cloud's intensity value is factored in colorizing the point cloud, where 0% colorizes the point cloud model by point color only, and 100% colorizes by intensity only.
Oversampling	The Lidar point cloud's Level of Detail (LOD). When set to Normal, PhotoMesh determines the optimal quality to performance balance based on the current view. The "High" and "Best" settings will load more points but are also more resource-intensive.
Flatten Underlying Terrain	Determines if the terrain under the model is leveled to the model's elevation value. This option is enabled only if the Altitude Method property in the point cloud property sheet is set to Absolute.
Hide Underlying Terrain	Hides the terrain texture under the Lidar point cloud model (the terrain appears as a black image).

Lidar List

In the Lidar list, you can view and set the Lidar's coordinate system, path and trajectory path, as well as set Lidar visibility and inclusion in the project. **See** "About the Item List" in the "Using the Project Tree and Item List" chapter for information on working with the Lidars list.

Lidar List

To open the Lidars list:

In the **Project Tree**, select the **Lidars** group. The Lidars list displays all the Lidar files in the project, and the following information:

Property	Description
Selection	Select the check box to select the Lidar in the 3D Window.
#	Number of the Lidar.
Excluded	Select the check box to exclude the Lidar point cloud from the project.
Box	Select the check box to show the Lidar's bounding box.
Name	Name of Lidar file.
Point Cloud	Select the check box to show a Lidar point cloud on the terrain.
Path	Full path to the Lidar point cloud.
Coordinate System	Coordinate system of the Lidar point cloud. Click Select ⬚ to set a different coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information.
Trajectory Path	Full path to file that provides information about the scanner's position while capturing the point cloud. **See** "Associating a Lidar Point Cloud with its Trajectory Data" in this chapter for more information on trajectory files.

124

Tag	Free text about the Lidar

Associating a Lidar Point Cloud with its Trajectory Data

Trajectory data provides information about the scanner's position while capturing the Lidar point cloud. The scanner positions should be ordered chronologically in the file.

Note: Some .e57 files include internal trajectory information and therefore do not require an external one.

Trajectory data can be imported in a tab delimited .trjt (text format) file that includes the following lines:

- First line: Define the Lidar version (Should always be 1.0)

```
#version=1.0
```

- Second line: Define the Lidar's coordinate system in WKT format.

```
wkt=……
```

  Note: The WKT information must be in a single line without any line breaks. You can use the Coordinate System Generator to generate a WKT string defining the required coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information.

- Third line (optional): Indicates a time offset that should be added to the GPS time field in each record (default is 0).

```
#timebase=
```

- For each known scanner position, add a separate line with the scanner's position (X, Y, Altitude) and the time the position was recorded (GPS Time) in tab delimited format. The scanner positions should be ordered chronologically.

```
[x coordinate] <TAB> [y coordinate] <TAB> [altitude] <TAB> [GPS time]
```

  Note: The GPS Time is the double floating point time tag value at which the scanner's position was recorded. This data must be in the same units and offset as the time data in the Lidar file.

Example:

```
#version=1.0
wkt=GEOGCS["WGS 84",DATUM["WGS_1984",SPHEROID["WGS
84",6378137,298.257223563,AUTHORITY["EPSG","7030"]],AUTHORITY["EPSG","6326
"]],PRIMEM["Greenwich",0,AUTHORITY["EPSG","8901"]],UNIT["degree",0.0174532
925199433,AUTHORITY["EPSG","9122"]],AUTHORITY["EPSG","4326"]]
#timebase=0

9.63390819201651 <TAB> 47.4233597051724 <TAB> 401.463806152344 <TAB>
583.161876711994
9.63390818337074 <TAB> 47.4233597019402 <TAB> 401.463793239556 <TAB>
583.161876711994
9.63390817652904 <TAB> 47.4233597069347 <TAB> 401.463349871528 <TAB>
583.161876711994
9.63390818337074 <TAB> 47.4233597019402 <TAB> 401.463407984571 <TAB>
```

```
583.160397257656
```

To associate a Lidar file with its trajectory file:

▪ In the **Project Tree** select the **Lidars** group, and then in the **Lidars** list, select the required Lidar and type the trajectory file's path in the **Trajectory Path** field.

Showing a Lidar Point Cloud on the Terrain

A Lidar can be shown on the terrain to see if it fits with the project's photos, other Lidar, ground control points, etc.

Note: The Lidar data's proper alignment with the project's photos can be verified and corrected using ground control points and through projection of the photos on the Lidars. **See** "Projecting a Photo on the Terrain" in the "Preparing the Project" chapter and "Creating Ground Control Points from Identifiable Locations on the Terrain" in the "Control Points" chapter for more information.

To show a Lidar on the terrain:

1. In the **Project Tree** select the **Lidars** group, and then in the **Lidars** list, select the **Point Cloud** check box for each of the Lidars you want to display. The Lidar (contextual) tab is displayed.

     Note: If a particular Lidar wasn't converted previously to CPT, PhotoMesh's internal format, it will first be converted and then shown on the terrain.

     Note: If you want to show more than one Lidar point cloud on the terrain, it is generally recommended to build the project, performing only the Data Preparation step since the conversion to CPT performed via View > Point Cloud may take a long time.

2. If you want to show a Lidar's trajectory data, select the Lidar and then on the **Lidar** tab, in the **View** group, toggle on **Trajectory**.

3. If you want to hide a Lidar's bounding box, select the Lidar and then on the **Lidar** tab, in the **View** group, toggle off **Bounding Box**.

Navigating to a Lidar

To fly or jump to a Lidar:

1. In the **Project Tree** select the **Lidars** group, and then in the **Lidars** list, select the required Lidar.

2. On the **Lidar** tab, in the **Navigation** group, select **Fly to**/ **Jump to**.

Excluding a Lidar File

To exclude a Lidar file from the project:

1. In the **Project Tree** select the **Lidars** group, and then in the **Lidars** list, select the required Lidar.

2. On the **Lidar** tab, in the **Selection** group, click **Exclude**. When a Lidar is excluded, its Excluded check box is selected in the Lidars list.

Deleting a Lidar File

To delete a Lidar from the project:

1. In the **Project Tree** select the **Lidars** group, and then in the **Lidars** list, select the required Lidar.

2. On the **Lidar** tab, in the **Selection** group, click **Delete**. The Lidar is deleted from the Lidars list.

CHAPTER 12 PREPARING THE PROJECT

About Preparing the Project

After photos have been loaded, do any of the following to prepare for the build:

Set display of photos

- Select the photos for which symbols are displayed

- Select the size and color coding of the camera symbols that display for each photo

Study photos and modify photo properties

- Navigate to a photo

- View photos in the photo viewer

- Project photos on the terrain to see if they fit the terrain imagery or to better understand their position

- Show a color-coded map analyzing photo coverage in every point of the AT area

- Show a color-coded connection map that indicates the level of photo interconnection

- Modify photo and photo collection properties (in "Photo Management" chapter)

- Use ground control points to position the model by associating specific locations (x, y, altitude) on the terrain with corresponding locations (in pixels) in three or more photos (in "Control Points" chapter)

- Use tie points to establishing a correspondence between project photos and improve aerotriangulation results through identification of the same physical point in three or more photos (in "Control Points" chapter)

Generate a flat surface with correct texturing over a user-defined water body

- Create a water body polygon

- Import a water body polygon

Set the aerotriangulation and reconstruction areas

- Set the AT area

- Set the reconstruction area

Displaying and Studying Photos

Displaying Camera Symbols

Each photo's camera position and orientation (if known) can be displayed on the terrain using camera symbols. The symbol's color also indicates either the photo's collection or its status ((in original position, in calculated position, failed to calculate, or excluded by user). You can show all photos or you can filter which photos are displayed:

- Display only the photos individually selected in the Photos list or the collections selected in the Project Tree.

- Display only photos that cover a selected point on the terrain.

- Display only photos that were not excluded from the project.

To select specific photos from the Project Tree for viewing on the terrain, do any of the following:

- In the **Project Tree**, select the check boxes of the collections you want to display. All of the selected collections' photos will be displayed.

- In the **Project Tree**, select the required photo collection, and then in the **Photos** list, select the **Visible** check box of the required photos.

- In the **Project Tree**, select the required photo collection, and then in the **Photos** list, select the required photos, and on the **Photo** tab, in the **Selection** group, toggle off **Hide Symbol**.

To select photos for viewing that cover a selected point on the terrain, do the following:

1. Do either of the following:
    - If you want to display all photos that intersect with a selected point based on original photo positions, on the **Home** tab, in the **Photos** group, click the arrow under **Show Photos in Point** and select **Show Original Photos in Point**.
    - If you want to display all photos that intersect with a selected point based on aerotriangulation calculations, on the **Home** tab, in the **Photos** group, click the arrow under **Show Photos in Point** and select **Show Calculated Photos in Point**.

2. In the **3D Window**, click the point of interest. All photos that cover the selected point are displayed.

3. To (re)display all photos, on the **Home** tab in the **Photo** group, click **Show All Photos**.

To display only photos that were not excluded from the project:

1. On the **View** tab, in the **Photos** group, select **Do not show excluded**. All excluded photos and photos outside the currently defined AT area are hidden. **See** "Excluding a Photo" in the "Photo Management" chapter for information about excluding photos.

2. To (re)display excluded photos, on the **View** tab, in the **Photos** group, select **Show excluded**.

Selecting Camera Symbol Size and Color Coding

To select a camera symbol size:

- On the **View** tab, in the **Photos** group, in the **Symbol size** drop-down, select a size for the camera symbols that display for each photo.

To select a camera color coding option:

- On the **View** tab, in the **Photos** group, in the **Photo color** drop-down, select either of the following:
    - **Color by status** – Camera symbols are color coded based on photo status (in original position, in calculated position, failed to calculate, or excluded by user), according to the colors defined in Options' settings. **See** "Setting PhotoMesh Options" in the "Basic Concepts" chapter for more information.
    - **Color by collection** – Camera symbols are color coded according to collection, with a random color assigned to each collection's camera symbol.

Navigating to a Photo

To fly or jump to a photo:

1. In the **Project Tree**, select the required photo collection, and then in the **Photos** list, select the required photo.

2. On the **Photo** tab, in the **Navigation** group, select **Fly to**/ **Jump to**.

   Note: You can also fly to the selected photo by double-clicking it in the **Photos** list.

Viewing Photos in the Photo Viewer

To view a photo in the Photo Viewer:

1. In the **Project Tree**, select the required photo collection, and then in the **Photos** list, select the required photo. Only one photo can be viewed in the Photo Viewer at a time. The Photo tab is displayed.

2. In the **View** group, click **Photo Viewer**. The Photo Viewer opens with the selected photo displayed.

3. Use the Photo Viewer ribbon commands to zoom in and out or Fit to Window.

4. If you want to see how many other photos have each of the features automatically identified in this photo, click **Show Features.** The photo's features are color coded according to the number of other photos in which they appear:

   - **Gray** – Feature wasn't found in any other photo (no match)
   - **Yellow** – Feature was found in one other photo
   - **Green** – Feature was found in two or more photos

   Note: This information is only available in Review mode after aerotriangulation was completed. **See** "Setting Build Parameters and Processing the Build" in the "Building" chapter. Build steps after aerotriangulation do not need to be completed to view this information.

Show Features

Projecting a Photo on the Terrain

A photo can be projected on the terrain (including Lidar) to see if it fits the terrain imagery or to better understand the position of the photo. The projection of the photo can be based either on its original position or its calculated position. Only photos that have positioning (XY coordinates and altitude) and orientation (Omega, Phi, and Kappa) information can be projected based on original positions.

To project a photo on the terrain in its original camera position:

1. In the **Project Tree**, select the required photo collection, and then in the **Photos** list, select the required photo. The Photo (contextual) tab is displayed.
2. On the **Photo** tab, in the **View** group, select **Project Original**. The photo is shown on the terrain, and the photo's Project Original check box is selected in the Photos list.

To project a photo on the terrain from the camera position calculated through aerotriangulation:

1. On the **Home** tab, in the **Photos** group, click the arrow under **Show Photos in Point** and select **Show Calculated Photos in Point**.
2. In the **3D Window**, click the point of interest. All photos that cover the selected point are displayed.
3. In the **3D Window**, select the required photo. The Photo (contextual) tab is displayed.
4. On the **Photo** tab, in the **View** group, select **Project Calculated.** The photo is shown on the terrain, and the photo's Project Calculated check box is selected in the Photos list.

Clearing Review Data from the Terrain

To clear review data:

- On the **Home** tab, in the **Review** group, click **Clean Review Data**. This clears any photo projections from the terrain.

Showing the Coverage Map

The coverage map is a color-coded map displaying photo coverage in every point of the defined AT area. This enables you to identify which areas of the project have sufficient coverage and which require more. The AT area can be defined to include all the project's photos or restricted to only the photos within a specific area on the terrain. **See** "Setting the AT (Aerotriangulation) Area" in this chapter for more information.

The map can display coverage based on either the original positions of the project's photos or their calculated positions.

Note: The coverage map may take some time to display the first time it is shown or after making changes to the project as PhotoMesh (re)calculates the photo coverage.

To show a coverage map:

- On the **Home** tab, in the **Photos** group, click the arrow next to **Coverage Map**. and select one of the following:
  - Coverage Map from Original
  - Coverage Map from Calculated
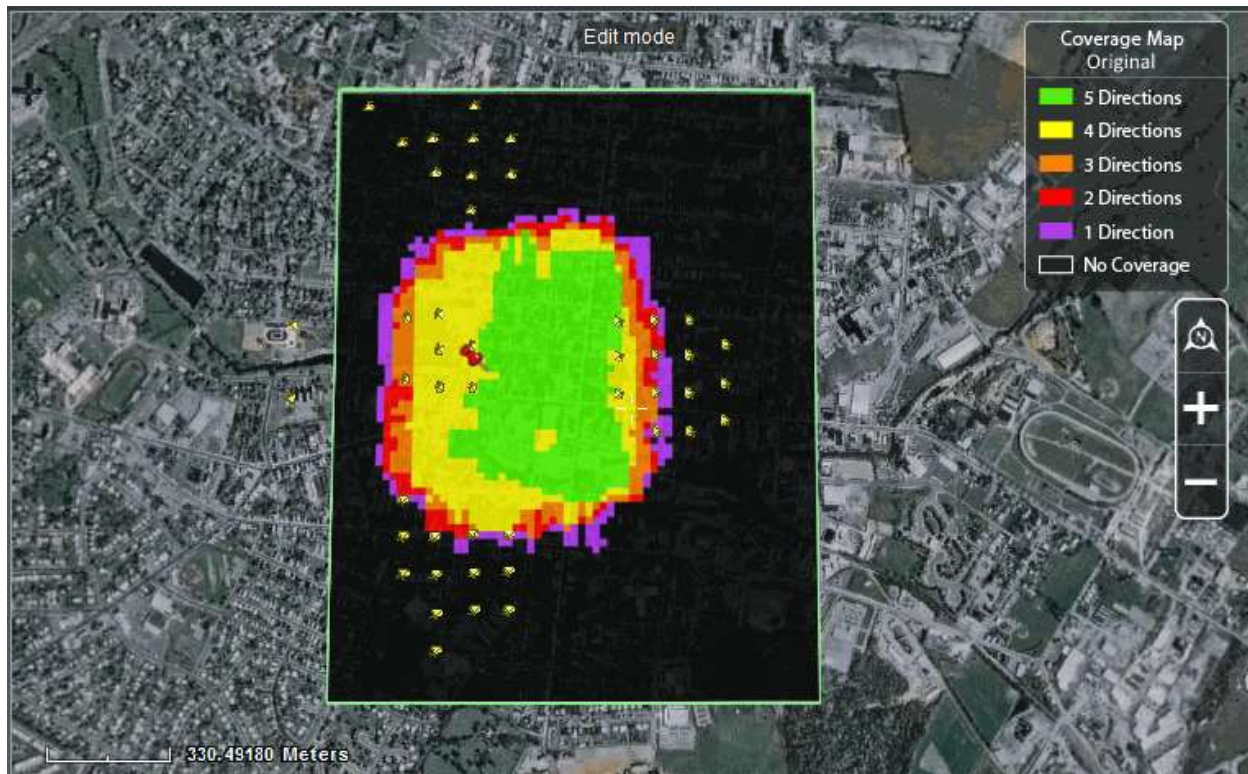
Coverage Map and Legend

In the coverage map, the AT area is color coded based on directionality and coverage.

- **Green** – Best coverage from all 5 directions: nadir and 4 oblique.

- **Yellow** – Sufficient coverage of 4 directions.

- **Orange** – Three coverage directions, the minimum required.

- **Red** – Two coverage directions.

- **Purple** – One coverage direction.

- **Black** – No coverage.

-

Showing the Connection Map

The connection map is a color-coded map that enables you to evaluate the quality of the AT. In the connection map, each point in the defined AT area is colored based on the level of interconnection of the photos that intersect with this point. A group of photos is considered interconnected if each photo in the group was directly or indirectly matched with every other photo. Even photos that do not precisely intersect with the point in the defined AT area can be used to create an indirect connection, as long as they come close to intersecting with the point (i.e., if photos A and D precisely intersect with the point in the AT area, and photos B and C nearly intersect with the point, photo A can be connected to photo D via photos B and C, where A intersects with B which intersects with C, which intersects with D).

▪ Green – All of the photos that intersect with this point are interconnected.

▪ Red – 50% of the photos that intersect with this point are interconnected

Each point in the defined AT area is colored on the gradient between green and red, based on the percentage of photos that intersect with the point.

If the connection map indicates that insufficient photo matches were generated for certain points in the AT area, tie points can be used to manually create matches for these points. See "About Control Points" in the "Control Points" chapter for information.

You can also display the number of matches for each photo by selecting Number of Matches as the color coding option for the photos. See "Displaying Calculated Camera Positions for Tiles" in the "Reviewing Your Build" chapter for more information.

To show a connection map:

▪ On the **Home** tab, in the **Photos** group, click the arrow next to **Coverage Map**. and select **Connection Map from Calculated**.

Setting the AT (Aerotriangulation) Area

Setting the AT area defines the area of the project on which PhotoMesh should perform aerotriangulation. Aerotriangulation is the process for determining the correct position and orientation of each of the project photos, based on inputted photo information, ground control points and tie points. If a project's photos do not have fully trusted camera positioning and orientation information, this process must be performed before generation of the 3D mesh model. PhotoMesh uses the highest-performance photogrammetry, computer vision, and computational geometry algorithms in performing aerotriangulation.

The defined AT area can include all the project's photos or be restricted to only the photos within a specific area on the terrain. This enables you to perform aerotriangulation on a small subset of the project to validate camera parameters and build settings and check the resulting AT and reconstruction tiles before building the full project. The area for the aerotriangulation can be defined by either drawing or importing an AT polygon.

If project photos have no positioning information, you should manually set an initial AT area based on an estimation of the geographical area of the project, which will then be adjusted by PhotoMesh in the aerotriangulation step.

There are two indicators in the 3D Window that photos are outside the current aerotriangulation area:

▪ When photos are "colored by status" (**See** "Selecting Camera Symbol Size and Color" in this chapter), the camera symbols for these photos will be colored in grey.

▪ When "Do not show excluded" is selected (**See** "Displaying Camera Symbols" in this chapter), the camera symbols for these photos are not displayed.

Note: Photos that are outside the currently defined AT area are still considered part of the project and therefore the Exclude check box remains cleared in the Photos list.

Setting the AT Area to Include All Photos

PhotoMesh can automatically calculate the AT area based on the polygon that includes all the project's photos plus a buffer. The automatic AT area's buffer can be based on the collections' largest effective range, i.e., the maximum distance from the camera for which photo information is considered valid for aerotriangulation, or on a buffer you set.

To set the AT area to include all the project's photos:

1. On the **Home** tab, in the **Area** group, click the arrow under **AT Area** and select **Change Automatic Setting.** The Automatic Area Setting dialog box is displayed.
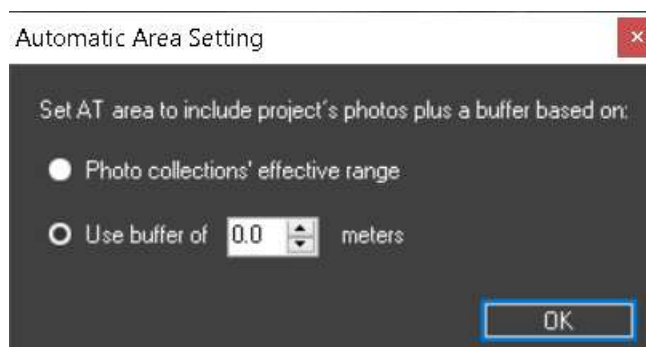
Automatic Area Setting Dialog Box

2. Select one of the following options:
   - **Photo collections' effective range** - Calculate the AT area based on the positions of all the project's photos plus a buffer that is based on the collections' largest effective range.
   - **Use buffer of** - Calculate the AT area based on the positions of all the project's photos plus the buffer that you set.

3. On the **Home** tab, in the **Area** group, click the arrow under **AT Area** and select **Automatic Using Effective Range/Automatic Using Buffer.** The option that is displayed corresponds to the option selected in step 2.

Drawing a New AT Area

To set an AT area within a specific area of interest, do the following:

Note: An AT Area polygon can only be redrawn when Review mode is off. **See** "About Reviewing Your Build" in the "Reviewing Your Build" chapter for more information about Review mode.

1. On the **Home** tab, in the **Area** group, click the arrow under **AT Area** and select **Redraw AT Area.**

2. In the **3D Window**, draw an AT Area polygon around the required area by clicking in the required locations. You must place at least three points. Right-click to finish the polygon creation.

Importing an AT Area Polygon

To import an AT area polygon;

1. On the **Home** tab, in the **Area** group, click the arrow under **AT Area**, and select **Import**. The Import Aerotriangulation Area Polygon Layer dialog box is displayed.

2. Browse to the shapefile with the required polygon, and click **Open**.

   Note: The first polygon found in the file is used as the aerotriangulation area polygon.

Editing an AT Area Polygon

To edit the current AT area:

Note: An AT Area polygon can only be edited when Review mode is off. **See** "About Reviewing Your Build" in the "Reviewing Your Build" chapter for more information about Review mode.

1. On the **Home** tab, in the **Area** group, click the arrow under **AT Area** and select **Edit AT Area**.

2. In the **3D Window**, place the cursor over one of the nodes of the AT Area polygon. The node turns blue, and the cursor changes to **Move** mode ⚡⊹.

3. Click the node and drag it to the required location.

4. Release the mouse button to set the node at the current location.

5. Red dots on the polygon's line segments indicate where a node can be added. Click to add a node, and then drag it to the required location.

6. Right-click to finish editing the polygon.

Navigating to the AT Area

To fly or jump to the AT area:

▪ On the **Home** tab, in the **Area** group, click the arrow under **AT Area** and select **Fly to Area**/**Jump to Area**.

Setting the Reconstruction Area

Setting the reconstruction area defines the area of the project on which PhotoMesh should perform the reconstruction steps of the build process, namely: point cloud creation, model creation, model texturing, and project output. The reconstruction area can be defined before building the project (after identifying which areas of the project have sufficient coverage using the [coverage map](#)) and also after the build is complete if modifications to the build area are necessary. The same area set for the aerotriangulation can be automatically set for the reconstruction area, or a different area can be defined by either drawing or importing a reconstruction polygon.

Setting a Reconstruction Area Equal to the AT Area

To set a reconstruction area that is the same as the AT area:

▪ On the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area**, and select **Same as AT Area**.

Drawing a New Reconstruction Area

To draw a reconstruction area polygon:

1. On the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area**, and select **Redraw Reconstruction Area**.

2. In the **3D Window**, draw a reconstruction area polygon around the required area by clicking in the desired locations. You must place at least three points. Right-click to finish the polygon creation.

Importing a Reconstruction Area Polygon

To import a reconstruction area polygon;

1. On the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area**, and select **Import**. The Import Reconstruction Area Polygon Layer dialog box is displayed.

2. Browse to the shapefile or KML with the required polygon, and click **Open**.

   Note: The first polygon found in the file is used as the reconstruction area polygon.

Editing a Reconstruction Area Polygon

To edit a current reconstruction area polygon:

1. On the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area**, and select **Edit Reconstruction Area.**

2. In the **3D Window**, place the cursor over one of the nodes of the reconstruction area polygon. The node turns blue, and the cursor changes to **Move** mode .

3. Click the node and drag it to the required location.

4. Release the mouse button to set the node at the current location.

5. Red dots on the polygon's line segments indicate where a node can be added. Click to add a node, and then drag it to the required location.

6. Right-click to finish the polygon editing.

Exporting a Reconstruction Area Polygon

To export a reconstruction area polygon:

1. On the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area**, and select **Export**. The Export Reconstruction Area Polygon dialog box is displayed.

2. Browse to the location you want to save to and click **Save**.

Navigating to the Reconstruction Area

To fly or jump to the reconstruction area:

▪ On the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area** and select **Fly to Area**/**Jump to Area**.

CHAPTER 13 CONTROL POINTS

About Control Points

PhotoMesh provides three types of control points:

- **Ground control points (GCP)** – Improve the accuracy of the model's position by associating real-world X, Y, Z coordinates (generally obtained through surveying or sampling methods) with corresponding locations (in pixels) in photos. In order to enable your control points to be fully utilized in the AT process, it is recommended to add at least 3-5 control points per AT tile. When only two control points are provided, simple vertical adjustment, scaling and rotation are performed. When only one control point is provided, only vertical adjustment is performed.

- **Check points** – Ground control points that do not affect the AT calculation and are only used to sample the AT accuracy in the result. The point is included in the build report and shows calculated errors for accuracy validation purposes.

- **Tie points** – Establish a correspondence between project photos and improve aerotriangulation results through identification of the same physical point in three or more photos. The XYZ position of the physical point is not necessarily known. Generally, it is recommended to run an initial aerotriangulation to identify photos that either weren't calculated in AT or that have high median errors, as well as areas of insufficient matches between groups of photos.

When a control point is added, a control point symbol is displayed in the 3D Window in the place where it was added. When selected, the control point turns to purple.
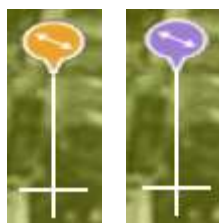
Control Point Symbol

Do any of the following:

- Create control points:
    - [Ground control points](#)
    - [Tie points](#)

- [Navigate to a control point](#)

- [Edit a control point](#)

- [Mark a control point in photos](#)

- [Hide a control point symbol](#)

- [Exclude a control point](#)

- [Delete a control point](#)

- [Export control point data to a tab delimited .gcp (text format) file](#)

- [Import control point data in a tab delimited .gcp (text format) file](#)

- [Use control point as a check point to sample the AT accuracy in the result](#)

The capabilities listed above (except for the export of control point data) are available from the Control Point contextual tab and also from the control point shortcut menu. **See** "[Control Point Menu](#)" in the "Using the Project Tree and Item List" chapter for more information. Control point data is exported from the Control Points command on the Home tab.

Recommendations for Ground Control Point Placement

The following guidelines are recommended for placement of control points in your AT tiles:

- Distribute control points uniformly across the AT tile.

- Make sure to add control points in the tile areas with the highest and lowest altitudes.

- Begin control point creation by adding control points around the perimeter of the tile.

- Control points should be created on a flat terrain or mesh that has no obstructions close by.

Creating Ground Control Points

Ground control points can be created/added in any of the following ways:

- [Create a control point at a known precise position, generally a surveyed point](#)

- [Create a control point by clicking on an identifiable location on the terrain or Lidar (e.g. road intersection or street corner)](#)

- [Import a tab delimited .gcp (text format) file with control point information](#)

After creating a ground control point, you mark it in the relevant photos using the Control Point Editor. **See "**[Marking Control Points in Photos](#)**"** in this chapter for information.

Creating Ground Control Points at Known Coordinates

To create a ground control point at known coordinates:

1. On the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Create Ground Control Point**. The Control Point Editor is displayed with the newly-created control point selected in the control point table at the bottom of the Editor.

   Note: You can also create a control point at known coordinates from the Control Point Editor: On the **Home** tab of the Control Point Editor, in the **Control Point** group, click **Add New**.

2. Set the control point coordinates and properties. **See** "[Editing Control Points](#)" in this chapter for information.

3. Mark the control point in the relevant photos, and then save the project. **See** "[Marking Control Points in Photos](#)" in this chapter for information.

Creating Ground Control Points from Identifiable Locations on the Terrain

To create a ground control point from a specific location on the terrain (including Lidar):

1. On the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Create Ground Control Point from Terrain**.

2. Click in the **3D Window** to define the control point's location. The Control Point Editor is displayed showing a photo viewing window, a photos panel, a control point table, and a sampled photos panel - that gradually becomes populated with photo thumbnails as you mark the control point in photos.

3. Mark the control point in the relevant photos. **See** "Marking Control Points in Photos" in this chapter for information.

4. Modify the control point properties as required, and then save the project. **See** "Editing Control Points" in this chapter for information.

Creating Tie Points

Tie points can be created in either of the following ways:

▪ By selecting in the Photos list the photos between which you want to establish a correspondence. The Control Point Editor opens with the Photos panel populated by the selected photos. You can also select photos after opening the Editor. **See** "Creating Tie Points in Selected Photos" in this chapter for information.

▪ By creating a tie point from an easily identifiable location on the terrain. The Control Point Editor opens with the Photos panel populated by the photos whose frustum intersects with the created point. **See** "Creating Tie Points from Identifiable Locations on the Terrain" in this chapter for information.

After creating a tie point, you mark it in the relevant photos using the Control Point Editor. **See "**Marking Control Points in Photos**"** in this chapter for information.

Creating Tie Points in Selected Photos

To create a tie point in selected photos:

1. In the **Project Tree**, select the required photo collection, and then in the **Photos** list**,** select the required photos.

2. On the **Photo** tab, in the **Tie Point** group, click **Create Tie Point**. The Control Point Editor is displayed with the newly-created tie point selected in the control point table at the bottom of the Editor, and with a Custom selection in the Photos panel that includes all the photos selected in the Project Tree.

   Note: You can also create tie points and filter the photos for sampling from the Editor itself: **On** the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Open Control Point Editor**. Then on the **Home** tab of the Control Point Editor, in the **Control Point** group, click the arrow under **Add New** and select **Add New Tie Point**. **See** "Marking Control Points in Photos" in this chapter for information on filtering the photos for sampling.

3. Set the tie point properties. **See** "Editing Control Points" in this chapter for information.

4. Mark the tie point in the relevant photos, and then save the project. **See** "Marking Control Points in Photos" in this chapter for information. The Sampled Photos panel gradually becomes populated with photo thumbnails as you mark the tie point in photos.

Creating Tie Points from Identifiable Locations on the Terrain

To create a tie point from a specific location on the terrain:

1. On the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Create Tie Point from Terrain**.
2. Click in the **3D Window** to define the tie point's position. The Control Point Editor is displayed with the Photos panel populated by photos whose frustum intersects with the tie point created.
3. Mark the tie point in the relevant photos. **See** "Marking Control Points in Photos" in this chapter for information. The Sampled Photos panel gradually becomes populated with photo thumbnails as you mark the tie point in photos.

Navigating to a Control Point

To fly or jump to a control point:

1. In the **Project Tree**, select the required control point.
2. On the **Control Point** tab, in the **Navigation** group, click **Fly To/ Jump To**.

Editing Control Points

To edit a control point:

1. Do either of the following:
   - In the **Project Tree**, select the required control point. The control point's properties are displayed in the Property Sheet panel.

     Note: To set the same properties for multiple control points, use CTRL-click or SHIFT-click to multi-select the points in the Project Tree. Then set the properties as required.

   - In the **Project Tree**, right-click the required control point, and select **Edit** from the shortcut menu. The Control Point Editor opens with this control point selected in the control point table at the bottom of the Editor.

   Note: When a control point is edited in either the Control Point Editor or property sheet, its properties are dynamically updated also in the other. Many of the control point properties below are also included in the Add New Control Point dialog box (that is displayed when adding a new control point from the Control Point Editor). **See** "Marking Control Points in Photos" in this chapter for information.

2. Set the control point properties, and then save the project.

Parameter	Definition
General	
Excluded	Select the check box to exclude the control point from the project. **See** "Excluding Control Points" in this chapter for more information.
Name	Name of the control point as it appears in the Project Tree.
Type	Select the point type: control point, tie point, or check point.
Horizontal Accuracy (meters)	Accuracy of the control point's X and Y coordinates. The AT is adjusted to the control point within the specified accuracy tolerance. This field is not displayed in the Add New Control Point dialog box.

Vertical Accuracy (meters)	Accuracy of the control point's altitude. The AT is adjusted to the control point within the specified accuracy tolerance. This field is not displayed in the Add New Control Point dialog box
Check Point	Select the check box to use this control point as a check point. Check points do not affect the AT calculation and are only used to sample the AT accuracy in the result. The point is included in the build report and shows calculated errors for accuracy validation purposes.
Horizontal Error	This field is only displayed in the Control Point Editor. If the horizontal error is greater than the horizontal accuracy, this field is marked in red.
Vertical Error	This field is only displayed in the Control Point Editor. If the vertical error is greater than the vertical accuracy, this field is marked in red.
Sampling Error	This field is only displayed in the Control Point Editor. If the sampling error is greater than 1 pixel, this field is marked in red.
Position	
Coordinate System	Coordinate system of the control point. Click Select ▣ to set a different coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information.
X	East-west coordinate of the control point.
Y	North-south coordinate of the control point.
Altitude	Altitude of the control point in meters.

Marking Control Points in Photos

Control points are marked in photos using the Control Point Editor. In addition to the ribbon, the Control Point Editor includes a photo viewing window, a photos panel, a control point table, and a sampled photos panel - that gradually becomes populated with photo thumbnails as you mark the control point in photos. The photos panel displays the photos from which to select the photos for marking control points, either as thumbnails or in table format with relevant photo information. You can also manually add photos to this list as described in the steps below. Mark the control point in a few photos from each collection or photo direction. It is recommended to have at least 10 sampled photos for each control point.

Note: In order to enable your ground control points to be fully utilized in the AT process, it is recommended to add at least 3-5 ground control points per AT tile. When only two ground control points are provided, simple vertical adjustment, scaling and rotation are performed. When only one ground control point is provided, only vertical adjustment is performed.

Note: Any control point marked in fewer than three photos will be marked with an exclamation mark in the Project Tree, and in red in the Control Point Editor.

To mark a control point in photos:

1. Create the control point. **See "**Creating Ground Control Points**"** and "Creating Tie Points" in this chapter for information. In the **Project Tree**, select the required control point.
2. On the **Control Point** tab, in the **Manage** group, click **Edit**. The Control Point Editor is displayed.
3. From the **Photos** filter, select which photos should be listed or displayed in the Photos panel:
   - **All in frustum (Calculated)** – All the photos whose calculated frustum intersects the control point.
   - **All in range (Calculated)** – All photos with the selected control point in their calculated range (according to their Effective Range).

- **All in project (Original)** – All photos in project (up to 2000 photos).

- **All in range (Original)** – All photos with the selected control point in their range (according to their <u>Effective Range</u>).

  Note: This option is only available if photos have positioning information.

- **All in frustum (Original)** – All the photos whose frustum intersects the control point.

  Note: For photos with positioning and orientation information, this is determined based on frustum, and for photos with only positioning, according to the effective range set (See "<u>Collection Property Sheet</u>" in the "Photo Management" chapter for information).

- **Custom selection** – All photos selected in the Photos list before creating the tie point from the Photo tab. See "<u>Creating Tie Points in Selected Photos</u>" in this chapter for more information.

4. Select a viewing format for the **Photos** panel:

   - ![thumbnail icon] – View as thumbnails

   - ![table icon] – View in table format with information about distance, sampling error, etc. for each photo

5. If viewing the photos in table format, select the column by which you want to sort the table (in ascending order). Select the column a second time to switch to descending order. For filtering and advanced sorting options, see "<u>Filtering Data in the Control Point Editor</u>" in this chapter for information.

6. In the **Photos** panel, select the photo on which you want to mark the position of the selected control point. The photo is displayed in the photo viewing window with a white cross hairs indicating the projected location of the control point that PhotoMesh automatically calculated. You can type 'n' to select the next photo, or 'p' to select the previous one.

   Note: Any photo thumbnails that are outside of the current build area are displayed in strikethrough font.

7. Find the correct position in the photo in which to mark the control point. Use the mouse to zoom in and out and pan in the photo or use the ribbon commands:
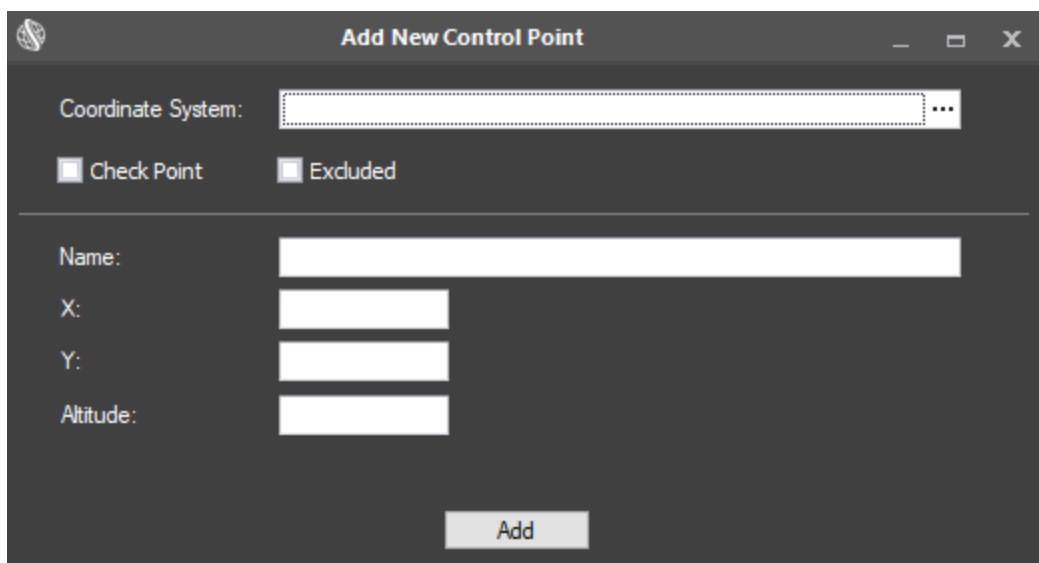
Zoom In	Zooms in.
Zoom Out	Zooms out
Zoom to Mark	Zoom in to the marked control point. This command is only available after a control point was marked in the photo.
Fit to Window	Resizes the photo to fit in the window.
Actual Size	Displays the photo at actual size.

8. On the **Home** tab of the Control Point Editor, in the **Control Point** group, click **Sample in Photo**, and then click on the photo where you want to set the position of the control point. You can also type 's' on the keyboard to toggle between navigation and sampling modes. A red crosshairs is displayed where you clicked in the 3D Window and in the corresponding position on the photo thumbnail which is added to the "Sampled Photos" panel. The photo's XY pixel coordinates are displayed in the Mark X and Mark Y fields on the Control Point Editor ribbon in the Control Point group. If the Photos panel is displayed in table format, the check box in the Marked column is selected (If displayed as thumbnails, a check is displayed on top of the reviewed photo). The following information is displayed in the 3D Window:

	User sample for position of control point.
	Projected location of control point.
	Projected triangulation – Projection of the triangulation center (adaptive weighted median of all ray intersections) back to the photos.
	Sampling error - RMS
Sampling Error - Photo	Distance, in pixels, between the projected triangulation and the user sample in this photo.
Sampling Errors - RMS	Root mean square (RMS) of all distances, in pixels, between the projected triangulation and the user sample in all photos.
Horizontal Error	Horizontal distance, in meters, between the projected triangulation and the control point position.
Vertical Error	Vertical distance, in meters, between the projected triangulation and the control point position.

9. If you want to modify the photo's XY pixel coordinates, on the **Control Point Editor** ribbon's **Control Point** group, in the **Mark X** and **Mark Y** fields, type the required values.

10. If you want to clear the previously marked position of a control point, in the Control Point Editor, in the **Control Point** group, click **Clear from Photo**. The photo is removed from the "Sampled Photos" panel and the check is cleared from that photo thumbnail in the Photos panel.

   Note: Only one position can be marked in each photo. If an additional position is marked, the previously marked position is automatically deleted and replaced by the new position.

11. Repeat steps 5-9 for each of the photos in which you want to mark the selected control point.

12. If you want to mark the position of the control point in a photo that does not appear in the Photos panel, you can either change the Photos filter to include that photo or do the following:

   a. On the **Home** tab of the Control Point Editor, in the **Photos** group, click **Add Photo by Name**. The Enter search term dialog box is displayed.

   b. Type a search term by which to identify the required photo, and click **Ok**. The Photos in Area section is populated with all photos that match the search term.

   c. Perform steps 5-9 above.

13. If you want to edit another control point in the Control Point Editor, in the **Control Points** table on the bottom of the Control Point Editor, select the required control point, and then perform steps 3-9 above. Type 'N' to select the next control point in the Control Points table, or 'P' to select the previous control point.

14. If you want to add a new ground control point, do the following:

   a. On the **Home** tab of the Control Point Editor, in the **Control Point** group, click **Add New**. The Add New Control Point dialog box is displayed.

Add New Control Point Dialog Box

b. Enter the control point information, and click **Add**. **See "**Editing Control Points" for a description of the control point fields.

c. Perform steps 3-9 above.

15. If you want to add a new tie point, on the **Home** tab of the Control Point Editor, in the **Control Point** group, click the arrow under **Add New**, and select **Add New Tie Point**. The tie point is added to the control point table.

Filtering Data in the Control Point Editor

To filter control point lists in the Control Point Editor:

1. Point to the column by which you want to filter, and click the **Filter** icon ▼. **See** "Filtering Data" and "Filter Editor" in the "Using the Project Tree and Item List" chapter for information.

2. Click **Close**. The applied filter(s) are listed at the bottom of the Photos panel, e.g.,

    ☒ ✓ [Sampling Error (px)] >= '10.864' And [Sampling Error (px)] <= '12.493' And [Marked] = 'Checked'   Edit Filter .

    Click X to remove a filter.

3. To access additional sorting commands, right-click any column header. Select any of the following commands:

Command	Description
Sort Ascending	Sort all data in the table by this column in ascending order.
Sort Descending	Sort all data in the table by this column in ascending order.
Clear All Sorting	If multiple columns were sorted, this option is displayed to clear all.
Group by This Column	Group the data by this column. **Note**: If you want to clear the grouping, right click above the column headers, and select **Clear Grouping**.
Show Group by Box	Display boxes, above the photo table, indicating how the data is grouped in the table.

	Note: If you want to clear the Group by boxes, right click above the column headers, and select **Hide Group by Box**.
Hide This Column	Hide the selected column
Column Chooser	Customize which columns show in the table. Drag and drop columns to and from the **Customization** dialog box.
Best Fit	Adjust the selected column's width for the best fit for its contents.
Best Fit (all columns)	Adjust the all the table's column widths for the overall best fit.
Filter Editor	Open the Filter Editor to edit the table filter. You can use the Filter Editor to add and remove conditions. A search expression can consist of more the one condition. In this case, you need to insert a connector between the conditions to specify their relations. In addition to AND, you can use the OR, NOT and [] connectors to define the relations between conditions.
Show Find Panel	Show a Find panel to search for a specific text. **Note**: To hide the Find panel, click the X next to it. it.
Show Auto Filter Row	Display a row with a filter option for each of the columns.

Control Point Editor Keyboard Shortcuts

The following keyboard shortcuts are available:

Shortcut	Description
s	Toggle between navigation and sampling modes.
n	Selects the next photo in the Photos panel and displays it in the photo viewing window.
p	Selects the previous photo in the Photos panel and displays it in the photo viewing window.
N	Selects the next control point in the Control Points table.
P	Selects the previous control point in the Control Points table.

Hiding Control Point Symbols

To hide a control point symbol:

▪ Clear its check box in the **Project Tree**.

Excluding Control Points

To exclude a control point from the project:

1. In the **Project Tree**, select the required control point.
2. On the **Control Point** tab, in the **Selection** group, click **Exclude**. When a control point is excluded, it is displayed in strikethrough font in the Project Tree and the Excluded property check box is selected in its property sheet.

Deleting Control Points

To delete a control point:

1. In the **Project Tree**, select the required control point.
2. On the **Control Point** tab, in the **Selection** group, click **Delete**.

Exporting Control Points

Control point data can be exported to a .gcp (text format) file.

Note: All control points are exported in EGM96 geoid vertical coordinate system even if they were defined in "WGS84ELLIPSOID".

To export a project's control point data:

1. On the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Export**. The Export Control Points dialog box is displayed.
2. Type a name for the .gcp (text format) file and browse to the required location in which to save the file.
3. Click **OK**.

Importing Control Points

Control point data can be imported in a tab delimited .gcp (text format) file:

- First line (optional): Define the control points' coordinate system in WKT format. (Default: geographic Lat-Long WGS84.)

```
wkt=……
```

  Note: The WKT information must be in a single line without any line breaks. You can use the Coordinate System Generator to generate a WKT string defining the required coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information.

- Second line (optional): Indicate the number of control points. If this information is not provided, each subsequent line in the file is interpreted as providing positioning information for a distinct control point. If the file includes this information, all lines after the number of lines in cp=[number] are interpreted as mapping between control points and photos.

```
cp=[number]
```

- For each control point, set the control point's position on the terrain (X, Y, Altitude (Optional), Name (Optional), Type (Optional; Possible values: 1-control; 2-check; 3-tie) in tab delimited format.

```
[x coordinate] <TAB> [y coordinate] <TAB> [altitude] <TAB> [name] <TAB> [type]
```

  Note: The Type parameter can only be passed together with the Name parameter.

  Note: The altitude base is assumed to be "GEOID". If you want to change the altitude base, you can select all the control points after import and change the altitude base value in their property sheet.

- Specify the position (in pixel coordinates) in a specified photo to associate with a particular control point (referenced by index).

```
[control point index] <TAB> [photo file path] <TAB> [x pixel coordinate] <TAB> [y pixel coordinate]
```

Example:

```
wkt=GEOGCS["WGS 84",DATUM["WGS_1984",SPHEROID["WGS
84",6378137,298.257223563,AUTHORITY["EPSG","7030"]],AUTHORITY["EPSG","6326
"]],PRIMEM["Greenwich",0,AUTHORITY["EPSG","8901"]],UNIT["degree",0.0174532
925199433,AUTHORITY["EPSG","9122"]],AUTHORITY["EPSG","4326"]]

cp=2

9.63390819201651 <TAB> 47.4233597051724 <TAB> 401.463806152344 <TAB>
Control point1

9.63390818337074 <TAB> 47.4233597019402 <TAB> 401.463793239556 <TAB>
Control point2

0 <TAB> C:\Photos\1.tif <TAB> 384 <TAB> 5456

0 <TAB> C:\Photos\2.tif <TAB> 7689 <TAB> 4761

0 <TAB> C:\Photos\3.tif <TAB> 8738 <TAB> 4340

1 <TAB> C:\Photos\4.tif <TAB> 388 <TAB> 5450

1 <TAB> C:\Photos\5.tif <TAB> 7690 <TAB> 4761

1 <TAB> C:\Photos\6.tif <TAB> 8741<TAB> 4326
```

To import control points:

1. On the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Import**. The Import Control Point File dialog box is displayed.
2. Browse to the required .gcp file and click **Open**.
3. Modify the control point properties as required, and then save the project.

Adding a Reference Project to Use in Aerotriangulation

Georeferencing a project is the process of adjusting photos in your project to the geographic location of photos of the same geographic area in a different project. When building a project that overlaps partially or entirely with a different project that will be overlaid on the same map in a client viewer, it is important to ensure that matching areas align properly. This can be achieved by adding the older "base" project as a reference project for the new "target" project. After adding the base project, it is listed in the Project Tree under Reference Projects, but its photos are not displayed in the 3D Window. When building the new project, PhotoMesh establishes a correspondence between points in the reference project and target project, so that the positions of identified features in the target project match the geographic positions of the same features in the reference project.

To add a reference project:

1. On the **Home** tab, in the **Add** group, click the arrow under **Control Points** and select **Add Reference Project**. The Add Reference Project dialog box is displayed.
2. Browse to the required reference project, i.e., the calculated AT of that build (atResult_v12.sqlite), and click **Open**. When the aerotriangulation step of the build process is performed, the project will be automatically aligned to the reference project.

CHAPTER 14 FUSERS

About Fusers

PhotoMesh can take advantage of computer clusters to accelerate database creation. Fusers allow you to share the demanding processing requirements of the different build steps (data preparation, AT, point cloud, mesh model, texture, 3DML/other outputs) between several computers on the same network. The master computer utilizes the computing power of the network's client computers by connecting to PhotoMesh fusers on the client machines.

The fuser remains idle until used by the master computer on the network. Once started, the fuser runs on the client machines, with only an icon in the system tray that opens a status log and a dialog to stop or suspend fusers. You can install the PhotoMesh Fuser on the same computer with the PhotoMesh Build Manager and/or on other computers.

In addition to standard fuser use, PhotoMesh also offers two options for auto-scaling of fusers to enable dynamic allocation of fusers based on project requirements:

- **Amazon Web Services (AWS) Fusers** – AWS fuser instances can be dynamically launched whenever there are unassigned pending fuser tasks, to further accelerate model creation.

- **Fusers Pool** – Fusers can be shared between several PhotoMesh Build Managers in your organization, enabling each machine running PhotoMesh to use the combined resources of all the machines in the pool. A distributed management approach is used to manage the pool fusers, whereby each PhotoMesh Build Manager manages its own fuser needs and resources by requesting fusers from the pool when generating a build and returning the fusers when they are no longer needed.

When beginning the build process the PhotoMesh Build Manager application determines what build steps are required and assigns them to the different fusers. PhotoMesh Build Manager continuously monitors the fusers' progress and assigns new processing tasks based on fuser availability.

Both PhotoMesh Build Manager and the PhotoMesh fusers require read-write access to the working and project folders.

Fuser Management (on the Master computer)

All available fusers are listed in the Fusers section of PhotoMesh Build Manager. As the build process is progressing, PhotoMesh Build Manager displays fuser information (which fusers are in use, which aren't, etc.). **See** "Monitoring a Build" in the "Building" chapter for more information.

Setting the Working Folder on the Master Computer

The working folder is the folder that PhotoMesh Build Manager (on the Master computer) and PhotoMesh fusers use to communicate: to deposit and collect pending tasks, and share information about fuser availability, status and progress.

The working folder should be set to a network folder with read-write access permissions, for both the master and client computers. It is recommended to use UNC format (*[UNC Master]* *folder\WorkingFolder, e.g.*: \\Machine1\c\PhotoMesh7.0\WorkingFolder), though a mapped network drive can also be used as long as the same letter is mapped to the same share on all computers.

Note: When using fusers, the project must also be created in a network folder with read-write access permissions, for both the master and client computers. It is recommended to use UNC format (*[UNC Master]* *folder\ProjectFolder, e.g.*: \\Machine1\c\PhotoMesh\ProjectFolder), though a mapped network drive can also be used as long as the same letter is mapped to the same share on all computers. **See** "Creating and Opening a PhotoMesh Project" in the "Project Management" chapter for more information.

The working folder must be on a device that supports the number of concurrent network connections you require, e.g., Windows 10 limits the number of supported concurrent connections to 20.

It is recommended to create the working folder on a on a fast drive that is different from the one used for the project data, most preferably on a local solid-state drive (SSD) on the master machine that is shared with the other fuser machines.

To set the working folder on the master computer, do either of the following:

▪ In the **Options** dialog box, set the working folder. **See** "Setting PhotoMesh Options" in the "Basic Concepts" chapter for more information.

OR

▪ In **PhotoMesh Build Manager**, next to the **Working Folder** field, click **Browse**. Then browse to the required location.

Using Fusers

To use fusers on the master computer:

1. In **PhotoMesh Build Manager**, select the required fusers.
2. If you want PhotoMesh Build Manager to automatically use any new fusers that connect to the project's working folder, select **Automatically use new available fusers** from the drop-down list.
3. If there are no available fusers, a dialog box is displayed asking if you want to start a local fuser. Click **Yes** to start a local fuser. Click **No** to start the build without adding a fuser. Processing will begin automatically when fusers are added.
4. If you want to change the working folder that PhotoMesh Build Manager and PhotoMesh fusers use to communicate (to deposit and collect pending tasks, and share information about fuser availability, status and progress), click **Browse**. Then browse to the required location.

   Note: The working folder in the PhotoMesh Fuser dialog box on the client computer should be set to the same location as the location set here. **See** "Fuser Management (on the client Computer)" in this chapter for information.

Fuser Management (on the client computer)

The owner of the fuser computer has full control over fuser usage. For information on installing and uninstalling fusers, **see** "Installing PhotoMesh Fusers" and "Uninstalling PhotoMesh Fusers" in the "Getting Started" chapter. Fusers can be installed locally, or run from a shared network path.

Setting the Working Folder on a Client Computer

The working folder is the folder that PhotoMesh Build Manager (on the master computer) and PhotoMesh fusers use to communicate: to deposit and collect pending tasks, and share information about fuser availability, status and progress.

The working folder on each of the client computers must be set to the same network folder as was set for the master computer. **See** "Setting the Working Folder (on the Master Computer)" in this chapter for more information.

To set the working folder on the client computer:

1. In the **PhotoMesh Fuser** dialog box, click **Change**. The Select Folder dialog box is displayed.
2. Browse to the required folder and click **Select Folder**.

    Note: If the working folder is changed during processing, processing is stopped.

Starting Fusers

Once a fuser is started on a client computer, it is added to the list of available fusers on the master computer. Fusers can be installed locally, or run from a shared network path.

To start a fuser:

▪   Click **Start** > **Skyline PhotoMesh** > **PhotoMesh Fuser**. A PhotoMesh fuser icon is placed in the Windows system tray.

    Note: In the full PhotoMesh installation, the name of the folder containing the fuser application is **Skyline PhotoMesh**, while in the standalone fuser installation, the folder name is **Skyline PhotoMesh Fuser**.

Fuser Log

To open the fuser log:

1. In the system tray, click the **PhotoMesh Fuser icon**, and then click **Open PhotoMesh Fuser**. The PhotoMesh Fuser dialog box is displayed, showing the fuser log in the lower half of the dialog box.
2. If you want to clear the log, click **Clear Log**.
3. If you want to stop the log from automatically scrolling to the last line of the log, click **Scroll Lock**.

Disabling a Fuser

Fusers can be temporarily disabled.

To disable a fuser:

▪   Click the **PhotoMesh Fuser icon**, and then click **Disable**. When you want to restart the fuser, click the **PhotoMesh Fuser icon** and then click **Enable**.

    Note: The Master computer reattempts to use the fuser every five minutes. The fuser is used only if its owner has enabled it.

Exiting a Fuser

You can permanently exit a fuser.

To exit a fuser:

▪   Click the **PhotoMesh Fuser icon**, and then click **Exit**.
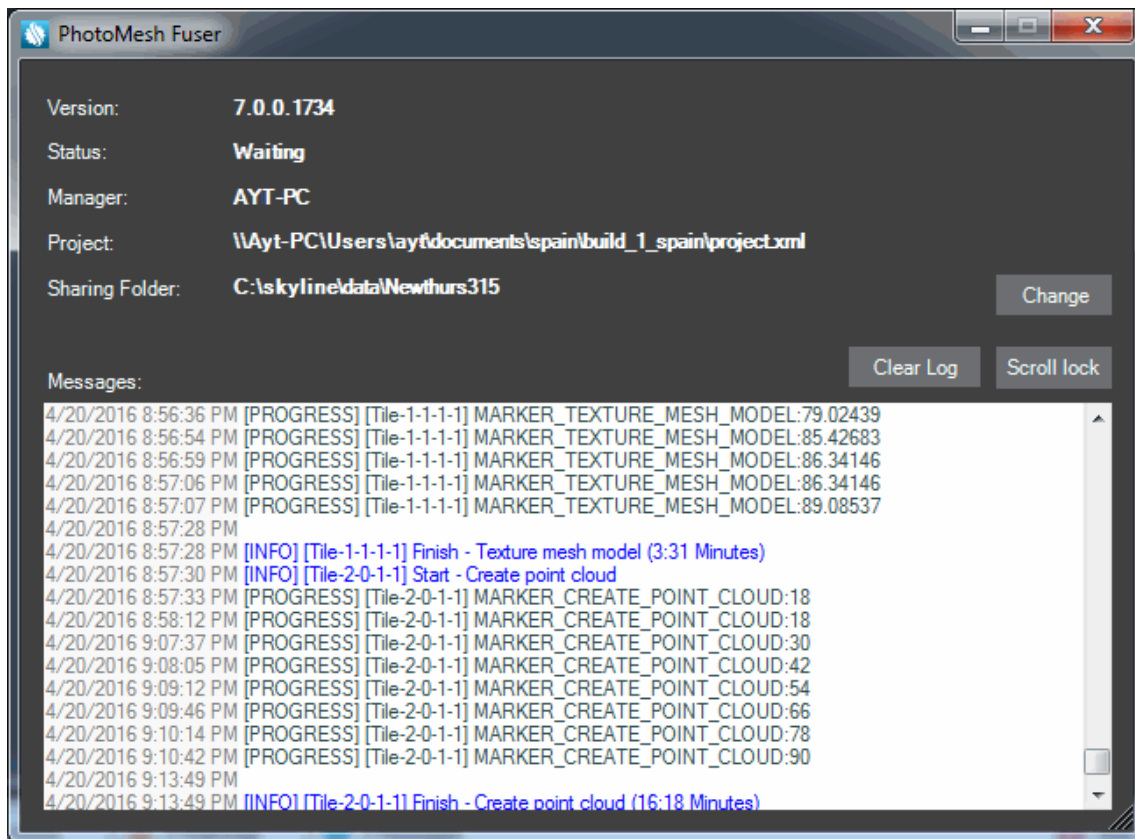
PhotoMesh Fuser Log

Running Fusers Upon Login

You can schedule fusers to be run automatically upon login, either individual fusers or the fuser service (PMFuserService.exe), if using a fusers pool. **See** "Fusers Pools" in this chapter for information.

To run fusers upon logon using the Task Scheduler

1. Right-click **Task Scheduler Library** and select **Create Task**.

2. **General** > **Name**: *Start PM Fuser* OR *Start PM Fuser Service*.

    Note: If using a fusers pool, then start the fuser service (PMFuserService.exe) rather than a fuser. See "Fusers Pools" in this chapter for information.

3. **Triggers**:

    a. Click **New**.

    b. **Begin the task**: Select **At log on**.

    c. **Settings**: Specific user.

    d. Click **OK**.

4. **Actions**:

    a. Click **New**.

    b. **Program/script**: *\\10.0.0.10\d\PhotoMesh\fuser\PhotoMeshFuser.exe* OR *\\10.0.0.10\d\PhotoMesh\fuser\PMFuserService.exe*.

> **Note:** If using a fusers pool, then run the fuser service (PMFuserService.exe) rather than a fuser. **See** "Fusers Pools" in this chapter for information.

    c.   Click **OK**.

5. **Settings**:

    a.   Select **if the task fails, restart every**, and keep the default 1 minute and 3 times.

6. Click **OK**.

If you want to log in automatically upon start up, do the following:

1. Click the **Start** ⊞ button, type "cmd", and then select **Command Prompt** from the list. In the command prompt window type the following:
   control userpasswords2

2. Select the user that you want automatically logged in, e.g., Administrator, and clear the **Users must enter a user name and password to use this computer checkbox**, and click OK**.

3. Enter and confirm the password for the fuser computer.

Fuser Auto-Scaling

PhotoMesh offers two options to scale project production up and down based on resource needs:

- Amazon Web Services (AWS) Fusers – AWS fuser instances can be dynamically launched whenever there are unassigned pending fuser tasks, to further accelerate model creation.

- Fusers Pool – Fusers can be shared between several PhotoMesh Build Managers in your organization, to dynamically allocate and free fusers based on each project's resource needs.

These options can be used together or separately. When either option is used, PhotoMesh Build Manager continuously monitors the fusers' progress and status and the processing requirements, and allocates or frees fusers as required.

Using Amazon Web Services (AWS) Fusers

PhotoMesh can work with Amazon Web Services to scale project production up and down based on resource needs. AWS fuser instances can be dynamically launched whenever there are unassigned pending fuser tasks, to further accelerate model creation. PhotoMesh Build Manager continuously monitors the fusers' progress and status and the processing requirements of the current step, adding and terminating AWS fusers as required. **See** "Getting Started with AWS" for more information.

To use AWS fusers:

1. In the **Build Manager**, toggle on **Use AWS Instances**. You can also access these settings from the Tools tab (AWS Instance Settings command) and from the Start page's Automation Dashboard.

   Note: The **Automatically use new available fusers** check box must also be selected, so that PhotoMesh will use the AWS fuser instances after they are launched.

2. If PhotoMesh Build Manager is running on a local computer, and this is the first time connecting to the AWS fusers, the AWS Security dialog box is displayed. Enter your AWS security information, and click **OK**.

   Note: If PhotoMesh Build Manager is running on an AWS instance with an IAM (Identity and Access Management role), security credentials are not required. **See** "Creating an IAM User for Running PhotoMesh Project Using an On-Premises Master" in the "Getting started with AWS" chapter for information.

AWS Security Dialog Box

Access Key ID	Token that allows PhotoMesh to communicate with your AWS account.
	Note: You can find your Secret Access Key on the My Security Credentials link in the AWS Management Console.
Secret Access Key	Token that allows PhotoMesh to communicate with your AWS account.
	Note: You can find your Secret Access Key on the My Security Credentials link in the AWS Management Console.
Region	Select an AWS region.
Remember my credentials	Select this option to save the password. The password will be saved and will be displayed using asterisks the next time the dialog box is opened.

3. In the **Auto-Scaling Fusers Settings** dialog box, select the **Use AWS Instances** check box, and then enter the following information:

Maximum Price per Spot Instance	Set the maximum price you want to pay for an EC2 fuser spot instance.
Maximum Instances	Set the maximum number of fuser instances to launch.
Launch Rate	Set the maximum number of fuser instances to launch per minute, e.g. if Maximum Instances was set to 200, and the Launch Rate to 50, the Build Manager will spread the launch of the fusers over 4 minutes. This prevents a potential bottleneck that could be caused by a large number of fusers attempting to read the build data at the same time.
Terminate AWS instances when build finished	Select the check box if you want to terminate AWS fuser instances when the build is complete.
Select fuser instance launch parameters	

Region	The region the master is running in, or the region set in the AWS Security dialog box (Read-only).
Configuration Name	Name of the JSON configuration file to use. The JSON file contains an array of fuser instance types that PhotoMesh should use, ordered based on preferability. During production, PhotoMesh iterates over the elements of the array, beginning with the first instance type, until it finds an available instance. PhotoMesh will always attempt first to launch instances from one of the subnets listed in the Subnet ID (below), before it moves to the next machine type listed. Select based on the steps that require processing and other relevant factors. Click **Edit** to edit the JSON file. You can also open and edit the file in a text editor **See** "Recommendations Based on PhotoMesh Step" in the "Getting Started with AWS" chapter for information. Additional JSON configuration files can be created and placed under ../photomesh/AWSConfigurations where the two example JSON files are also saved. Each instance object in the JSON file's array has three properties: ami, instanceType, and demandType. The ami is the ID of the Amazon Machine Image (AMI). The instanceType defines the hardware of the host computer used for your instance. Each instance type offers different compute, memory, and storage capabilities. The demandType determines the pricing arrangement: on-demand or spot.
Build Priority	Select either **Choose all listed options**, to use all machine types listed in the JSON file, with priority based on the order they are listed in the file, or select a specific machine.
Security Group ID	Select the instance's security group ID. Security groups control inbound and outbound traffic using rules.
Subnet ID	Select the subnet of the virtual private cloud (VPC) dedicated to your AWS account, into which to launch your fuser instances. If you want to use additional subnets, click **Additional Subnets**, and type a list of all the subnet ID's you want to use. **See** "Building a PhotoMesh Project Using the Master Instance" in the "Getting Started with AWS" chapter for more information.
Key Pair Name	The name of the Key Pair used to configure the PhotoMesh AWS fuser Amazon Machine Images (AMI's). Key Pairs are used to encrypt and decrypt login information.

Additional Per Project Fuser Instance Tags

Add tags to help you easily identify a fuser instance in the Amazon EC2 console after it is launched. These tags are saved in the JSON configuration file.

Customer	Name of customer who commissioned the project.
Description	Free text describing the project.
Shutdown inactive fuser instance…	Select one of the options for shutting down inactive fuser instances: ▪ Shut down inactive fuser instances before the next billing hour ▪ Shut down inactive fuser instances after XX minutes

4. If you want to use another account, click **Use another AWS account**, and enter the AWS security information (see step 2 for more information).

5. If you also want to use pool fusers, toggle on the **Use Fusers Pool** link, and enter the required information. **See** "Using a Fusers Pool" in this chapter for information.

6. Click **OK**.

Fusers Pools

The fusers pool is used to share fusers between several PhotoMesh master computers in your organization. When working with a pool, the PhotoMesh master computer's Build Manager requests fusers from the pool, and returns them when they are no longer needed. A distributed management approach is used to manage the pool fusers, whereby each PhotoMesh master computer manages its own fuser needs and resources. Fusers are allocated on a first come first served basis, i.e., to the first PhotoMesh Build Managers that request fusers from the pool. If a manager submits a request for a user, but no fusers are currently available in the pool, the manager continually checks the fusers pool for a fuser that was returned to the pool, at which point it assigns the fuser to its project's working folder.

Fuser allocation can be manually set from the Pool Manager page. From this page, you can monitor and manage all the fusers in the pool, e.g., assign fusers to specific projects' working folders and prevent their being returned to the pool.

Running the Fuser Service

The PMFuserService.exe should be run on all fuser computers in the pool since it is responsible for running the required fuser (PhotoMeshFuser.exe) version for the requesting manager and assigning it to the manager's working folder. This enables different PhotoMesh versions to use the same fusers pool. The PMFuserService.exe can be run from the fuser folder under the installation folder (.\Application Files\Fuser\PhotoMeshFuser.exe) or from a file share or network drive that is accessible to all fuser computers. It is recommended to run the fuser service on startup using auto-logon. **See** "Running Fusers Upon Login" in this chapter for information.

Creating a Fusers Pool

To create a fusers pool:

1. Designate a pool directory which will hold the pool.json file created in the next step as well as the working folders that will be created automatically for each of the projects using the fusers pool.

   Note: If you are also using local fusers for a project, the working folder must be the one automatically created in the pool directory. It is recommended to create the working folder on a fast drive that is different from the one used for the project data, most preferably on a solid-state drive (SSD) that is shared with the master machine and other fuser machines.

2. Create a pool.json file with the name and IP of all fusers in the pool. Use the following structure and syntax in creating the file:

   - Create an array called "Pool" (enclosed in square brackets).
   - Enclose within curly braces each of the fuser objects. Each fuser object is composed of two name/value pairs separated by a comma: Name: FuserName, and IP: FuserIP.

```
{
"Pool": [
        {
            "Name": "pm-020",
            "IP": "http://192.168.204.20:8085"
        },
```

```
        {
            "Name": "pm-021",
            "IP": "http://192.168.204.21:8085"
        }
    ],
    "Name": "ProductionPool"
}
```
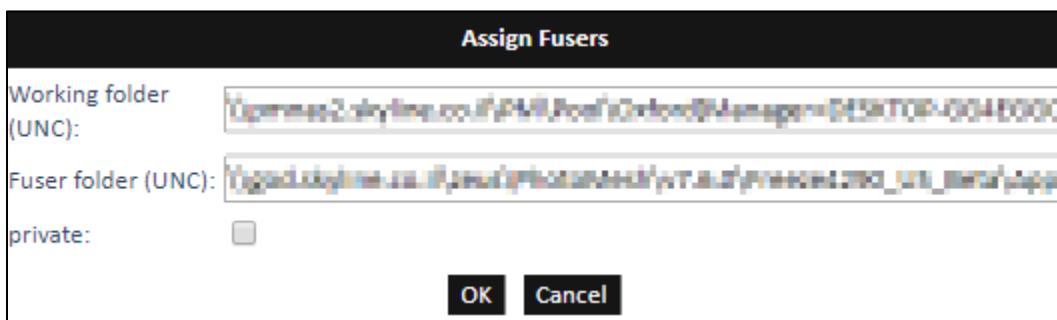
Managing a Fusers Pool

Fusers are allocated on a first come first served basis, i.e., to the first PhotoMesh Build Manager that requests fusers from the pool. If a subsequent manager submits a request for a user, but no fusers are currently available in the pool, the manager continually checks the fusers pool for a fuser that was returned to the pool, at which point it assigns the fuser to its project's working folder. If you want to manually allocate fusers, you can do so from the Pool Manager page. From this page, you can monitor and manage all the fusers in the pool, e.g., assign fusers to specific projects' working folders and prevent their being returned to the pool.

To manage a fusers pool:

1. In the **Build Manager** dialog box, in the Auto Scaling section, click the **Pool Manager** link. The PhotoMesh – Pool Manager page is displayed, with information on all the fusers in the pool. You can also access the Pool Manager from the Tools tab and from the Start page's Automation Dashboard. You can set any of the following for either selected or all fusers:
   - Assign to a project
   - Modify their idle timeout
   - Set private or public status
   - Add a comment

2. If you want to manually assign fusers to a project's working folder, do the following:
   a. Select the fusers' check boxes, and click **Assign**. The Assign Fusers dialog box is displayed.

Assign Fusers

   b. Enter the working folder and fuser folders.

   c. If you want to prevent these fusers from being restored to the pool after completing their current tasks, select the **private** check box.

d. Click **OK.**

3. If you want to modify the idle timeout set for certain fusers, select the fusers' check boxes, and click **Set Idle Time**. Then enter the time in seconds, and click **OK**.
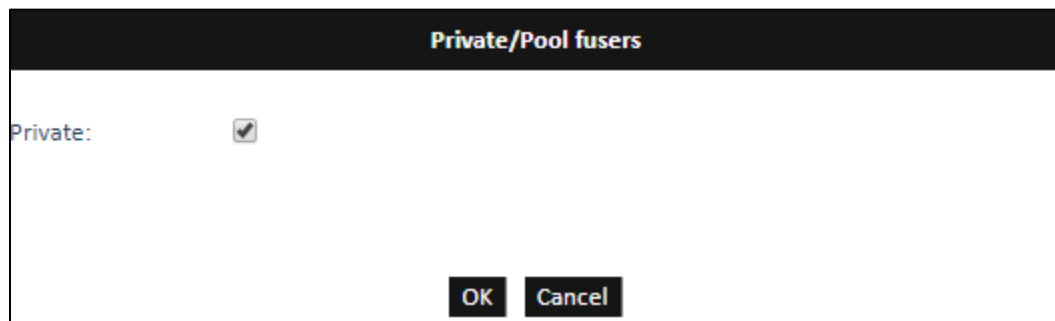
Set Idle Time

4. Fusers can be made available to all master computers in a pool or restricted to a specific project's working folder. If you want to change fusers' availability in the pool, i.e., to either make them available in the pool (public) or restrict their use to a specific project (private), select the fusers' check boxes, and click **Private/Pool**. Then select the private check box to make them private (or clear to make them public), and click **OK**.

Note: Private fusers are not restored to the pool even after completing their current tasks. If you want to assign fusers to a specific project's working folder, but have them returned to the pool after they complete their tasks, clear this check box, and see step 3.

5. If you want to write a comment for certain fusers, select the fusers' check boxes, and click **Comment**. Then type the comment, and click **OK**.

Comment	
Comment:	
	OK Cancel

Using a Fusers Pool

The PhotoMesh master should be run from a shared network drive that is accessible to all fuser computers, to enable the PMFuserService.exe to launch a fuser that is the same version as the master.

To use a fusers pool:

1. Copy the PhotoMesh application files from the PhotoMesh master computer to a shared network drive that is accessible to all fuser computers, to enable the PMFuserService.exe to launch a fuser that is the same version as the master. Then run PhotoMesh from that path on the master computer.

2. In the **Build Manager**, toggle on **Use Fusers Pool**. The **Auto Scaling Fusers Settings** dialog box is displayed. You can also access these settings from the Tools tab (Fuser Pool Settings command).

   Note: The **Automatically use new available fusers** check box must also be selected, so that PhotoMesh will use the pool fusers when they become available.

3. Select the **Use Fusers Pool** check box, and enter the following information:

Fusers Pool Folder	Click **Change** to browse to the location of the pool folder. The pool folder must contain the pool.json file. **See** "Creating a Fusers Pool" in this chapter for more information.
	Note: If you are also using local fusers for a project, the working folder for the entire project, i.e. for both the local and pool fusers must be located in the pool directory.
Fuser Folder	Click **Change** to browse to the location of the fuser folder. This folder is set, by default, to the UNC path of the folder from where PhotoMesh is running so that the Fuser service launches the fuser version required by the master. Make sure that this path is accessible to all the fusers, e.g., by running PhotoMesh from a file share or network drive that is accessible to all fuser computers.
Return inactive fuser to pool after	Set the number of seconds after which inactive fusers should be returned to the pool.
Maximum Fusers	Type the maximum number of fusers to run. If fusers are available in the pool, PhotoMesh Build Manager adds pool fusers up to the maximum number set.
	Note: Make sure your pool limit does not exceed your license fuser limit, to avoid rejection of your fuser request by the pool.

4. If you also want to use AWS fusers, select the **Use AWS Instances** check box, and then enter the required information. **See** "Using Amazon Web Services (AWS) Fusers" in this chapter for information.

5. Click **OK**.

6. If you want to monitor and manage all pool fusers, e.g., assign fusers to specific projects' working folders and prevent their being restored to the pool, in the **Build Manager** dialog box, click the **Pool Manager** link. **See** "Managing a Fusers Pool" in this chapter for information. You can also access the Pool Manager from the Tools tab.

Launching Multiple Fusers

PhotoMesh supports an **experimental** feature of running multiple fusers on powerful machines for potentially enhanced performance. To run multiple fusers, a computer requires a high number of CPU cores (16 or more) and a large amount of RAM (32 GB or more). It is recommended that a minimum of 16 GB RAM and 8 cores be available per fuser. Multiple fusers on the same machine cannot be currently used when using Auto-Scaling AWS instances or a Fusers Pool.

As a general guideline, the optimal number of PhotoMesh fusers per machine can be calculated by taking the lower of – the total number of CPU cores in the machine divided by 8 and the total RAM (in GB) divided by 16 (rounded down). It is highly recommended, however, to perform tests with different numbers of fusers since various factors such as I/O speed, network, memory cache can affect performance.

To launch multiple fusers:

▪ Add a unique ID parameter to the PhotoMeshFuser.exe command line.

Example:

To launch three PhotoMesh fusers in parallel on one machine, run these command lines:

```
>"C:\Program Files\Skyline\PhotoMesh\Fuser\PhotoMeshFuser.exe" 1
>"C:\Program Files\Skyline\PhotoMesh\Fuser\PhotoMeshFuser.exe" 2
>"C:\Program Files\Skyline\PhotoMesh\Fuser\PhotoMeshFuser.exe" 3
```

CHAPTER 15 GETTING STARTED WITH AWS

Overview

PhotoMesh is designed and built to fully exploit computer clusters and cloud computing. Cloud computing provides the flexibility to quickly scale up and down based on resource needs - even within a single project. Projects (or even steps within a single project) with demanding processing requirements can be run simultaneously on hundreds of virtual fuser machines (instances), vastly accelerating mesh model creation, while for less demanding projects (or project steps), users can quickly scale down. Users only pay-per-use, thus avoiding any significant upfront investment or wasted resources.

Amazon Web Services (AWS) is a secure cloud services platform that provides a perfect fit for PhotoMesh users interested in leveraging cloud products for scalable processing power with only minimal investment in infrastructure.

The general workflow involves the following steps:

- Step 1: Setting up an Amazon Virtual Private Cloud (VPC).

- Step 2: Creating the Master PhotoMesh machine, by launching and customizing a Windows instance from an existing Amazon Machine Image (AMI) and setting up storage on Amazon EBS Volumes.

- Step 3: Creating a Linux AMI for a fuser computer by launching and customizing an instance from an existing AMI, and then creating a new AMI from the instance.

- Step 4: Building a PhotoMesh project by using the configured environment to create and then build a project.

The workflow outlined below is intended only as a recommendation for a typical workflow, and it can be modified based on other AWS and IT knowledge and preferences.

Note: The steps below based on Amazon's AWS documentation, adapted for PhotoMesh's specific requirements.

Step 1: Create a Virtual Private Cloud (VPC)

To create a Virtual Private Cloud for PhotoMesh production, configure a VPC and then create a security group.

See: What is Amazon VPC in Amazon's AWS documentation for more information.

Configuring a VPC

1. Open the Amazon VPC console at https://console.aws.amazon.com/vpc/.
2. On the dashboard choose **Launch VPC Wizard**.
3. Choose the first option, **VPC with a Single Public Subnet**, and then choose **Select**.

   Note: If you want to use multiple subnets, they should be configured in this step.

4. On the configuration page, enter the following information:

   a. **VPC name**: *PM VPC.*

   b. **Public subnet's IPv4 CIDR**: Change X.X.X.X/24 to X.X.X.X/22 (E.g., The default 10.0.0.0/22). This allows up to 1019 instances on your network.

   c. **Subnet name**: *PM Subnet.*

5. Click **Create VPC**.
6. In the navigation pane, choose **Subnets**.

7. Select **PM subnet**, choose **Actions**, and then **Modify Auto-Assign IP Settings**.

8. Select the **Enable Auto-assign Public IPv4 address** check box.

9. Click **Save**.

Creating a Security Group

See: Create a Security Group in Amazon's AWS documentation for more information.

1. Open the Amazon VPC console at https://console.aws.amazon.com/vpc/.

2. In the navigation pane, choose **Security Groups**.

3. Choose **Create Security Group**.

4. In the **Security group name** field, enter *PM_SG* as the name of the security group, and type a description.

5. Select the ID of your PM VPC from the **VPC** menu, and then choose **Create**.

6. Select the **PM_SG** security group that you just created (you can view its name in the Group Name column).

7. Click **Actions** and then select **Edit inbound rules.**

8. On the **Inbound Rules** tab, click **Add rule** and input the parameters as follows below. Click **Save Rules** when you're done:

   a. Select **RDP** (Remote Desktop Protocol) from the **Type** list, and enter your network's public IP address range in the **Source** field. If you don't know this address range, you can use 0.0.0.0/0.

   b. Choose **Add Rule**, and select **SSH** from the **Type** list**.** Then enter your network's public IP address range in the **Source** field. If you don't know this address range, you can use 0.0.0.0/0.

   c. Choose **Add Rule**, select **Custom TCP Rule** from the **Type** list, and enter *5900-5901* (default VNC ports) in the **Port Range** field. Then enter your network's public IP address range in the **Source** field. If you don't know this address range, you can use 0.0.0.0/0.

   Note: For the rules above**,** when you use 0.0.0.0/0 for the Source field, you enable all IP addresses to access your instance using RDP, SSH, or VNC. This is suitable for a short exercise, but it is unsafe for production environments. In production, you will want to authorize only a specific IP address or range of addresses to access your instance.

   d. Choose **Add Rule**, select **Custom TCP Rule**, from the **Type** list, and enter *445* in the **Port Range** field. In the **Source** field, start typing *sg*, and select the group ID of your security group.

   Note: You can also add another identical rule for your network's IP address range, to allow file sharing directly from your computer. If you don't know this address range, you can use 0.0.0.0/0, but keep in mind that this will enable all IP addresses to access your instance using file sharing. This is suitable for a short exercise, but it is unsafe for production environments. In production, you will want to authorize only a specific IP address or range of addresses to access your instance.

Step 2: Create a Master Instance

Creating an Amazon Machine Image (AMI) for the master computer is a multi-part step that entails: launching an instance from an existing AMI and customization of the instance.

To create a master Instance, follow the steps below. For more information, **see**: Getting Started with Amazon EC2 Windows Instances in Amazon's AWS documentation. If you are using a hybrid cloud architecture and running your PhotoMesh Manager locally, there is no need to create a master instance.

Creating an IAM Policy and Role

1. Open the IAM console at https://console.aws.amazon.com/iam/.
2. In the navigation pane on the left, choose Policies.
   If this is your first time choosing Policies, the Welcome to Managed Policies page appears. Choose **Get Started**.
3. Choose **Create policy**.
4. Choose the **JSON** tab.
   Note: You can switch between the **JSON** and **Visual editor** tabs any time and add each of the permissions below individually.
5. Paste the following JSON policy document:

```
{
  "Version": "2012-10-17",
  "Statement": [
    {
      "Sid": "VisualEditor0",
      "Effect": "Allow",
      "Action": [
        "ec2:DescribeInstances",
        "ec2:TerminateInstances",
        "ec2:RequestSpotInstances",
        "ec2:DescribeTags",
        "ec2:CreateTags",
        "ec2:DescribeRegions",
        "ec2:RunInstances",
        "s3:ListBucket",
        "ec2:DescribeSpotInstanceRequests",
        "ec2:DescribeSecurityGroups",
        "ec2:DescribeSpotPriceHistory",
        "ec2:DescribeImages",
        "ec2:CancelSpotInstanceRequests",
        "s3:GetObject",
        "iam:CreateServiceLinkedRole",
        "ec2:DescribeSubnets",
        "ec2:DescribeKeyPairs",
        "ec2:DescribeInstanceStatus"
      ],
      "Resource": "*"
    }
  ]
}
```

6. When you are finished, choose **Review policy**.

7. On the **Review policy** page, enter *PM_RUNFUSER_POLICY*. Review the policy **Summary** to see the permissions that are granted by your policy. Then choose **Create policy** to save your work.

8. In the navigation pane, choose **Roles**, and then choose **Create role**.

9. For **Select type of trusted entity**, choose **AWS service**.

10. For **Choose a use case**, choose **EC2**. Then choose **Next: Permissions**.

11. In the list of policies, select the **PM_RUNFUSER_POLICY** policy. You can use the Filter menu to filter the list of policies.

12. Choose **Next: Tags**.

13. Choose **Next: Review**.

14. In the **Role name**, enter *PM_RUNFUSER_ROLE*.

15. Review the role and then choose **Create role**.

Launching an Initial Instance and Creating Storage

To launch an initial instance and create storage:

1. Open the Amazon EC2 console at https://console.aws.amazon.com/ec2/.

2. From the Amazon EC2 console dashboard, choose **Launch Instance**.

3. On the **Choose an Amazon Machine Image (AMI)** page, choose the following AMI from the **AWS Marketplace** list: **NVIDIA Gaming PC - Windows Server 2019**, and then choose **Select**.

4. On the **Choose an Instance Type** page, select the **g4dn.xlarge** type, and choose **Next: Configure Instance Details**.

   Note: Only instance types that match the NVIDIA AMI are available. If you want to use a different instance type, select a different AMI in step 3 of "Launching an Initial Instance and Creating Storage", e.g., Windows Server 2019. Make sure to install any graphics drivers that are required for your instance.

   Note: Using a GPU instance is recommended for normal operation of PhotoMesh Master over RDP connection. For productions with hundreds of fusers, other types can be used such as g4dn.8xlarge, which has faster network performance. For more information, see Instance Types in Amazon's AWS documentation.

5. On the **Configure Instance Details** page, enter the following information:

   a. **Network**: Choose the **PM VPC** VPC created in step 4 of "Configuring a VPC".

   b. **IAM role**: Choose the **PM_RUNFUSER_ROLE** created in step 14 of "Creating an IAM Policy and Role".

   c. **Enable termination protection**: Select the check box.

   d. **Network Interfaces**: Set the **Primary IP** of **eth0** to *10.0.0.10*.

   e. Choose **Next: Add Storage**.

6. On the **Add Storage** page, enter the following information:

   a. Keep the default volume types.

   b. Click **Add New Volume**. Then set **Size** based on your expected need (E.g. 100 GiB). This is the EBS storage that will hold your data, projects, PM files, etc.

      Note: The EBS volumes will incur additional charges. For more information, see Amazon EBS Volumes in Amazon's AWS documentation.

      c.   **Volume Type:** Choose a volume type based on your expected need (e.g., General Purpose SSD (gp2)). For more information, see [Amazon EBS Volume Types](#).

      d.   Choose **Next: Add Tags**.

7. On the **Add Tags** page, tag your instance to help you identify it in the Amazon EC2 console after you launch it. Enter the following information:

      a.   Select Add Tag.

      b.   Key: *Name.*

      c.   Value: *PM Master.*

      d.   Click Next: Configure Security Group.

8. On the **Configure Security Group** page, do the following:

      a.   Choose Select an existing security group.

      b.   Select the PM_SG group.

      c.   Choose Review and Launch.

9. On the **Review Instance Launch** page, check the details of your instance, and make any necessary changes by clicking the appropriate Edit link. When you are ready, choose **Launch**.

10. In the **Select an existing key pair or create a new key pair** dialog box, create a new key pair.

      a.   Choose Create a new key pair.

      b.   Key pair name: *PM_KeyPair.*

      c.   Choose Download Key Pair.

          Note: Store the file in a secure and accessible location since you will need the contents of the private key to connect to your instance after it is launched.

      d.   Choose Launch Instances.

11. To launch your instance, select the acknowledgment check box, and then choose **Launch Instances**.

Connecting to the Instance

See: [Connecting to Your Windows Instance Using RDP](#) in Amazon's AWS documentation for more information.

1. Open the Amazon EC2 console at https://console.aws.amazon.com/ec2/

2. In the navigation pane, choose **Instances.**

3. Select the instance, and then choose **Connect**.

4. In the **Connect To Your Instance** dialog box, choose **Get Password** (it will take a few minutes after the instance is launched before the password is available).

5. Choose **Choose File** and navigate to the private key file you created when you launched the instance. Select the file and choose **Open** to copy the entire contents of the file into the contents box.

6. Choose **Decrypt Password**. The console displays the default administrator password for the instance in the **Connect To Your Instance** dialog box, replacing the link to **Get Password** shown previously with the actual password.

7. Record the default administrator password, or copy it to the clipboard. You need this password to connect to the instance.

8. Choose **Download Remote Desktop File**. Your browser prompts you to either open or save the .rdp file. Either option is fine. When you have finished, you can choose **Close** to dismiss the **Connect To Your Instance** dialog box.

9. You may get a warning that the publisher of the remote connection is unknown. Choose **Connect** to connect to your instance.

10. When prompted, log in to the instance, using the administrator account for the operating system and the password that you recorded or copied previously. If your **Remote Desktop Connection** already has an administrator account set up, you might have to choose the **Use another account** option and enter the user name and password manually.

11. Due to the nature of self-signed certificates, you may get a warning that the security certificate could not be authenticated. Choose **Yes** or **Continue** to continue.

12. A Windows **Networks** message is displayed asking if "you want to allow your PC to be discoverable by other PCs and devices on this network?" Click **Yes**.

13. After you connect, we recommend that you change the administrator password from the default value. You change the password while logged on to the instance itself, just as you would on any other Windows Server.

    Note: Due to the Remote Desktop Protocol (RDP), the recommended method for changing the password is to press CTRL+ALT+END, and then select **Change a password**.

Customizing the Instance

See: Making an Amazon EBS Volume Available for Use on Windows and Windows GPU Instances in Amazon's AWS documentation for more information.

1. While connected to the Master instance using RDP, start **Windows File Explorer**.

2. Start the Disk Management utility. On the taskbar, open the context (right-click) menu for the Windows logo and choose **Disk Management**.

3. Open the context (right-click) menu for the right panel for Disk 1 and choose **New Simple Volume**. Complete the wizard with the default settings.

4. Give share permissions with **Full Control** to **Everyone** on the *D* drive.

5. Install PhotoMesh using the standard installation to *D:\PhotoMesh*.

6. Create the folder *D:\PMWorkingFolder,* and then then the subfolder *A* directly below it, so that you have the directory structure: *D:\PMWorkingFolder\A*.

7. Create the folder *D:\PMProjects*.

8. Start PhotoMesh from "\\10.0.0.10\D\PhotoMesh\PhotoMesh.exe".

9. Click the **PhotoMesh** button, and then click **Options (F9)**.

10. Change the **Working Folder** to *\\10.0.0.10\D\PMWorkingFolder\A*.

11. Click **OK** and then **close** PhotoMesh.

Step 3: Create a Fuser AMI (Linux)

Creating a Linux AMI for a fuser computer is a multi-part step that entails: launching an instance from an existing AMI, customizing the instance, and finally creating a new AMI from the instance.

To create a fuser AMI, follow the steps below. For more information, **see**: Getting Started with Amazon EC2 Linux Instances in Amazon's AWS documentation.

Linux fuser AMI's are generally recommended over Windows AMI's since Linux instances are more cost effective to run.

Launching an Initial Instance

1. Open the Amazon EC2 console at https://console.aws.amazon.com/ec2/.

2. From the Amazon EC2 console dashboard, choose **Launch Instance**.

3. On the **Choose an Amazon Machine Image (AMI)** page, choose the following AMI from the **Quick Start** list: **Ubuntu Server 18.04 LTS (HVM), SSD Volume Type**, and then choose **Select**.

4. On the **Choose an Instance Type** page, select the **g4dn.xlarge** type, and choose **Next: Configure Instance Details**.

   Note: The instance type selected here depends on the fusers you intend to use to build the project. The **g4dn.xlarge** instance type should be used to create an AMI for G4 and G3 instances.

5. On the **Configure Instance Details** page, enter the following information:

   a. **Network**: Choose the **PM VPC** VPC created in step 4 of "[Configuring a VPC](#)".

   b. **IAM Role**: Choose the IAM role "PM_RUNFUSER_POLICY" created in "[Creating an IAM Policy and Role](#)".

   c. Choose **Next: Add Storage**.

6. On the **Add Storage** page, enter the following:

   a. Change the default **Root** volume **Size** to at least *16* GiB.

      Note: The EBS volumes will incur additional charges. For more information see [Amazon EBS Volumes](#) in Amazon's AWS documentation.

   b. Choose Next: Add Tags.

7. On the **Add Tags** page, enter the following information:

   a. Select **Add Tag**.

   b. **Key**: *Name.*

   c. **Value**: *PM Linux Fuser Initial.*

   d. Choose **Next: Configure Security Group** when you are done.

8. On the Configure Security Group page:

   a. Choose **Select an existing security group**.

   b. Select the **PM_SG** group.

   c. Choose **Review and Launch**.

9. On the **Review Instance Launch** page, review the details of your instance, and make any necessary changes by clicking the appropriate Edit link. When you are finished, choose **Launch**.

10. In the **Select an existing key pair or create a new key pair** dialog box:

    a. Choose **Choose an existing pair**.

    b. **Key pair name**: *PM_KeyPair.*

    c. Choose **Launch Instances**.

11. To launch your instance, select the acknowledgment check box, and then choose **Launch Instances**.

Convert Your Private Key Using PuTTYgen

See: [Connecting to Your Linux Instance from Windows Using PuTTY](#) in Amazon's AWS documentation for more information.

Note: For information on additional ways to connect to your Linux instance, see:
https://docs.aws.amazon.com/AWSEC2/latest/UserGuide/AccessingInstances.html

1. Download and install PuTTY from the [PuTTY download page.](#) If you already have an older version of PuTTY installed, we recommend that you download the latest version. Be sure to install the entire suite.

2. From the **Start** menu, choose **All Programs**, **PuTTY**, **PuTTYgen**.

3. Under **Type of key to generate**, choose **RSA**. If you're using an older version of PuTTYgen, choose **SSH-2 RSA**.

4. Choose **Load**. By default, PuTTYgen displays only files with the extension .ppk. To locate your .pem file, choose the option to display files of all types.

5. Select your .pem file for the key pair that you specified when you launched your instance and choose **Open**. PuTTYgen displays a notice that the .pem file was successfully imported. Choose **OK**.

6. To save the key in the format that PuTTY can use, choose **Save private key**. PuTTYgen displays a warning about saving the key without a passphrase. Choose **Yes**.

   Note: A passphrase on a private key is an extra layer of protection. Even if your private key is discovered, it can't be used without the passphrase. The downside to using a passphrase is that it makes automation harder because human intervention is needed to log on to an instance, or to copy files to an instance.

7. Specify the same name for the key that you used for the key pair (for example, *PM_KeyPair*) and choose **Save**. PuTTY automatically adds the .ppk file extension.
   Your private key is now in the correct format for use with PuTTY. You can now connect to your instance using PuTTY's SSH client.

Connecting to the Instance

To connect:

1. Start PuTTY (from the **Start** menu, choose **All Programs**, **PuTTY**, **PuTTY**).

2. In the **Category** pane, choose **Session** and complete the following fields:

   a. In the Host Name box, to connect using your instance's public DNS, enter *ubuntu@public_dns_name*.
   You can get the public DNS for your instance using the Amazon EC2 console. Check the **Public DNS (IPv4)** column. If this column is hidden, choose the **Show/Hide** icon and select **Public DNS (IPv4)**.

   b. Ensure that the **Port** value is *22*.

   c. Under **Connection type**, select **SSH**.

3. (Optional) You can configure PuTTY to automatically send 'keepalive' data at regular intervals to keep the session active. This is useful to avoid disconnecting from your instance due to session inactivity. In the **Category** pane, choose **Connection**, and then enter the required interval in the **Seconds between keepalives** field. For example, if your session disconnects after 10 minutes of inactivity, enter *180* to configure PuTTY to send keepalive data every 3 minutes.

4. In the **Category** pane, expand **Connection**, expand **SSH**, and then choose **Auth**. Complete the following:

   a. Choose **Browse**.

   b. Select the .ppk file that you generated for your key pair and choose **Open**.

   c. (Optional) If you plan to start this session again later, you can save the session information for future use. Under **Category**, choose **Session**, enter a name for the session in **Saved Sessions**, and then choose **Save**.

d.   Choose **Open**.

5.   If this is the first time you have connected to this instance, PuTTY displays a security alert dialog box that asks whether you trust the host to which you are connecting. Choose **Yes**. A window opens and you are connected to your instance.

Customizing the Instance

See: Linux Accelerated Computing Instances in Amazon's AWS documentation for more information.

1.   To download and install the Nvidia drivers and update the operating system, in the PuTTY terminal window, type:

   a.   wget http://us.download.nvidia.com/tesla/440.33.01/NVIDIA-Linux-x86_64-440.33.01.run

   b.   sudo apt-get update

   c.   sudo apt-get -y install gcc make xserver-xorg libglu1-mesa-dev freeglut3-dev mesa-common-dev libxmu-dev libxi-dev mesa-utils x11vnc xfce4 dkms xvfb

   d.   sudo apt-get remove xscreensaver
   Type **Y** when asked to continue.

   e.   sudo apt-get remove xscreensaver-data
   Type **Y** when asked to continue.

   f.   chmod +x NVIDIA-Linux-x86_64-440.33.01.run

   g.   sudo ./NVIDIA-Linux-x86_64-440.33.01.run
   Click **Yes** when asked to register the kernel module. Click **OK** when prompted to do so during the installation.

   h.   *sudo reboot*
   The following message is displayed: "Remote side unexpectedly closed network connection".
   Click **OK**.

2.   Right-click the title bar of the PuTTY window, and select **Restart Session**.

3.   To configure the GPU and start the X window system, in the PuTTY terminal window, type:

   a.   nvidia-xconfig --query-gpu-info | grep BusID
   Copy the value that is outputted. E.g., if the output is "PCI BusID : PCI:0:30:0", select and copy "*PCI:0:30:0*".

   b.   *sudo nvidia-xconfig -a --virtual=1280x1024 --busid=<BUSID>*
   Where <BUSID> is replaced with the value from 2c above, e.g., sudo nvidia-xconfig -a --virtual=1280x1024 --busid=**PCI:0:30:0**.
   Note: If a warning is displayed that the xconfig file was not located/opened, click **OK**.

   c.   sudo /usr/bin/X :0 &

   d.   export DISPLAY=:0

4.   (Optional) To configure VNC in order to verify X runs properly, in the PuTTY terminal window type:

   a.   x11vnc -storepasswd
   Then type and verify the password

   b.   *y*
   This confirms writing the password to /home/Ubuntu/.vnc/passwd.

   c.   x11vnc -rfbauth ~/.vnc/passwd &

   d.   xfce4-session &

5. (Optional) Connect with VNC client using your instance's public DNS and the password set in step 4 above.
   You can get the public DNS for your instance using the Amazon EC2 console. Check the **Public DNS (IPv4)** column. If this column is hidden, choose the **Show/Hide** icon and select **Public DNS (IPv4)**.

6. To install and configure Wine, do the following:

   a. Open the **Start** menu and search for **cmd** to start a command prompt.

   b. Type *pscp -i <.ppk file> <wineinst.tar full path> ubuntu@<fuser IP>:/home/ubuntu/wineinst.tar*
      Where <.ppk file> is replaced with a full path to the ppk file created in "Convert Your Private Key Using PuTTYgen"; <wineinst.tar full path> is replaced with the full path to the wineinst.tar that is included in the zip folder that can be downloaded from here; and <fuser IP> is replaced with the Public DNS (IPv4) of the instance.

   c. In the PuTTY terminal window, type:
      i. tar -xvf wineinst.tar
      ii. sh fuser.sh

      Note: This process can take a few minutes to complete.

7. In the Amazon EC2 console, right-click the instance, point to **Instance State**, and select **Stop.**

8. After the instance has stopped, right-click the instance, point to **Instance Settings**, and select **Change Instance Type**. The Change Instance Type dialog box is displayed.

9. From the Instance Type drop-down list, select ***g4ad.4xlarge***.

10. Restart the instance by right-clicking the instance in the Amazon EC2 console, pointing to **Instance State**, and selecting **Start.**

11. Reconnect to the instance via PuTTY by performing the steps in "Connecting to the Instance".

12. To download and install AMD drivers, in the PuTTY terminal window type:

   a. sudo apt install awscli
      i. Y to continue

   b. aws s3 cp --recursive s3://ec2-amd-linux-drivers/latest/ .

   c. tar -xf amdgpu-pro*ubuntu*.xz

   d. sudo apt install linux-modules-extra-$(uname -r) -y

   e. Change to the folder of the extracted driver:
      i. Type the following (without pressing ENTER): *cd amd*
      ii. Press tab once.
      iii. Press ENTER.

   f. cat RPM-GPG-KEY-amdgpu | sudo apt-key add -

   g. ./amdgpu-pro-install -y --opencl=pal,legacy

   h. sudo reboot

13. The following message is displayed: "Remote side unexpectedly closed network connection". Click **OK**.

14. Right-click the title bar of the PuTTY window, and select **Restart Session**.

15. Test that the installation was successful:

   a. dmesg | grep amdgpu

b. Check the response and look for "Initialized amdgpu". This indicates that the installation was successful.

Saving a Fuser AMI

See: [Creating an Amazon EBS-Backed Linux AMI](#) in Amazon's AWS documentation for more information.

1. In the navigation pane, choose Instances and select the **PM Linux Fuser Initial instance**. Choose **Actions, Image,** and **Create Image**.

2. In the **Create Image** dialog box, specify values for the following fields, and then choose **Create Image**.

    ▪ **Image name**: *PM Linux Fuser Image*.

3. While your AMI is being created, you can choose **AMIs** in the navigation pane to view its status. Initially, this is pending. After a few minutes, the status should change to available.

4. Record the AMI ID of the created AMI, or copy it to the clipboard. You need this AMI ID for PhotoMesh to launch fuser instances.

5. In the navigation pane, choose **Instances** and select the **PM Linux Fuser Initial** instance.

6. Choose **Actions**, select **Instance State**, and then choose **Terminate**.

Step 4: Build a PhotoMesh Project

This step outlines the workflow for building a PhotoMesh project. The option to run the entire PhotoMesh production with only cloud resources is the main workflow discussed above, and then in "Building a PhotoMesh Project Using the Master Instance" below. It is also possible to apply a hybrid cloud architecture that mixes cloud and on-premises computing power and storage solutions. This option is covered in "Creating an IAM User for Running PhotoMesh Project Using an On-Premises Master" and "Building a PhotoMesh Project Using an On-Premises Master and Storage".

There are a number of variables involved in the selection of fuser instances to process a build:

- **CPU/GPU** – The hardware of the host computer used for your instance determines its compute, memory, storage, and graphic capabilities. GPU-based instances provide access to NVIDIA GPUs to accelerate processing. The need for GPUs is generally determined based on build step. See the table below.

- **Price/Performance** – Optimized balance between machine price and its computing power. See here for more information: https://aws.amazon.com/ec2/pricing/. See the table below for Skyline recommendations (based on our experience in the US East – N. Virginia region).

- **Availability Zones-Subnets** – The availability of the required number of instances and their price should be considered when selecting an availability zone. Since you are also charged for data transfer from one availability zone to another, if you require many fusers, it is recommended to launch the master in an availability zone with good spot availability to avoid significant cross-availability zone charges. A subnet must be created for each availability zone. For more information, consult the Amazon EC2 console as described below.

- **On-Demand/ Spot** – On-demand instances provide guaranteed compute resources, while spot instances are dependent on the availability of unused instances: spot instances will only be launched if an instance is available and instance use may be interrupted unexpectedly if capacity is no longer available. On-demand instances also have a static price, while spot instance price varies based on demand. For more information see Amazon EC2 Using Spot Instances in Amazon's AWS documentation.

Analyzing the price history of spot instances in Amazon EC2 console's **Pricing History** can help you determine availability of different spot instance types in different availability zones. Price is generally determined by the law of supply and demand, so if the price for a particular instance type is low in a specific zone, this usually indicates higher availability of spot instances of that type in the zone. No price is listed for a particular zone if there are no spot instances of that specific machine type in this zone.

See the chart below for our recommendations for each PhotoMesh step.

Recommendations Based on PhotoMesh Step

PhotoMesh Step	I/O (Disk IOPS and Network bandwidth)	CPU	GPU	Recommended # of fusers / 1Gbps	Recommended Machine Types	Preferred Method for Hybrid
Data Preparation	Intensive	Intensive	Low	5	M5dn.4xlarge M5d.4xlarge M5.4xlarge	Local
AT	Intensive	Intensive	No	20	M5dn.4xlarge M5d.4xlarge	Local

					M5.4xlarge	
Point Cloud	Low	High	No	200	M6i.2xlarge	AWS
					M5.2xlarge	
					M5dn.2xlarge	
Model	Low	High	No	200	M6i.2xlarge	AWS
					M5.2xlarge	
					M5dn.2xlarge	
Texture Mesh	High	Low	High	50	G4ad.4xlarge	AWS
					G4dn.4xlarge	
					G3.4xlarge	
					G4dn.2xlarge	
Output	High	Low	High	20	G4ad.4xlarge	Local
					G4dn.4xlarge	
					G3.4xlarge	
					G4dn.2xlarge	

Building a PhotoMesh Project Using the Master Instance

To use the configured environment to produce PhotoMesh projects, follow the steps below:

1. Connect to the **PM Master** instance.

   Note: If the PM Master instance is not already running, choose **Actions**, select **Instance State**, and then choose **Start**.

2. Create the directory: "D:\PhotoMesh\AWSConfigurations", and then copy the JSON files ("DataPrep_AT.json", "Reconstruction_CPU.json", and "Reconstruction_GPU.json") from the zip file downloaded in step 6b of "Customizing the (Fuser) Instance" to this directory.

3. Start **PhotoMesh**, and **create** a project.

   Note: The project must be created in a location that is accessible to both master and fuser instances. It is recommended to organize all projects and sources in the \\10.0.0.10\D\PMProjects\ directory.

4. On the **Home** tab, in the **Build** group, click **Build**, then enter the Build Parameters and click **Build**.

5. In the **PhotoMesh Build Manager** dialog box, toggle on **Use AWS Instances (Change)**.

6. In the **Auto-Scaling Fusers Settings** dialog, do the following:

   a. Check the **Use AWS Instances** check box.

   b. **Maximum Price Per Spot Instance:** Set the maximum price you want to pay per EC2 spot instance. For more information, see Amazon EC2 Spot Instance Pricing in Amazon's AWS documentation.

   c. **Maximum Instances:** Set the number based on the number of EC2 instances you want to run.

   d. **Launch Rate:** Set the maximum number of fuser instances to launch per minute, e.g. if Maximum Instances was set to 200, and the Launch Rate to 50, the Build Manager will spread the launch of the fusers over 4 minutes. This prevents a potential bottleneck that could be caused by a large number of fusers attempting to read the build data at the same time.

   e. **Terminate AWS instances when build finished:** Select the check box if you want to terminate AWS fuser instances when the build is complete.

    f.   **Build Priority:** Select either **Choose all listed options**, to use all machine types listed in the JSON file, with priority based on the order they are listed in the file, or select a specific machine.

    g.   **Security Group ID:** Select **PM_SG.**

    h.   **Subnet ID:** Click **Additional Subnets...** and type a list of all the subnet ID's you want to use. These were created in "Configuring a VPC". A subnet must be created for each availability zone. Since you are charged for data transfer from one availability zone to another, it is recommended to launch the master in an availability zone with good spot availability. PhotoMesh will always attempt first to launch instances from the same availability zone as the master, and only if no instances are available, will it launch an instance from a different zone. See the overview to step 4 for some of the issues to consider when selecting a subnet/availability zone.

    i.   **Key Pair Name:** Select **PM_KeyPair**.

    j.   **Configuration Name:** Select based on the current processing step. Select the Reconstruction_GPU option if you want to run an entire build without needing to reselect the configuration before each build step, since this option can be used for all build steps.

         ▪ Reconstruction_GPU – For texture and output steps.

         ▪ AT_MultiCoreCPU – Multi-core CPU for data preparation and aerotriangulation steps.

         ▪ Reconstruction_CPU – For point cloud and model steps.

        Note: These configuration options are read from the JSON files copied in step 2 above. Each of these JSON files contains an array of instance types that PhotoMesh should use, ordered based on preferability. During production, PhotoMesh iterates over the elements of the array, beginning with the first instance type, until it finds an available instance. Each instance object in the array has three properties: ami, instanceType, and demandType. The instanceType defines the hardware of the host computer used for your instance. Each instance type offers different compute, memory, and storage capabilities. The demandType determines the pricing arrangement: on-demand or spot. See the overview to step 4 for some of the issues to consider when selecting instance types.

    k.   Click **Edit** (next to **Configuration Name**), insert in all fields the AMI ID of the PM Linux Fuser Image (created in "Saving a Fuser AMI"), and then click **OK**.

    l.   Select **Shutdown inactive fuser instances after** and set its value to "1".

    m.   Click **Edit Startup Script** and paste the template script from the zip file downloaded in step 6b of "Customizing the (Fuser) Instance" (*Startup_Script_awsQG_Template*). Edit the script so it includes the username and the password of your PM Master machine set in step 13 of "Connecting to the (Master) Instance" 13(In the script, replace all <pass> and <user> placeholders with the required value.

    n.   Click **OK**.

7. Click **Build**.

    Note: If a message is displayed that "PhotoMesh's fuser request was rejected by AWS, since it exceeded the instance limit", you need to request a quota increase for "Spot instance requests". **See**: Amazon EC2 Service Limits in Amazon's AWS documentation for more information.

8. In the **PhotoMesh Build Manager** dialog box, you can monitor the creation and automatic use of the automatically launched fuser instances. You can also monitor the creation of your instances by choosing **Instances** in the navigation pane of the Amazon EC2 console, and reviewing the custom tags added to PhotoMesh launched instances: Build start time, Owner, Project, Type, and Working folder.

9. When processing is complete, stop the PM Master instance.

Note: Do not terminate the master instance; only stop and start as needed.

Note: It is recommended to verify that the instances are terminated according to the settings defined in the **AWS Cloud Settings** dialog, particularly the first time they are terminated.

Creating an IAM User for Running PhotoMesh Project Using an On-Premises Master

To allow PhotoMesh Master running on an on-premises computer to launch EC2 instances, create an IAM user and assign permissions.

See: Creating IAM Users (Console) in Amazon's AWS documentation for more information.

1. Open the Amazon IAM console at https://console.aws.amazon.com/iam/.
2. In the navigation pane, choose **Users** and then choose **Add user**.
3. Type the user name for the new user.
4. Select the **Programmatic access** check box. Then choose **Next: Permissions**.
5. On the **Set Permissions** page, choose **Attach existing policies to user directly**.
6. In the list of policies, select the **PM_RUNFUSER_POLICY** policy. You can use the Filter menu to filter the list of policies.
7. Choose **Next: Tags**.
8. Choose **Next: Review**.
9. Review the role and then choose **Create user**.
10. To view the users' access keys (access key IDs and secret access keys), choose **Show** next to each password and access key that you want to see. To save the access keys, choose **Download .csv** and then save the file to a safe location.

    Note: This is your only opportunity to view or download the secret access keys. Save the user's new access key ID and secret access key in a safe and secure place. You will not have access to the secret keys again after this step.

Building a PhotoMesh Project Using an On-Premises Master and Storage

Before beginning a build using an on-premises master and storage, ensure that your fuser machines have access to your local storage, e.g., via AWS Direct Connect, Site-to-Site VPN, or Security Group and firewall rules. Sufficient bandwidth for the data transfer between your on-premises storage and the fuser machines is significantly affects production performance. See the overview to step 4 for some of the issues to consider regarding the hybrid architecture.

To use the configured environment to produce PhotoMesh projects using on-premises storage and a PhotoMesh that is installed on an on-premises computer while utilizing AWS fuser instances, follow the steps below:

1. Make sure your on-premises computer has PhotoMesh installed.

    Note: File access can be achieved in different ways, e.g. via AWS Direct Connect, Site-to-Site VPN, or Security Group and firewall rules. Consult AWS documentation, support, your IT person, or Skyline support for more information.

2. Create the directory: ".\PhotoMesh\AWSConfigurations", and copy the JSON files ("DataPrep_AT.json", "Reconstruction_CPU.json", and "Reconstruction_GPU.json") from the zip file downloaded in step 6b of "Customizing the (Fuser) Instance" to this directory.

3. Start **PhotoMesh** and **create** a project.

    Note: The project must be created in a location that is accessible to all fuser instances.

4. On the **Home** tab, in the **Build** group, click **Build**, then enter the Build Parameters and click **Build**.

5. In the **PhotoMesh Build Manger** dialog box, toggle on **Use AWS Instances (Change)**.

6. In the **AWS Security** dialog, enter your AWS credentials created in "Creating an IAM User for Running PhotoMesh Project Using an On-Premises Master".

7. In the **Auto-Scaling Fusers Settings** dialog, do the following:

   a. Check the **Use AWS Instances** check box.

   b. **Maximum Price Per Spot Instance:** Set the maximum price you want to pay per EC2 spot instance. For more information see Amazon EC2 Spot Instance Pricing in Amazon's AWS documentation.

   c. **Maximum Instances:** Set the number based on the number of EC2 instances you want to run.

   d. **Launch Rate:** Set the maximum number of fuser instances to launch per minute, e.g. if Maximum Instances was set to 200, and the Launch Rate to 50, the Build Manager will spread the launch of the fusers over 4 minutes. This prevents a potential bottleneck that could be caused by a large number of fusers attempting to read the build data at the same time.

   e. **Terminate AWS instances when build finished:** Select the check box if you want to terminate AWS fuser instances when the build is complete.

   f. **Build Priority:** Select either **Choose all listed options**, to use all machine types listed in the JSON file, with priority based on the order they are listed in the file, or select a specific machine.

   g. **Security Group ID:** Select **PM_SG.**

   h. **Subnet ID:** Click **Additional Subnets...** and type a list of all the subnet ID's you want to use. These were created in "Configuring a VPC". A subnet must be created for each availability zone. See the overview to step 4 for a comparison of on-demand/spot instances and information on using pricing history to determine instance availability.

   i. **Key Pair Name:** Select **PM_KeyPair**.

   j. **Configuration Name:** Select based on the current processing step. Select the Reconstruction_GPU option if you want to run an entire build without needing to reselect the configuration before each build step, since this option can be used for all build steps.
      - Reconstruction_GPU – For texture and output steps.
      - AT_MultiCoreCPU – Multi-core CPU for data preparation and aerotriangulation steps.
      - Reconstruction_CPU – For point cloud and model steps.

      Note: These configuration options are derived from the JSON files copied in step 2 of "Building a PhotoMesh Project Using the Master Instance". Each of these JSON files contains an array of instance types that PhotoMesh should use, ordered based on preferability. During production, PhotoMesh iterates over the elements of the array, beginning with the first instance type, until it finds an available instance.
      Each instance object in the array has three properties: ami, instanceType, and demandType. The instanceType defines the hardware of the host computer used for your instance. Each instance type offers different compute, memory, and storage capabilities. The demandType determines the pricing arrangement: on-demand or spot. See the overview to step 4 for some of the issues to consider when selecting instance types.

   k. Click **Edit** (next to **Configuration Name**), insert in all fields the AMI ID of the PM Linux Fuser Image (created in "Saving a Fuser AMI"), and then click **OK**.

   l. Select **Shutdown inactive fuser instances after** and set its value to "1".

   m. Click **Edit Startup Script** and paste the template script from the zip file downloaded in step 6b of "Customizing the (Fuser) Instance (*Startup_Script_Template.txt*). Edit the script so it includes the username and the password of your PM Master machine set in step 13 of "Connecting to the

(Master) Instance" (In the script, replace all <serverip >, <share>, <pass> and <user> placeholders with the required value).

    n.   Click **OK**.

8. Click **Build**.

    Note: A message is displayed informing you that no available fusers are currently running and asking if you want to start a local fuser. If you want to start a fuser on the master machine to take advantage of its computing resources, click **Yes**. If you are running many spot fuser instances, however, it is recommended to click **No** to free up computing resources for management of the build and file server tasks.

    Note: If a message is displayed that "PhotoMesh's fuser request was rejected by AWS, since it exceeded the instance limit", you need to request a limit increase for "Spot instance requests". **See**: Amazon EC2 Service Limits in Amazon's AWS documentation for more information.

9. In the **PhotoMesh Build Manger** dialog box, you can monitor the creation and automatic use of the automatically launched fuser instances. You can also monitor the creation of your instances by choosing **Instances** in the navigation pane of the Amazon EC2 console, and reviewing the custom Tags added to PhotoMesh launched instances: Build start time, Owner, Project, Type, and Working folder.

10. It is recommended to verify that the instances are terminated according to the settings defined in the **AWS Cloud Settings** dialog, particularly the first time they are terminated.

CHAPTER 16   BUILDING

About Building

After all photos are added to the project and all properties are set as required, you can begin the build process. The build process provides flexibility with regard to the AT area, reconstruction area, build steps that are to be completed, and output formats.

The build process entails a series of sequential steps beginning with data preparation and aerotriangulation and followed by three reconstruction steps: point cloud creation, model creation, and model texturing, and then project output. The required build steps are selected in the Build Parameters dialog box. The final output can be created in any of the following output formats: 3DML (for loading directly into TerraExplorer and streaming with SkylineGlobe Server), DAE, OBJ, 3D Tiles, I3S/SLPK, OSGB, Orthophoto, DSM, DTM/DEM, and LAS (Point cloud public file format). **See** "Setting Output Formats" in this chapter for information on the different file formats.

As the build process is progressing, the 3D Window displays the interim build results and PhotoMesh Build Manager displays information about the current step and active fusers.

If a previously built project requires modification, instead of building the entire project again from the first step of the build process, you have the option to rebuild, incorporating some of a previous build's results. Rebuilding provides flexibility both with regard to the project section (entire project, only error tiles or selected tiles) and project steps that are to be rebuilt. **See** "Rebuilding" in this chapter for more information.

Recommended Aerotriangulation Workflow

For optimal AT results, it is recommended to initially perform only the aerotriangulation step ("AT Only"), and then review the results, to ensure a precise AT which includes all project's photos that intersect with the AT area, before proceeding with the rest of the build. If the results of the AT Only were in fact satisfactory, you can continue with the rest of the build process. PhotoMesh will automatically determine what processing is required to apply the changes to the project, by rebuilding the project from "Auto". **See** "Rebuilding" in this chapter for more information.

If the results of the initial AT Only build were unsatisfactory, modify the AT parameters or collection properties and/or edit/add control points based on the aerotriangulation result. Then repeat the aerotriangulation by selecting AT Only, and review your results. When repeating the aerotriangulation, PhotoMesh determines precisely which tiles and which specific AT sub-processes were affected by the modification of properties or control points, enabling a much lighter and faster AT process. This iterative process should be repeated until satisfactory AT results are achieved.

The diagram below outlines the recommended aerotriangulation workflow.
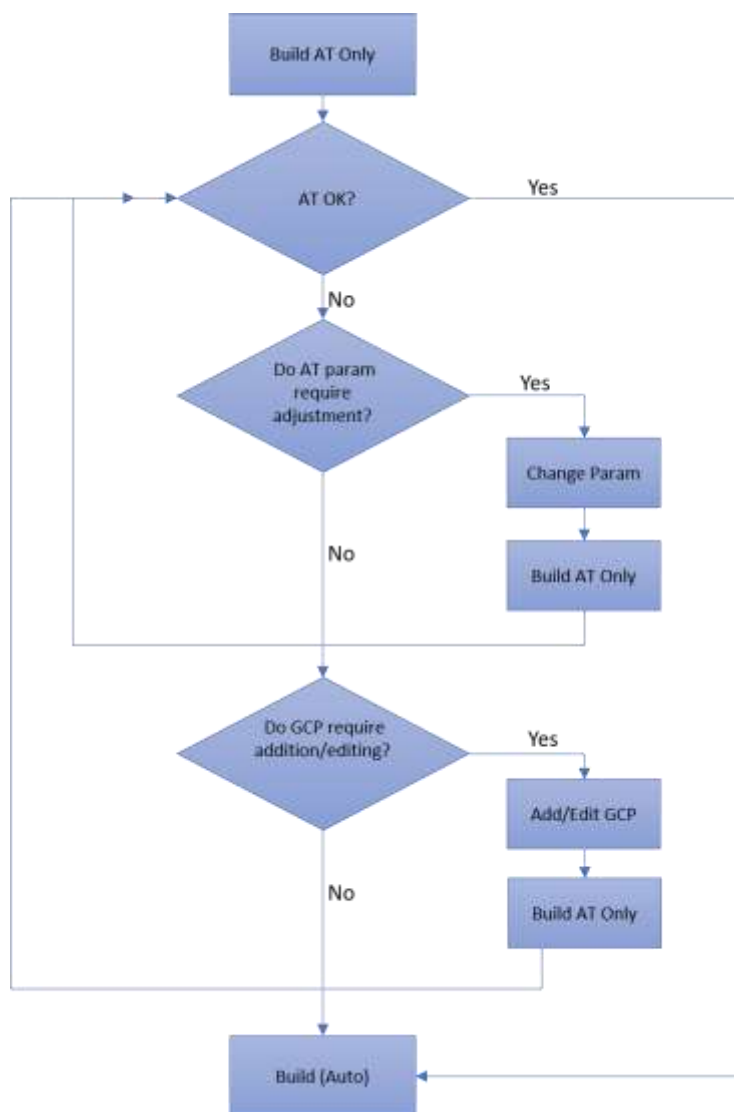
Defining the Aerotriangulation (AT) Area

Before starting a build, you should define the aerotriangulation (AT) area.

To define the aerotriangulation area:
1. If you want to perform aerotriangulation on the entire project, on the **Home** tab, in the **Area** group, click the arrow under **AT Area** and select **Automatic.**
2. If you want to perform aerotriangulation on a subset of the project define the AT area (**See** "Setting the AT Area" in the "Preparing the Project" chapter for more information).
   Note: It is generally recommended to perform aerotriangulation on a preview model of a small subset of the project to validate camera parameters and build settings and check the resulting AT tiles before building the full project.

Defining the Reconstruction Area

By default, the reconstruction area is set to the same as the AT area.

To define the reconstruction area:
1. If you want to perform reconstruction on the same area as aerotriangulation, on the **Home** tab, in the **Area** group, click the arrow under **Reconstruction Area** and select **Same as AT Area.**
2. If you want to perform reconstruction on a different area than the area set for aerotriangulation, set the reconstruction area (**See** "Setting the Reconstruction Area" in the "Preparing the Project" chapter for more information).

Setting Build Parameters and Processing the Build

Before PhotoMesh can build the project, build steps and project parameters are set in the Build Parameters dialog box. The Build Parameters dialog box includes a list of Build Presets that make it easy to automatically set the required parameters to generate the optimum output for the project's data as well as to share user-defined build settings between projects. Presets only include changes from the default settings. **See** "Working with Presets" in this chapter for information.

To set build steps and parameters and start processing:
1. On the **Home** tab, in the **Build** group, click **Build**. The Build Parameters dialog box is displayed.
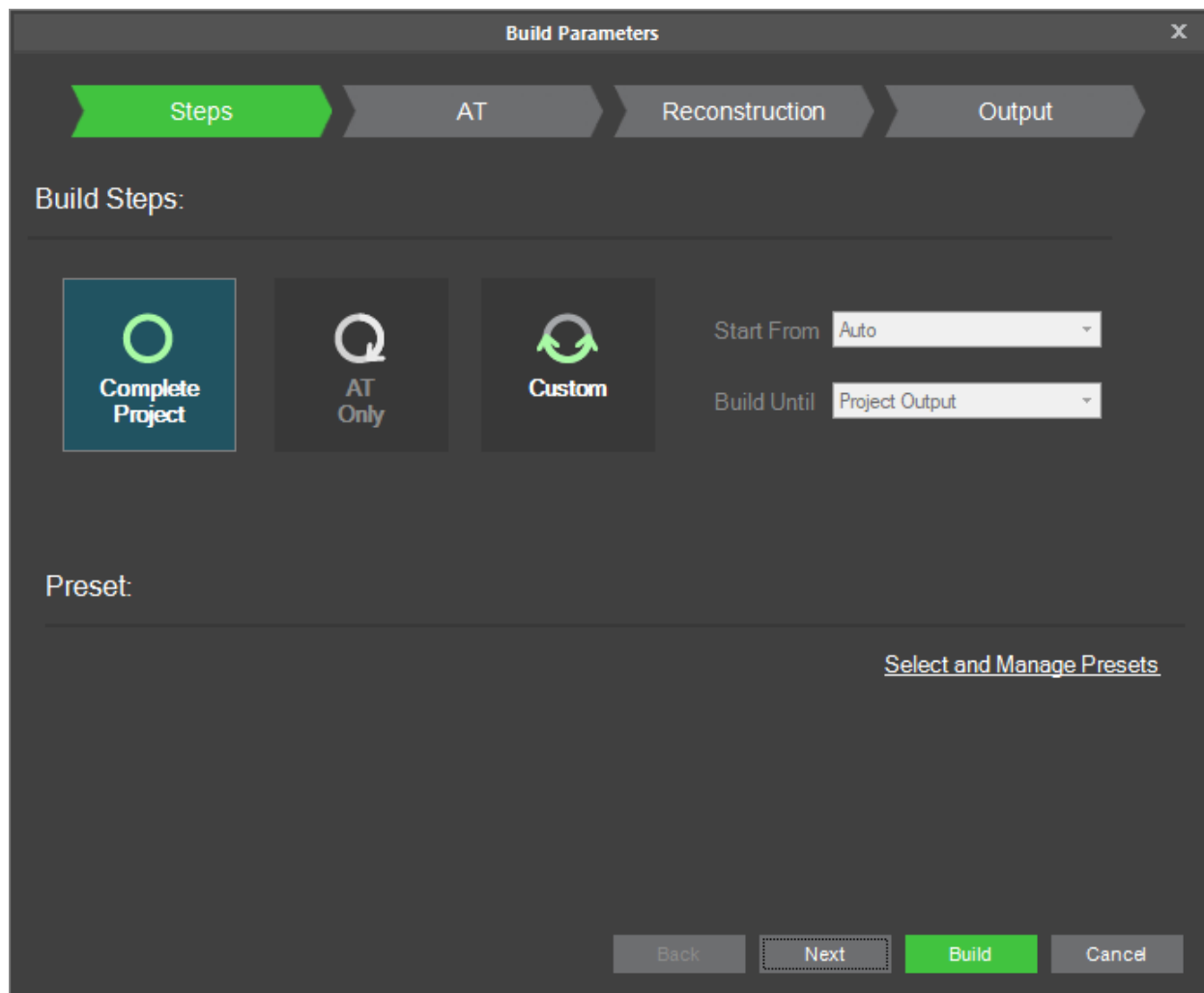
Build Parameters Dialog Box

2. Select the Build Steps to perform:
   - **Complete Project** – Perform all build steps. The Start From field is automatically set to Auto to automatically perform only the steps that are required to complete the project, and Build Until is set to Project Output.
   - **AT Only** – Perform only AT so you can review results before proceeding with the rest of the build to ensure a precise AT which includes all project's photos that intersect with the AT area. When an AT Only build is performed and then repeated following project adjustments, PhotoMesh determines precisely which tiles and which specific AT sub-processes were affected by the modification of properties or control points, enabling a much lighter and faster AT process.
   - **Custom** – Perform selected build steps. To select the steps:
     i. In the **Start From** field, select the step from which you want to start the build. Select "Auto" to have PhotoMesh automatically determine what processing must be performed to complete the project, avoiding unnecessary rebuilding. E.g., when rebuilding a project after modifying a water polygon, only affected tiles are rebuilt from the point cloud step. Similarly, when rebuilding after reimporting exported models, only affected tiles are rebuilt, either from the

texturing step, if just models were imported, or from the output step, if both models and textures were imported.

    ii.   In the **Build Until** field, select the step until which you want to build.

3. If you want to set build parameters automatically by using one of the available presets, click **Select and Manage Presets.** Select the required presets in the Preset Manager that is displayed, and click **OK**. **See** "Selecting Presets" in this chapter for information.

4. If you want to manually set build parameters, click each of the Build Parameters tabs and set the required parameters:

   - Build Steps
   - AT
   - Reconstruction
   - Output Formats

5. Click **Build**. The New build description dialog box is displayed. Enter a build name and click **Ok**. The PhotoMesh Build Manager is displayed.
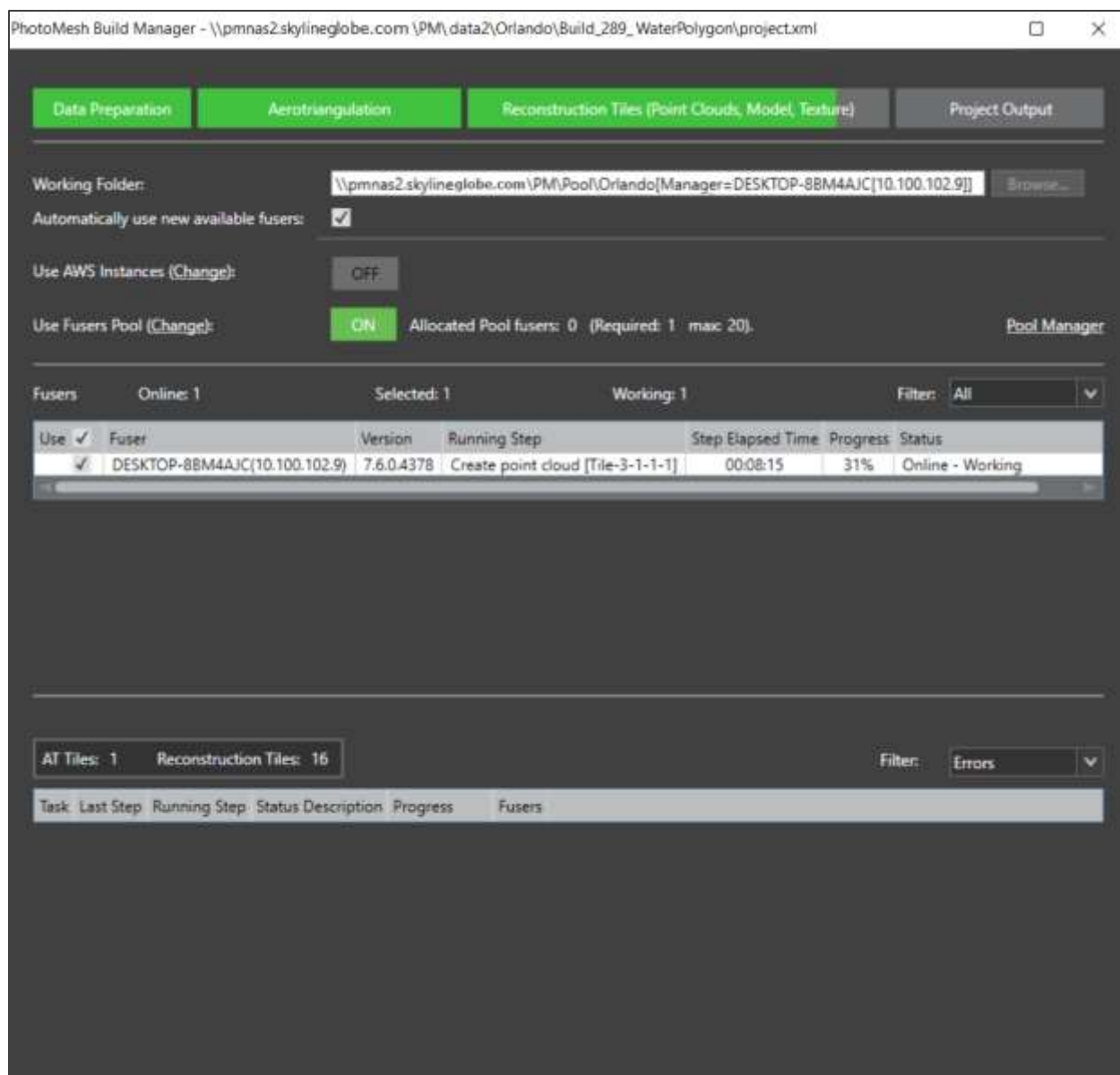
PhotoMesh Build Manager – Starting a Build

6. If you want to change the working folder, in the **Working Folder** field, click **Browse** and in the dialog box that opens browse to the required folder. **See** "Setting the Working Folder (on the Master Computer)" in the "Fusers" chapter for information.

7. If you want PhotoMesh to automatically use any new fusers that connect to the project's working folder, select **Automatically use new available fusers**. See "Using Fusers" in the "Fusers" chapter for information.

   Note: This setting should generally be selected when using AWS and pool fusers.

8. If you want to use AWS fusers, toggle on **Use AWS Instances**. **See** "Using Amazon Web Services (AWS) Fusers" in the "Fusers" chapter for information.

9. If you want to use pool fusers, toggle on **Use Fusers Pool**, and enter the required information. **See** "Using a Fusers Pool" in this chapter for information

10. Select the required fusers. **See** "Fuser Management (on the Master computer)" in the "Fusers" chapter and "Monitoring a Build" in this chapter for more information on fusers.

   Note: If there are currently no available fusers, a dialog box is displayed asking if you want to start a local fuser. Click **Yes** to start a local fuser. Click **No** to start the build without adding a fuser; processing will begin automatically when fusers are added. If you want to cancel the build, click **Cancel**.

11. Click **Build**.

Setting Build Steps

To set build steps:

1. Click the **Steps** tab.
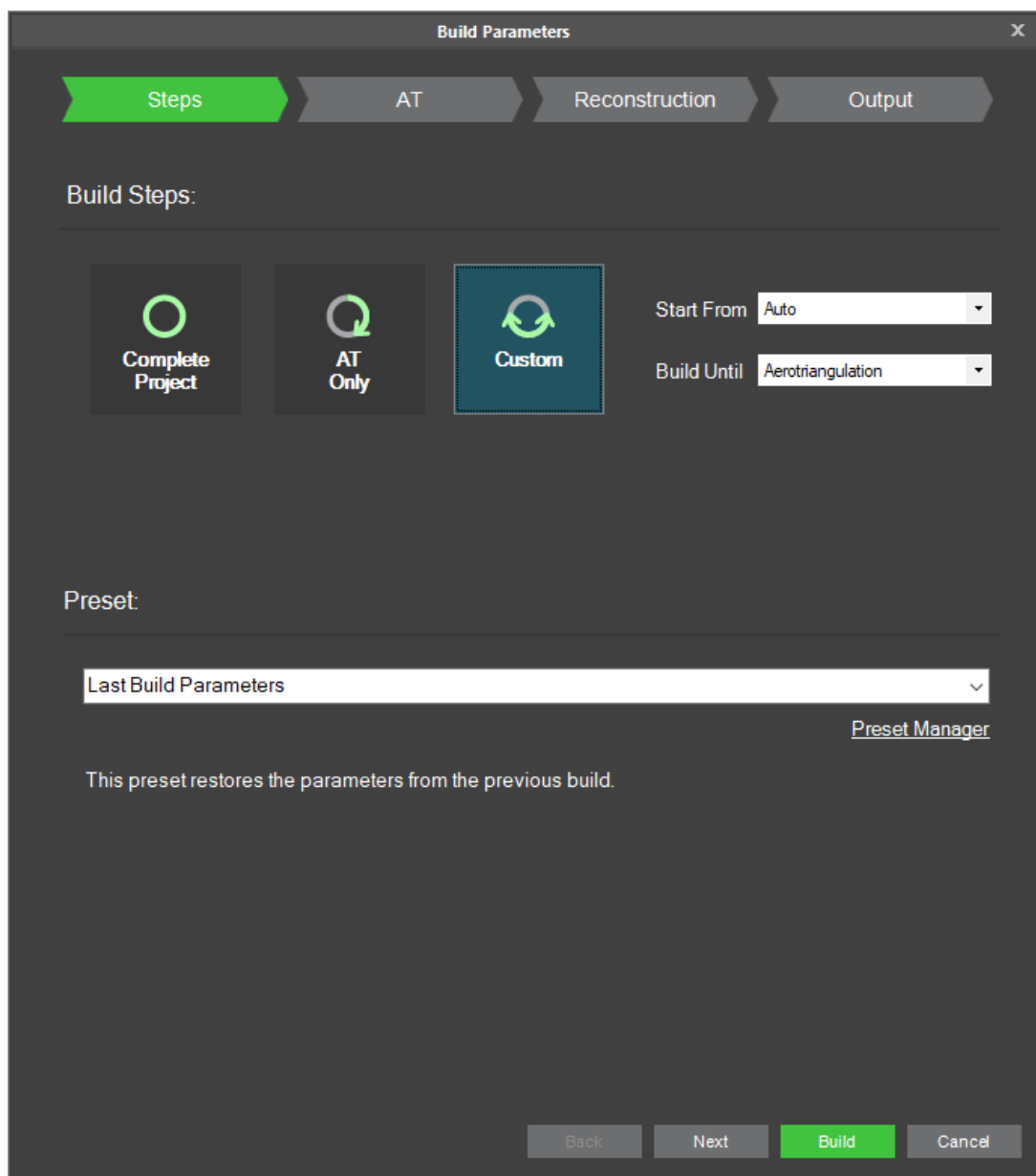
Build Parameters – Build Steps

2. Select the Build Steps to perform:

- **Complete Project** – Perform all build steps to generate project output.
- **AT Only** – Perform only AT so you can review results before proceeding with the rest of the build to ensure a precise AT which includes all photos in the project that intersect with the AT area.
- **Custom** – Perform selected build steps. To select the steps:

    i. In the **Start From** field, select the step from which you want to start the build. Select "Auto" to have PhotoMesh automatically determine what processing must be performed to complete the project, avoiding unnecessary rebuilding. E.g., when rebuilding a project after modifying a water polygon, only affected tiles are rebuilt from the point cloud step. Similarly, when rebuilding after reimporting exported models, only affected tiles are rebuilt, either from the

texturing step, if just models were imported, or from the output step, if both models and textures were imported.

    ii.   In the **Build Until** field, select the step until which you want to build.

3. Click **Next** to set AT parameters (**see** "Setting AT Parameters" in this chapter), or click **Build** to open PhotoMesh Build Manager to start processing (**see** "Monitoring a Build" in this chapter).

Setting AT Parameters

To set AT parameters:

1. Click the **AT** tab.

Build Parameters – AT Parameters

2. Set the parameters on the AT tab:

Parameter	Description
Project	
Coordinate System	Project's internal coordinate system, in which the aerotriangulation and reconstruction are performed. By default,

this is set to the coordinate system of the input data. If the inputted data uses multiple coordinate systems, or you want to use a different coordinate system from your input's, select the required coordinate system.

Select the **Custom** option to create a new custom coordinate system. **See** "Setting Output Formats (Coordinate System)" in this chapter and "Coordinate System" in the "Basic Concepts" chapter for more information.

Note: Click **View** to view the full Well-Known Text (WKT) for the coordinate system.

Collections	
Camera Position Accuracy	Accuracy of the camera positioning data: ▪ Fully trusted input (AT Result)) – PhotoMesh uses inputted camera positioning and orientation information and camera parameters (including focal length, principal point and radial/tangential distortion) without performing aerotriangulation. This option is recommended when reliable AT data in a **Cartesian** coordinate system, such as ENU or ECEF, is added to PM from other AT systems, e.g, Inpho and Bingo. **Note:** AT data in a **projected** coordinate system such as UTM may be unreliable either due to earth curvature or earth curvature corrections performed in the other system. This is particularly true for datasets that contain oblique photos or large frames, or cover a large geographic area. For data of this type, aerotriangulation should be re-performed in PhotoMesh, i.e., use the **Reliable position and orientation** option. ▪ Reliable position and orientation (DGPS and INS) – PhotoMesh assigns a strong weight to inputted camera positioning. If the inputted camera positions are inaccurate or inconsistent with any provided control points, it may be advisable to use the Standard GPS option. This option is recommended for collections captured by professional systems with accurate positioning and orientation information (e.g. using DGPS and INS). ▪ Standard GPS – PhotoMesh performs a robust calculation of positioning and orientation information and camera parameters. This option is recommended for collections using standard GPS with or without orientation information (e.g. drones). ▪ Unreliable or unavailable – PhotoMesh performs a robust calculation of positioning and orientation information and camera parameters. All AT tiles are then aligned to each other before they are aligned as a single block to the provided positions. This option is recommended when the difference between the input position and the actual position may be greater than 50m. In most cases this is caused by a lag between the acquisition of the positioning

information and the photo capture time, or when multiple collections are taken at different times with inaccurate positioning hardware.

AT Tiles

Target Photos per AT Tile

Number of photos to be included in each AT tile. The default setting of 5,000 photos is recommended for most projects and can be safely run on 16 GB computers. In projects where it is necessary to have AT tiles with more than 5,000 photos, the recommended RAM is as follows. Results may vary depending on the type of collection and settings:

Maximum Photos / AT Tile	Recommended Fuser RAM
7.5K	16 GB
15K	32 GB
30K	64 GB
50K	128 GB

Certain settings or AT-related presets may require more RAM than recommended above. E.g., it is recommended to set the Target Photos per AT Tile to 30% less than the value recommended above in both of the following scenarios:

- When using the aggressive match preset

- When photo overlap is higher than 75%

When running a large AT, it is highly recommended to leave the CPU free of other processes, since the AT process utilizes a large percentage of the available processing power.

PhotoMesh includes photos in an AT tile based on the photos' frustum, if orientation information is available, or based on its position and effective range in cases where no orientation information is provided. If the effective range is large and there is a very high overlap of photos, it may be advisable to set the orientation filter to Auto. This filter will try to accelerate the AT process by filtering out photos after a rough calculation of their orientation before proceeding with the remaining aerotriangulation steps. **See** "Collection Property Sheet" in the "Photo Management" chapter for information about the effective range property. **See** "Setting AT Parameters" in the "Building" chapter for more information about the orientation filter

Larger AT tiles with more photos may be necessary if the default AT tiles have fewer than five ground control points that fall in or close to each tile, as is recommended for good accuracy. **See** the "Control Points" chapter for more information.

	Note: An AT with a large number of photos may take many hours to complete, and if interrupted will need to be rebuilt.
Orientation Filter	Determines whether camera positions and orientations are quickly approximated to filter out photos that do not intersect with the AT tile area. For each AT tile, only photos that intersect with it go through the remaining aerotriangulation process steps. If you select **Auto**, the filter is run only if there are photos without orientation information in the tile.
Match Mode	Select Auto to dynamically adjust the complexity of the feature matching process based on the photos. For difficult datasets, select Rigorous to have PhotoMesh perform an intensive matching process with a higher success rate.
Advanced	
Focal Length	Select whether to calculate or use the original value provided.
Radial Distortion	Select whether to calculate or use the original value provided.
Tangential Distortion	Select whether to calculate or use the original value provided.
	Note: When using non-professional camera equipment, calculation of tangential distortion is recommended.
Principal Point	Select whether to calculate or use the original value provided.
GPS Factor	Weight factor to assign to the photos' GPS data when performing the aerotriangulation. Select **Auto** to base on Camera Position Accuracy where "Unreliable" will assign a factor of 0 and only align the block solution to the GPS at the end of the aerotriangulation, and all other Camera Position Accuracy options will set the GPS factor to 1.
Ground Control Point Factor	Weight factor to assign to the ground control points (0-10000) when performing the aerotriangulation. Select Auto to use the default value of 1000.
Calculate Camera Parameters	Select one of the following options for calculating the camera's intrinsic parameters: ▪ Per collection settings – For each collection, use the collection's Calculate Camera Parameters setting set in its property sheet. **See** "Collection Property Sheet" in the "Photo Management" chapter for more information. ▪ Always per entire collection – For all collections (regardless of the collection's Calculate Camera Parameters setting), ensure that all camera intrinsic parameters are the same for all photos in each collection. ▪ Always per photo – For all collections (regardless of the collection's Calculate Camera Parameters setting), always calculate intrinsic parameters per photo. This option is recommended for photos with variable zoom or video collections.
Multiple AT Groups in Tile	Select whether to include only a tile's largest AT group (photo block) or all AT groups in AT Calculated Photos.

Align Photos to Camera Position	Select one of the following options:
	▪ Per AT Tile – Each AT tile is aligned separately to its camera positions and ground control points. This option is recommended for projects with accurate camera positions as well as for nadir and corridor (linear) projects (e.g., roads or rivers) where the AT tile may come out arched.
	▪ Entire Project – If the tiles have ground control points, each AT tile is aligned separately to its camera positions and GCP's, and then aligned to the other tiles. Otherwise, each AT tile is aligned separately to its camera positions and then the entire project is aligned as a single block to the camera positions. This option is recommended for projects with unreliable camera position accuracy.
Skip Data Preparation	When this option is selected, PhotoMesh skips the data preparation step entirely, saving the time that would otherwise be necessary to verify that it had been performed. Only select this option if this step was previously performed, and there has been no change to the AT area or to the project.
AT Flags	Flags that specify alternate behaviors and settings that PhotoMesh should apply. These flags can be obtained from Skyline support.

3. Click **Next** to set reconstruction parameters (**see** "Setting Reconstruction Parameters" in this chapter), or click **Build** to open PhotoMesh Build Manager to start processing (**see** "Monitoring a Build" in this chapter).

Setting Reconstruction Parameters

To set reconstruction parameters

1. Click the **Reconstruction** tab.

Build Parameters – Reconstruction Parameters

Parameter	Description
Tiles	
Reconstruction Tile Size	Size of the reconstruction tiles. This can be set to Auto or to a specified size in meters. It is generally recommended to use

the Auto option. When tile division results were non-optimal, setting a size in meters is recommended.

If you want to control tile size based on size in meters, three factors should be taken into account: RAM, resolution and overlap. For projects with standard resolution (5-10 cm photos) 100-150 meters is recommended on computers with 16 GB of RAM. Larger reconstruction tiles require more memory. On computers with over 64 GB of RAM, a tile size up to 400 meters can used be used. As a general rule, the reconstruction tile size should be the ground resolution of the data multiplied by 2000. For high overlap datasets, the ratio between the reconstruction tile size and the ground resolution should be lower.

Note: If point cloud build steps are repeatedly failing, and you observe a very high RAM usage towards the end of the point cloud step, it is recommended to reduce memory requirements or reconstruction tile size.

Point Cloud	
Point Cloud Quality	The resolution of the photos at which 3D points are extracted.
Min. Number of Matches	The minimum number of matches required to generate each 3D point. When the point cloud generated is sparse with little overlap of photos, consider reducing the minimum number of matches. When there is noise in the point cloud output, it is recommended to increase the minimum number of matches, providing there is high overlap of the photos. Select Auto to dynamically calculate, for each 3D point in the point cloud, the minimum number of matches required to generate the point to produce an optimal point cloud from the available data.
Correlation Window Size	Size in pixels of the window used to perform the correlation between photos and extraction of 3D points from them. Select Auto to automatically calculate optimal window size for performing the correlation.
Merge Lidar Points	The data to use in creating the point cloud: only Lidar points, only points generated from photos, or Auto – supplement Lidar with points generated from photos where Lidar information is insufficient.
Lidar Collection Mode	The direction of Lidar collection assists in calculating surfaces from the point cloud. When there is no trajectory file and the Lidar collection is known to be airborne, select **Aerial**. Otherwise, select **Auto**.
Point Cloud Collections	Click the **More Options** ⸬ button to select the collections to include in the point cloud. **See** "Setting the Collections and Bands to Include in the Build Outputs" in this chapter for information.
Model	
Mesh Resolution	Level of mesh complexity. The Low option will aggressively simplify the mesh.
Smooth Surfaces	Determines whether surfaces are flattened.

Enhanced Edges	Determines whether 2D edges in the project's photos are correlated and 3D edges extracted from them, to produce the 3D model. This setting produces better-defined 3D models with sharper edges, which is important for urban area reconstructions and thin structures such as poles and wires.
Remove Floating Components	Level of removal of small floating components in the mesh model (e.g. noise, electric lines, lamp poles).
Tile Vertical Clipping Buffer	Vertical buffer to apply to the tile. Adding a buffer with a positive value is useful when there are objects that were cut off in a specific tile. A buffer with a negative value can be used to crop areas with small floating components.
Texture	
Color Saturation	Vividness of the colors of the textured model.
Texture Blending	Determines whether to use data blended from multiple photos in texturing the model, or only data from the optimal photo. Select **No** if blending results in ghosting artifacts. Select **Extra** if Yes produces color patches in areas with strong color variance.
Sharpening	Level of image sharpening applied to the final result.
Texture Bands and Collections	Click the **More Options** ⋯ button to select the bands to assign to each RGB of the mesh model texture, and the collections to include in generating the texture. **See** "Setting the Collections and Bands to Include in the Build Outputs" in this chapter for information.
Use Lidar Colors	The data to use in texturing the model: only color from Lidar points, only texturing from photos, or Auto – supplement photo textures with color from Lidar points where photo texturing information is insufficient.
Advanced	
Reconstruction Flags	Flags that specify alternate behaviors and settings that PhotoMesh should apply. These flags can be obtained from Skyline support.

2. Click **Next** to set output formats (**see** "Setting Output Formats" in this chapter), or click **Build** to open PhotoMesh Build Manager to start processing (**see** "Monitoring a Build" in this chapter).

Setting Output Formats

The available output formats vary depending on your license level. Contact Skyline Support for more information.

To set output formats

1. Click the **Output** tab.

Build Parameters – Output Formats

Parameter	Description
Coordinate System	Coordinate system of the output: select the required coordinate system, or click **View** to see the full WKT. From the **Coordinate System** drop-down list, select **Custom** to define a custom coordinate system. **See** "Coordinate System" in the "Basic Concepts" chapter for more information. **Note**: 3D Tiles output is always produced in ECEF coordinate system and I3S/SLPK output in WGS84 coordinate system with EGM96 vertical datum, regardless of the coordinate system selected here. **Note**: All outputs are saved to the output folder, with a WKT to identify the output coordinate system that was selected in this parameter and special WKT's for the coordinate systems of the 3D Tiles and I3S/SLPK output (Output-WKT_Cesium.txt and WKT_I3S.txt).
Output Folder	Output folder in which the build results are saved (read-only).

3D Mesh	Select all required 3D Mesh output formats:
	
	▪ **3DML** – Stream-optimized, high-resolution 3D Mesh Layer database for loading directly into TerraExplorer and streaming with SkylineGlobe Server.
	
	▪ **3D Tiles** – Multi-resolution 3D tile textured model for streaming to Cesium clients. If you want to reproject the elevation values to ellipsoid (as required by the Cesium 3D Tiles standard) for compatibility with terrain that uses ellipsoid elevation, select **Reproject elevation**. This option should only be selected for terrain providers that use TRUE ellipsoid elevation.
	
	    Note: Some applications, e.g., Cesium-based TE4W, use a TerrainProvider and other data with geoid elevation values even though they are declared as ellipsoid when delivered to TE4W/Cesium. Thus, they are actually compatible without reprojection.
	
	    Note: PhotoMesh outputs can easily be published to TE4W as well as other Cesium viewers by publishing the resulting 3DML to SGS. SGS will then make them available to TerraExplorer clients as 3DML as well as to TE4W/Cesium by converting them on-the-fly to 3D Tiles. Use the 3D tiles output if you need to produce a full 3D Tiles output for a non-SGS web-server.
	
	▪ **DAE** – Textured Collada model per reconstruction tile.
	
	▪ **OBJ** – Textured OBJ model per reconstruction tile.
	
	▪ **OSGB** – Multi resolution textured model for use in OpenSceneGraph.
	
	    Note: The file pyramid \p-00000.osgb is the root of the hierarchy that can be used to load all data.

	Note: If you want to reference the DAE/OBJ/OSGB model's coordinates to the project's center, select **Center pivot to project**. This setting improves the stability of 3D model display in 3rd party viewers that are limited regarding display of large coordinates. When selected, only DAE, OBJ, OSGB can be produced. If you want to set a custom pivot, click **More Options** . **See** "Setting a DAE/OBJ/OSGB Model Pivot" in this chapter for information. If you need the project center's coordinate information to reference the model in external software, this information can be extracted from the Output-CenterPivotOrigin JSON file created in the build's output folder when the output is generated. The **Origin** and **WKT** name/value pairs in this file contain the coordinates of the project's center to which the model is referenced and their coordinate system WKT.
	▪ **I3S/SLPK** – ESRI's Indexed 3D Scene (I3S) Layers delivery format used to stream very large 3D geospatial datasets to mobile, web and desktop clients. When this output format is generated, a tool is also produced in the output folder that can convert the I3S output to a compressed SLPK package.
	Note: To convert the I3S to SLPK, run the "SPKMaker.exe" that is placed in the output folder, and browse to the "I3SOut.slpk" file created in that folder. The I3S layer is converted to SLPK v1.5.
	Note: For I3S/SLPK v1.7, it is recommended to output the mesh model as 3DML, and then export the mesh to I3S in TerraExplorer Pro. **See** "Exporting 3D Mesh Layers to Other Formats" in the TerraExplorer User Manual. Alternatively, you can output to I3S v1.5 in PhotoMesh and then convert the output to SLPK v1.7 using the I3S converter.
Point Cloud	Select the **LAS** check box to create a Lidar LAS file with X,Y,Z position and color information for each point of the point cloud. Then select one of the following options: ▪ **Sampled refined 3D model** – To create a point cloud by sampling the final textured model. Then in the **Resolution** field, enter the density for sampling the model. ▪ **Points from photo correlation** - To generate a point cloud based on the correlation of points in the photos.
Raster	Select all required raster formats: ▪ **Orthophoto –** A true-orthophoto that is created from the resulting textured mesh in GeoTiff format.

- **Resolution –** The raster resolution in meters per pixel. Select **Auto** to have PhotoMesh automatically select the optimal resolution based on the inputted data.
- **Image Quality** – Select the check box to compress the orthophoto output, and then enter a JPEG compression level (0-100%). Select 99% to apply an LZW lossless compression.
- **Tile Size** – Set the maximum tile size in pixels. For orthophotos with RGB bands being processed on fuser computers with about 16 GB RAM, tile size should not exceed 50,000 pixels.
  The required RAM in bytes for processing can be roughly calculated as follows: Tile size (in pixels) * Number of bands. E.g., generation of an RGB orthophoto with tile size of 50,000 pixels will use about 7.5 GB (50,000*50,000*3= 7,500,000,000 = about 7.2 GB). Generating this orthophoto on fuser computers with about 16 GB RAM, as recommended above, leaves sufficient RAM for other usages.
- **Overlap Between Tiles** – Overlap in pixels between tiles.
- **Texture Blending** – Determines whether to use data blended from multiple photos in texturing the orthophoto, or only data from the optimal photo. Select **Auto** to base this setting on Photo Priority, where "Nadir Direction" will set blending to No, and "Used in Reconstruction" will set blending to Yes. Select **No** if blending results in ghosting artifacts.
- **Photo Priority** – Determines what type of photo is preferred when selecting the optimal photo for texturing the orthophoto. Select "Nadir Direction" to give priority to photos photographed from above. This option is recommended to avoid objects such as trees being textured from multiple photos photographed from different directions. Select "Used in Reconstruction" to give priority to the photo used to reconstruct the mesh for this area. This option is recommended when there are moving objects in the photos, to avoid having parts of objects appear in different areas of the orthophoto.
- **No-Data Color** – Determines the color to assign to areas of the orthophoto for which there is no data or which are outside the reconstruction area.
- **Bands and Collections** – Click the **More Options** button to select the bands to include in the orthophoto, and the collections to include in generating the orthophoto. **See** "Setting the Collections and Bands to Include in the Build Outputs" in this chapter for information.
- **Tile Grid Origin** – Set the X and Y coordinates of the origin of the tile grid in the output coordinate system. This allows you to set a common grid for multiple projects. Select **Auto** to have PhotoMesh

197

automatically select the optimal grid origin based on the inputted data.

- **DSM** (Digital Surface Model) – An elevation model in GeoTiff format that includes all elevation data, including buildings and reconstructed objects in the mesh.

  - **Resolution –** The raster resolution in meters per pixel. Select **Auto** to have PhotoMesh automatically select the optimal resolution based on the inputted data.

  - **Tile Size** – Set the maximum tile size in pixels. For DSM being processed on fuser computers with about 16 GB RAM, tile size should not exceed 45,000 pixels. The required RAM in bytes for processing can be roughly calculated as follows: Tile size (in pixels) * Number of bands. E.g., generation of a DSM with tile size of 45,000 pixels will use about 7.7 GB (45,000*45,000*4= 8,100,000,000 = about 7.7 GB). Generating this DSM on fuser computers with about 16 GB RAM, as recommended above, leaves sufficient RAM for other usages.

  - **Overlap Between Tiles** – Overlap in pixels between tiles.

  - **Tile Grid Origin** – Set the X and Y coordinates of the origin of the tile grid in the output coordinate system. This allows you to set a common grid for several projects. Select **Auto** to have PhotoMesh automatically select the optimal grid origin based on the inputted data.

- **DTM/DEM** (Digital Terrain Model /Digital Elevation Model) – An elevation model that represents the bare ground surface without any objects such as plants and buildings.

  - **Resolution –** The raster resolution in meters per pixel. Select **Auto** to have PhotoMesh automatically select the optimal resolution based on the inputted data.

2. To save your building parameters as a Build Preset, click **Save as Preset** (**see** "Creating a New Preset" in this chapter) or click **Build** to begin the build (**see** "Setting Build Parameters and Processing the Build" in this chapter).

Selecting the Collections and Bands to Include in the Build Outputs

You can select the collections to include when generating the mesh model texture, orthophoto, or point cloud, as well as the bands to include in the orthophoto and to assign to each RGB of the mesh model texture. **See** "Setting Reconstruction Parameters" and "Setting Output Formats" in the "Building" chapter for more information.

To select the collections:

1. On the **Collections** tab, select the collections that you want to include for the particular output.

2. Click **OK**.

To select the bands:

1. On the **Bands** tab, from the **Bands Available** section, select a required band, and click **Insert** [>] to insert it in the **Bands Used** section. Insert the bands in the required order. If you want to remove a band from the Bands Used section, select the band, and click **Remove** [<] .

   Note: When selecting bands for mesh texture generation, three bands are required.
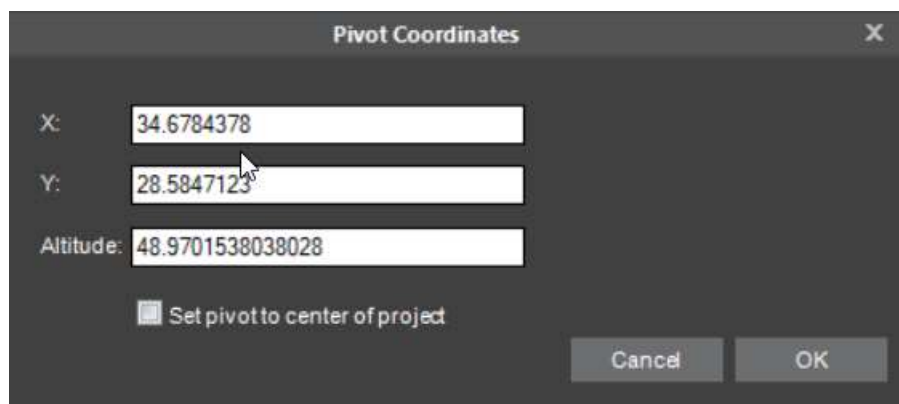
2. Click **OK**.
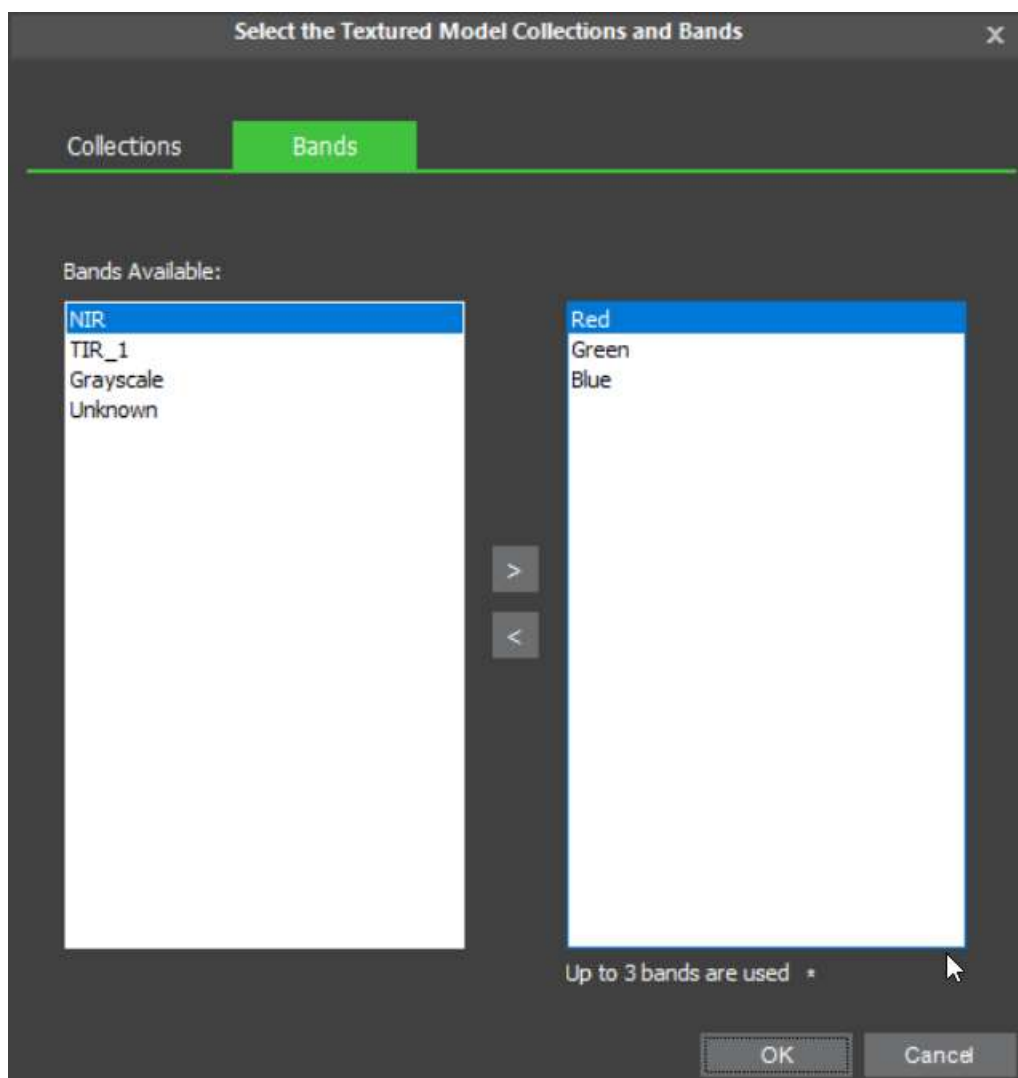
Selecting the Textured Model Collections and Bands

Setting a DAE/OBJ/OSGB Model Pivot

Setting the pivot for a DAE/OBJ/OSGB model improves the stability of its display in 3rd party viewers that are limited regarding display of large coordinates. A DAE/OBJ/OSGB model's pivot can be set to the project's center or set to custom coordinates.

To set a DAE/OBJ/OSGB model's pivot:

1. In the **Build Parameters** dialog box, on the **Output** tab, in the **3D Mesh** section, select **Center Pivot to Project**.

2. If you want to set a custom pivot, click **More Options** [....]. The Pivot Coordinates dialog box is displayed.

Pivot Coordinates Dialog Box

3. If you want to set the pivot to the center of the project, select the check box.

4. If you want to set a custom pivot, enter its coordinates.

5. Click **OK**.

Applying Gigapixel Credits

PhotoMesh Gigapixel (GX) is a licensing mechanism that allows you to apply gigapixel credits towards the building of your PhotoMesh projects based on the number of gigapixels in each project. Within a single project, multiple build versions in all output types can be created using the same gigapixel credits.

This mechanism can be implemented by pre-purchasing a bank of gigapixel credits, loading them to your local Skyline Floating License Server (SFLS), and then applying them to project photos when building a project. You can also send a request to Skyline, using an online form or request file, for gigapixel credits to apply to photos in a project.

If you have a PhotoMesh GX license (see "PhotoMesh License Options" in the Getting Started chapter for information), PhotoMesh automatically launches the Apply Gigapixel Credit (GPC) dialog box when starting a build process. From this dialog, you can apply the required gigapixel credits to the project's photos, as well as access the License Server (SFLS) to load additional gigapixel credits, if required. PhotoMesh calculates the number of gigapixels in the project based on the number of gigapixels in the selected AT area. When rebuilding a project that gigapixel credits were already applied to, additional credits are only required for photos that were added or if the AT area was expanded to include additional photos.

If you don't want to apply gigapixel credits to a build, you can instead generate a build with a watermark, in only 3DML format. You can later apply gigapixel credits and rebuild without a watermark in any output format.

200

Apply Gigapixel Credit Dialog Box

To apply gigapixel credits:

1. Review the following settings in the Apply Gigapixel Credit (GPC) dialog box:

Setting	Description
Project Name	Name of project.
Number of Photos in AT	Number of photos in the AT area.
Total Gigapixels in AT	Total number of gigapixels in the photos that intersect with the AT area.
GP Credits Already Applied to Project	Number of gigapixel credits already applied to photos in this project. Additional credits are only required if photos were added to the project or if the AT area was expanded to include additional photos.
GP Credits Required	Number of gigapixel credits required to build the project. If gigapixel credits were already applied to photos in this project, additional credits are only required for photos that were added or if the AT area was expanded to include additional photos.

2. If you want to apply gigapixel credits from the available credits on your Skyline Floating License Server (SFLS), to photos in this project, select **From Server (SFLS)**. Then enter the following:

License Server (SFLS)	URL to your Skyline Floating License Server.
Available GP Credits on Server	Number of gigapixel credits in your server (SFLS) balance, that could be applied towards building this project.

3. If you want to send a request to Skyline, using an online form or request file, for gigapixel credits to apply to photos in this project, select **Send Request**. Your request includes a key that includes information about the total number of gigapixels in your AT. The following additional fields are displayed:

Company Name	Name of company.
Contact Person	Name of contact person in company.
Email Address	Email address.
Phone Number	Phone number.
Country	Location of company.
GP Credits Request Key	A key that includes information about the total number of gigapixels in your AT, which is used to request gigapixel credits from Skyline. This key is automatically passed to Skyline when you use the Save Request File or Fill Online Form. The build key is a unique key generated automatically based on the current build, and does not contain private data, project coordinates, or client or computer information of any sort.

4. If you want to submit your request offline, click **Save Request File**, enter the contact information in the fields that are displayed, and send it to Licensing@SkylineGlobe.com. After you receive your build license code from Skyline, enter the code in the **GP Credits License Key** field.

5. If you want to submit your request online, click **Fill Online Form**, and enter the required contact information in the online form that is displayed. After you receive your build license code from Skyline, enter the code in the **GP Credits License Key** field.

6. Click **Apply GP Credits**. If you don't want to apply gigapixel credits currently, click **Continue to Watermark**. The build is generated with a watermark, and restricted to only 3DML format. You can later apply gigapixel credits and rebuild without a watermark in any output format.

Working with Presets

Build parameter presets make it easy to automatically set the required parameters to generate the optimum output for the project's data as well as to share user-defined build settings between projects. New presets can easily be created by saving your build settings to a preset file. You have the flexibility to save all settings to the preset, or only selected ones. Presets only include changes from the default settings. This allows multiple presets to be selected for a build, each of which defines settings for a different subset of the build steps. If selected preset files have conflicting properties or flags, the value from the last file on the list is applied. Presets are managed (e.g., loaded, deleted, edited, and set as the default) from the Preset Manager.

Selecting Presets

To select presets for your build:

1. On the **Home** tab, in the **Build** group, click **Build**. The Build Parameters dialog box is displayed.

2. Click **Select and Manage Presets.** The Preset Manager is displayed.
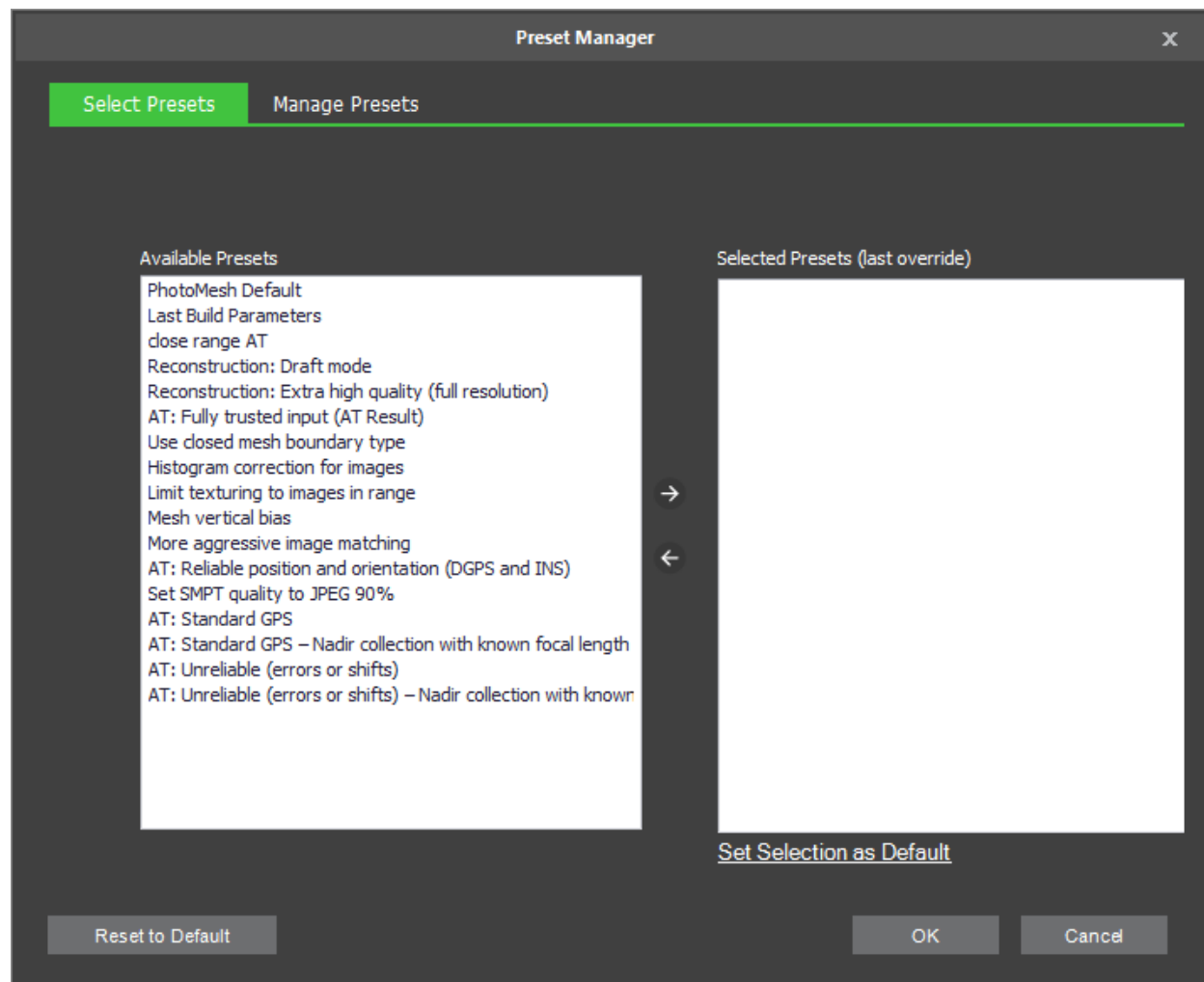
Preset Manager

3. On the **Select Presets** tab, on the left side of the Preset Manager is a list of available presets. Select one or more presets from this list, and click **Add** ➡. If you want to change the selected preset(s), use the **Remove** ⬅ icon to remove a preset from the **Selected Presets** side. If selected preset files have conflicting properties or flags, the value from the last file on the list is applied. The selected Presets side also includes "Custom changes" with all custom changes that were made to AT, reconstruction, and output settings. The selected presets and custom changes are listed in the Preset section of the Build Parameters dialog box.

   Note: The Last Build Parameters preset includes all the Build settings from the last build.

   Note: The PhotoMesh Default preset is designed to provide optimal results for the majority of projects.

4. If you want to set your selections as the default presets for new projects, click **Set Selection as Default**.

5. If you want to clear your selections and restore the last saved default, click **Reset to Default**.

6. Click **OK.**

Loading a Preset from File

To load a preset from file:

1. In the **Steps** tab of the **Build Parameters** dialog box, click **Select and Manage Presets**. The Preset Manager dialog box is displayed.

   Note: The Preset Manager can also be opened from the PhotoMesh Options dialog box. **See** "Setting PhotoMesh Options" in the "Basic Concepts" chapter for more information.
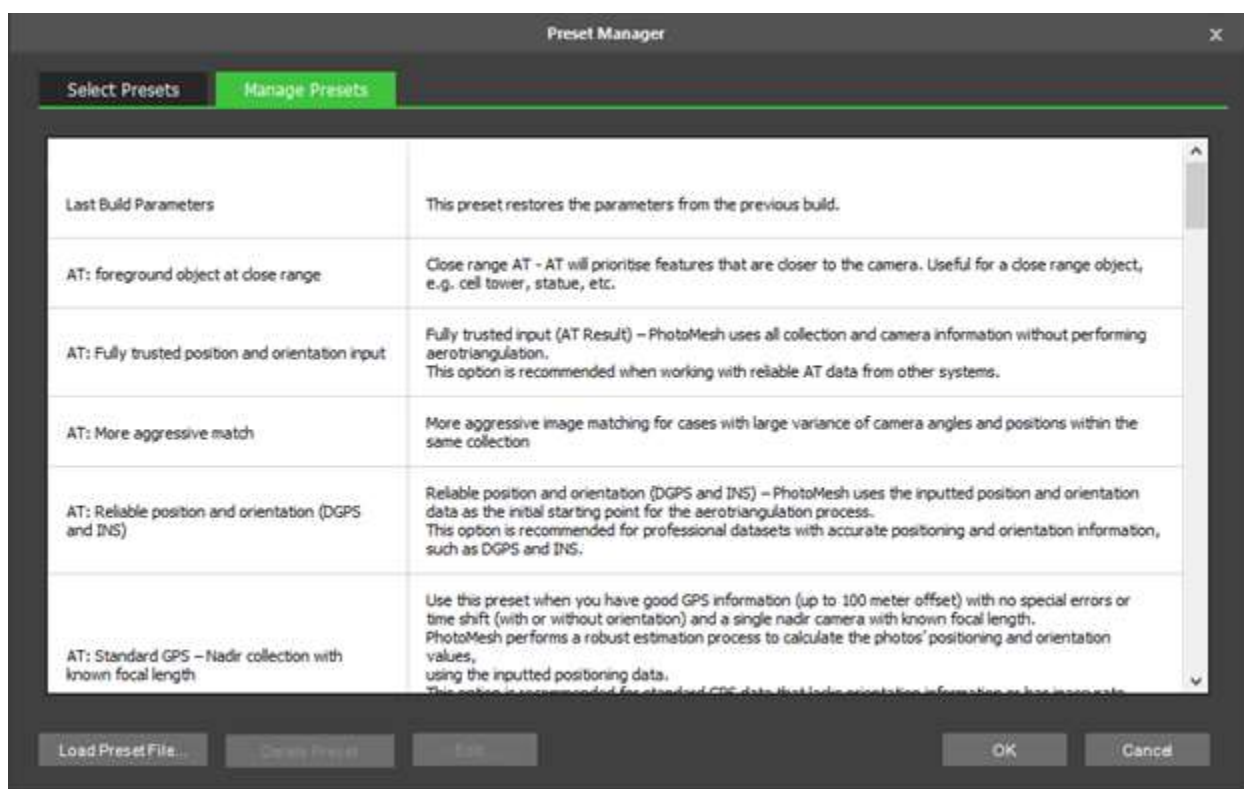
Preset Manager
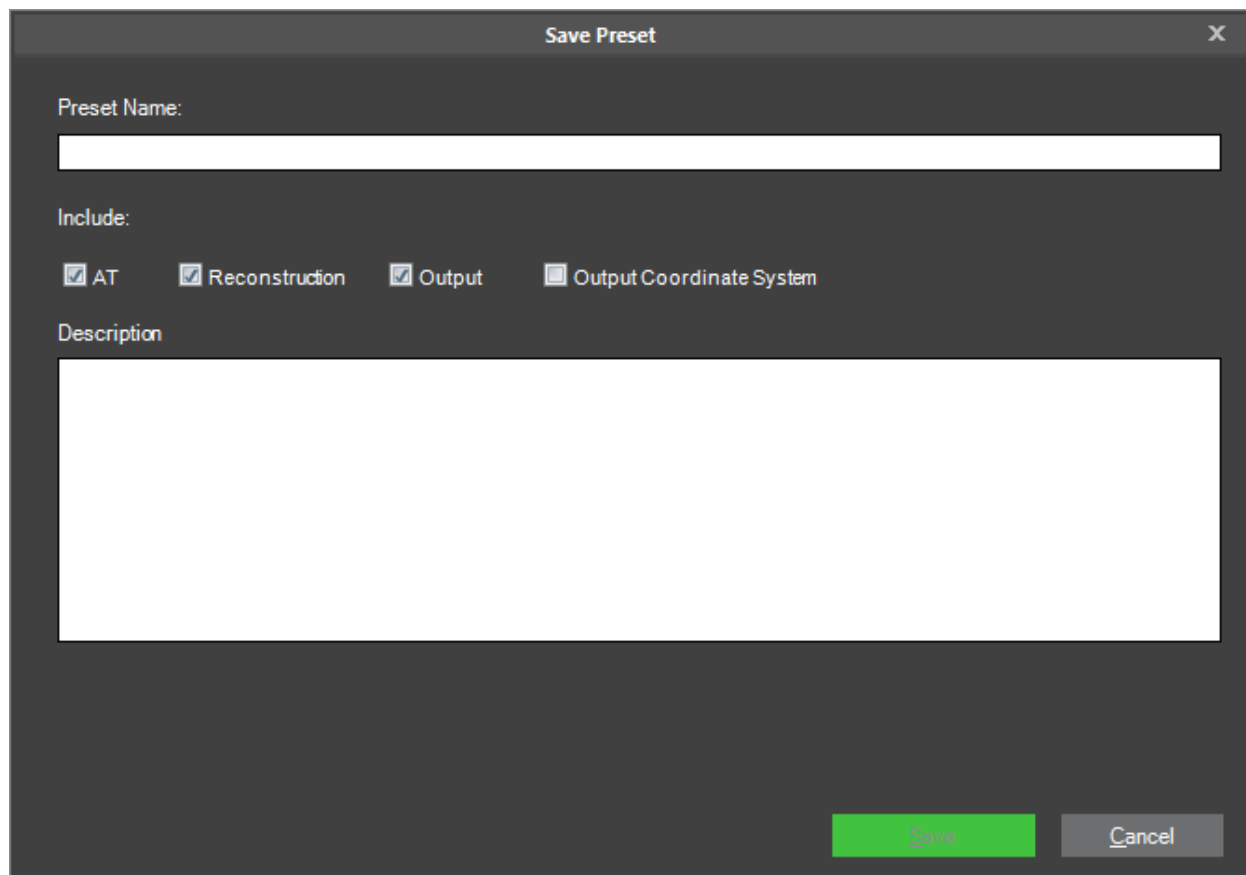
2. On the **Manage Presets** tab, click **Load Preset File**. The Load Preset from File dialog box is displayed.

3. Browse to the required file, and click **Open**.

Creating a New Preset

A new preset can be easily created by saving your build settings to a preset file. You can save all your build settings, or only the settings from specific build steps. Only changes to the default settings are saved.

To create a new preset:

1. Set the build parameters as required. **See "**Setting Build Parameters and Processing the Build**"** in this chapter for information.

2. On the **Output** tab, click **Save as Preset**. The Save Preset dialog box is displayed.

Save Preset Dialog Box

3. Type a **Preset Name** for the preset and a free text **Description**.

4. Select the check boxes of the build steps whose settings you want to save in your preset file.

5. Click **Save**.

Deleting Presets

To delete a preset file:

1. In the **Steps** tab of the **Build Parameters** dialog box, click **Select and Manage Presets**. The Preset Manager dialog box is displayed.

205
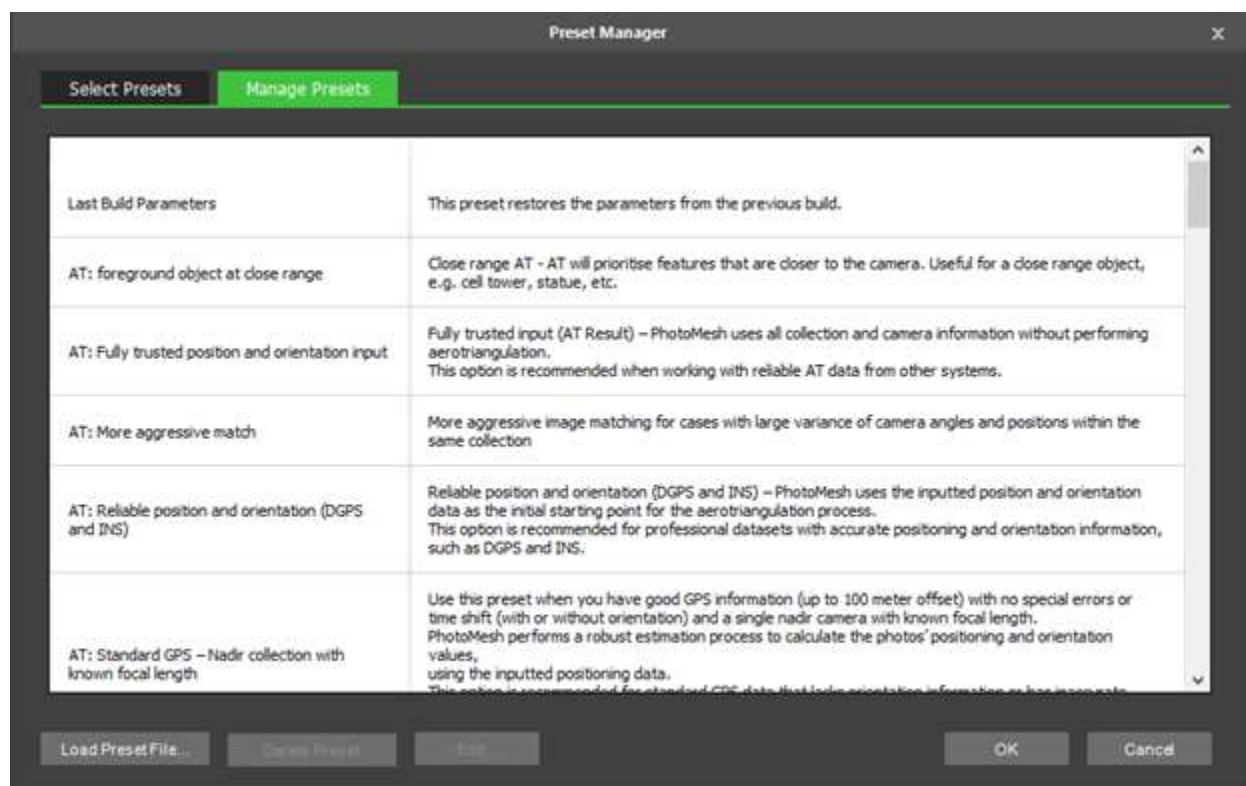
Preset Manager

2. On the **Manage Presets** tab, select the required preset from the list, and click **Delete Preset**.

   Note: Built-in presets cannot be deleted.

3. Click **OK**.

Editing Presets

To edit a preset file:

1. In the **Steps** tab of the **Build Parameters** dialog box, click **Select and Manage Presets**. The Preset Manager dialog box is displayed.
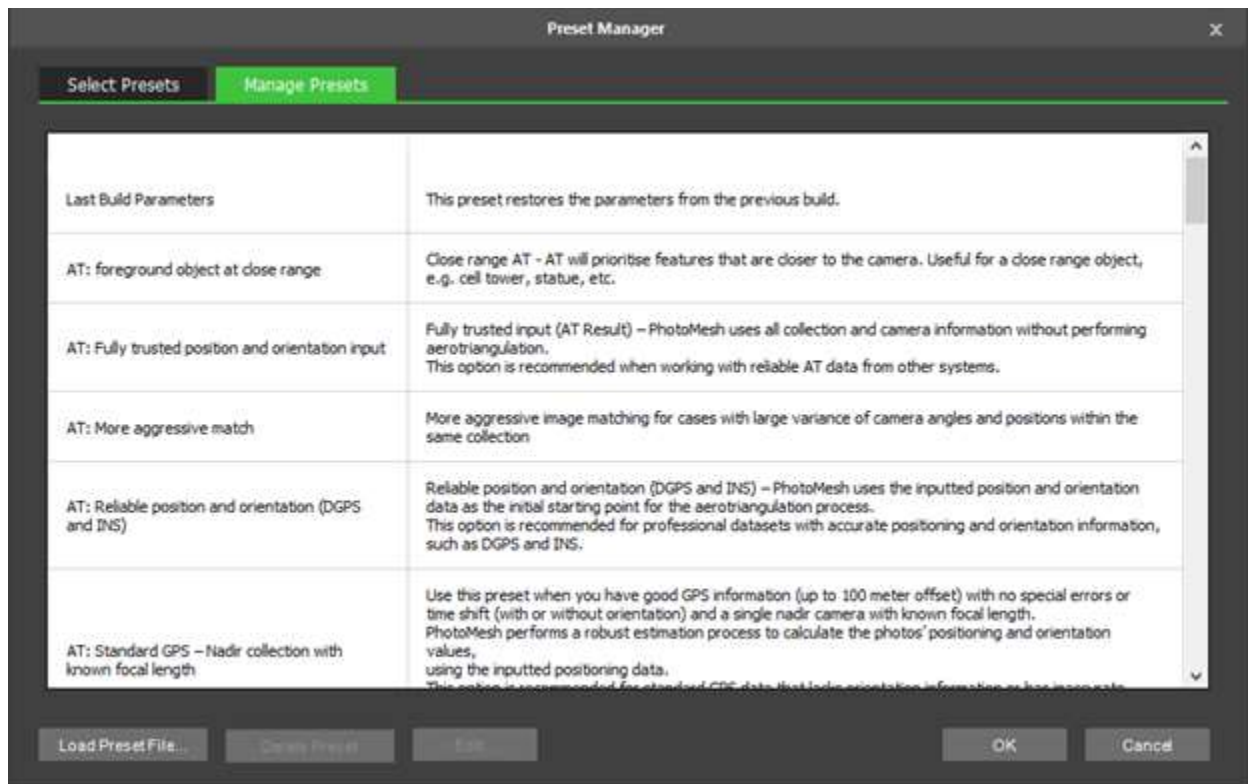
Preset Manager

2. On the **Manage Presets** tab, select the file you want to edit from the list, and click **Edit**. The preset XML file is opened in a text editor

   Note: Built-in presets cannot be edited.

3. Edit the file in the text editor, and save your changes.

4. In the **Preset Manager** dialog box, click **OK**.

Monitoring a Build

PhotoMesh Build Manager has two sections for monitoring the current build:

▪ The upper monitoring section of PhotoMesh Build Manager displays information for each fuser, including status, its currently running step, and its progress for that step.

▪ The lower monitoring section of PhotoMesh Build Manager displays information for each aerotriangulation and reconstruction tile, including currently running step, last step, progress, and currently running fusers.

To monitor the build:

1. In the upper monitoring section of PhotoMesh Build Manager, from the **Filter** drop-down list, select one of the following:

   ▪ **All** – To view all fuser information

   ▪ **Selected** – To view only information related to selected fusers.

2. The following data is available in the upper monitoring section for each of the available fusers:

Field	Description
Fuser	Fuser name.
Running Step	Step that the fuser is currently processing.
Step Elapsed Time	Time that has elapsed since the currently processing step was started.
Progress	Percentage of currently running step that has been completed.
Status	Fuser status: ▪ **Online – Working –** Online and processing ▪ **Online – Waiting –** Online but not currently processing ▪ **Offline – Will be used when online** – Fuser was either disabled or closed.

3. In the lower monitoring section of PhotoMesh Build Manager, from the **Filter** drop-down list, select one of the following:

   ▪ **All** – to view all task information.

   ▪ **Errors** – to view only information related to task errors.

4. The following data is available in the lower monitoring section:

Field	Description
Task	All project tasks including a list of AT and reconstruction tiles.
Last Step	The previous step that was performed
Running Step	Step that the fuser is currently processing.
Status Description	Information about the build status.
Progress	Percentage of currently running step that has been completed.
Fusers	Fuser that is currently processing this task.

PhotoMesh Build Manager – Monitoring a Build

5. If you want to view a build report, click **Build Report**. **See** "Viewing a Tile's Build Report", and "Viewing an Aerotriangulation Tile's Detailed Report" in the "Reviewing Your Build" chapter.

6. If you want to view build settings (read-only), click **Show Settings**.

7. When the build process is complete, the Build Finished dialog box is displayed.
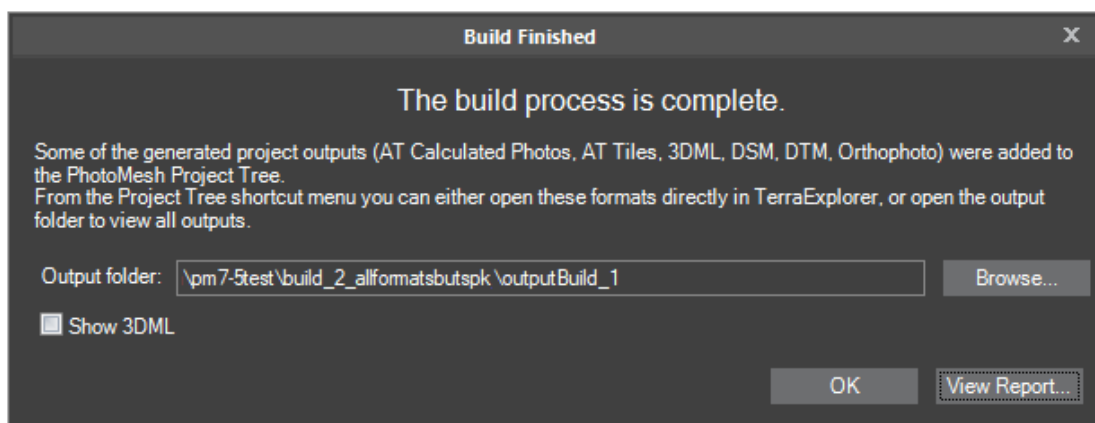
Build Finished Dialog Box

8. If you want to open the output folder, select the link in the **Output Folder** field, and open it in your browser.

   Note: The output folder can also be opened at any time from the Project Tree's shortcut menu. **See** "Outputs Menu" in "Using the Project Tree and Item List" chapter for more information.

9. If a 3DML was created in this build, and you want to show it in the 3D Window, click **Show 3DML**.

   Note: The 3DML's display can also be turned on or off by clicking its entry's show/hide icon in the Project Tree.

10. If you want to view a build report, click **View Report**.

Canceling a Build

To cancel a build:

▪ On the bottom right of **PhotoMesh Build Manager**, click **Cancel**.

Resuming a Build

A PhotoMesh build process can be resumed in any of the following cases:

▪ The initial build process was canceled by the operator in the middle.

▪ Unexpected hardware or software failure occurred during the build causing the build to stop

To resume a build:

Note: You can only resume a build when you are in Review mode for that build.

1. If you are not in Review mode, on the **Home** tab, in the **Review** group, click **Review Mode**.

2. On the **Home** tab, in the **Build** group, click the arrow under **Build** and select Resume **Build**.

   Note: If you want to create an entirely new build, **see** "Creating a New Build" in this chapter for information.

Opening Build Output in TerraExplorer

Many outputs can be opened in TerraExplorer after the build is complete. The supported outputs include: 3DML, AT Calculated Photos, AT Tiles, DSM, DTM, and Orthophoto. These outputs are added to the Project Tree when a build is finished.

To open in TerraExplorer:

▪ Right-click the output folder in the **Project Tree**, and select **Open with TerraExplorer.** A FLY file opens in TerraExplorer with the 3DML, AT Calculated Photos, AT Tiles, DSM, DTM, and Orthophoto outputs, if they were generated in the PhotoMesh project.

> **Note:** The Project Tree's shortcut menu also provides an option to access to any of the project's outputs by opening the location of the project's output folder.

Exporting DSM Output to LAS/LAZ

DSM output can be exported to LAS/LAZ format.

To export DSM output to LAS/LAZ:

1. Right-click the DSM output in the **Project Tree**, and select **Export to LAS/Export to LAZ**. The Select LAS/LAZ dialog is displayed.
2. Browse to the output folder, and click **Select Folder**.

Rebuilding

If a previously built project requires modification, instead of building the entire project again from the first step of the build process, you have the option to run the build again, incorporating some of a previous build's results. Rebuilding provides flexibility both with regard to the project section (entire project, only error tiles or selected tiles) and project steps that are to be performed. Alternative build parameters can also be set. PhotoMesh will automatically determine what processing is required to apply the changes to the project, by rebuilding the project from "Auto". E.g., when rebuilding a project after modifying a water polygon, only affected tiles are rebuilt from the point cloud step. Similarly, when rebuilding after reimporting exported models, only affected tiles are rebuilt, either from the texturing step, if just models were imported, or from the output step, if both models and textures were imported.

> **Note:** If you want to create an entirely new build, **see** "Creating a New Build Version" in this chapter for information.

To rebuild:

1. On the **Home** tab, in the **Build** group, click the arrow next to **Build**, and select the required option:
   ▪ Build
   ▪ Build and Rerun Failed Tasks – Continue the build process including rerunning of any build failed task. Tiles with insufficient photo coverage for reconstruction are not rebuilt. To rebuild these tiles, select the Rebuild Selected Tiles option, and perform all build steps (i.e., select Complete Project).
   ▪ Rebuild Selected Tiles – Rebuild only the tiles that are currently selected.

The Build Parameters dialog box is displayed.

2. Select the project steps and enter the required parameters. **See** "Setting Build Parameters and Processing the Build" in this chapter for information.

> **Note:** It is generally recommended to select Auto for the Start From step. In this case, PhotoMesh will automatically determine what processing must be performed to complete the project.

> **Note:** Some of the parameters will only apply if tiles need to be recreated.

> **Note**: Output fields are not available when rebuilding error tiles.

3. PhotoMesh Build Manager then opens. Select the required fusers, and then click **Build**. **See** "Using Fusers" in the "Fusers" chapter for more information.

Note: The expected production times for a 9 gigapixel project (0.06 km2 and 3 cm GSD) on an i7 10th generation, 8 core CPU, with 16 GB RAM, and a local SSD drive are as follows:

- Approximately 20 minutes for photo preparation (207 photos)
- 2 minutes for a single AT tile
- 7 hours and 30 minutes for the reconstruction tiles

The total machine time adds up to 7 hours and 55 minutes (approximately 52 minutes per gigapixel).

Creating a New Build Version

You can create a new build version with different project steps and different build parameters.

Note: If you want to create a completely new build version with a copy of the AT information from the current build, **see** "Creating a New Build with a Copy of the Current Aerotriangulation Results" in this chapter.

To create a new build version:

1. On the **Home** tab, in the **Build** group, click the arrow next to **New Version**, and select **New Build Version**. The Build Parameters dialog box is displayed.

2. Select the project steps and then enter the required parameters. **See** "Setting Build Parameters and Processing the Build" in this chapter for information. The New build description dialog box is displayed with a consecutive build number, e.g. "Build_2", "Build_3" etc.

3. Type a new build description, and click **OK**. PhotoMesh Build Manager then opens. Select the required fusers, and then click **Build**. **See** "Using Fusers" in the "Fusers" chapter for more information.

Creating a New Build Version with a Copy of the Current Aerotriangulation Results

When no AT parameters were changed from a previous build, you can create a new build version with different build parameters (e.g. reconstruction tile size) but with a copy of the AT information from the current build, thereby significantly accelerating the build process.

Note: If you want to create a completely new build version without copying the AT information from the current build, **see** "Creating a New Build Version" in this chapter.

To create a new build version with a copy of the AT:

1. On the **Home** tab, in the **Build** group, click the arrow next to **New Version**, and select **New Build Version – Copy AT**. The Start From field is automatically set to Auto to automatically select the steps that were not yet completed, and is read-only.

2. In the **Build Until** field, select the step until which you want to build.

3. Click **Next** and then enter the required parameters, and click **OK**. **See** "Setting Build Parameters and Processing the Build" in this chapter for information.

Note: Some of the parameters will only apply if tiles need to be recreated.

Note: Output fields are not available when rebuilding error tiles.

The New build description dialog box is displayed.

4. Type a new build description, and click **OK**. PhotoMesh Build Manager then opens. Select the required fusers, and then click **Build**. **See** "Using Fusers" in the "Fusers" chapter for more information.

CHAPTER 17 REVIEWING YOUR BUILD

About Reviewing Your Build

Upon initiating a build, PhotoMesh automatically switches to Review mode, enabling an array of capabilities relating to the processed data. These include:

▪ Viewing all processed build data in the 3D Window, e.g. AT and reconstruction tiles, calculated camera positions, and point cloud, 3D model, and texture data.

▪ Reading build and AT reports containing statistical and graphical information about the build process.

▪ Exporting/importing tiles for manual retouching.

▪ Rerunning the build (for the entire project or for selected tiles) with different parameters.

As soon as a build step is completed for a tile, its build data becomes immediately available and viewable, e.g. if aerotriangulation was performed, PhotoMesh can display the calculated camera positions in the 3D Window, and if reconstruction tiles were calculated, they can be displayed in the 3D Window, with each tile colorized according to the build step completed.

In addition to these capabilities which are specific to Review mode, you can also perform in Review mode nearly all operations that were available in preparing for the build, e.g. add or edit control points, water polygons, and reconstruction polygons, add terrain objects and layers and perform measurements. The only operations that cannot be performed in Review mode involve changes to the project's photos (addition, exclusion, or deletion) or modifications to the AT area.

Turning on Review Mode

To turn on review mode:

▪ On the **Home** tab, in the **Review** group, click **Review Mode**.

  Note: Review mode is activated automatically after a build was performed.

Clearing Review Data from Terrain

To clear review data:

▪ On the **Home** tab, in the **Review** group, click **Clean Review Data**. This clears any point clouds, models, or textured models displayed on terrain or photo projection on the terrain.

Reviewing Aerotriangulation (AT) and Reconstruction Tiles

Upon completion of aerotriangulation or any of the build steps, the aerotriangulation/reconstruction tiles can be previewed in the 3D Window, with the tiles colorized according to their processing status. Additional tile information (tile camera positions and build data) can be viewed from the Tile tab.

The sections that follow describe the activities that can be performed to evaluate the aerotriangulation/reconstruction tile results:

▪ View aerotriangulation tiles

▪ View reconstruction tiles

▪ Navigate to a tile

- [Display original camera positions for tiles](#)

- [Display calculated camera positions for tiles](#)

- [View an aerotriangulation tile's sparse point cloud](#)

- [Display reconstruction tile results](#)

- [View and set tile status](#)

- [Display user status flags](#)

- [Set a reconstruction tile's selection style](#)

- [View a tile's build report](#)

- [View an aerotriangulation tile's detailed report](#)

- [Export tiles](#)

- [Import tiles](#)

Viewing Aerotriangulation (AT) Tiles

To view aerotriangulation tiles:

Note: This information is also available in the AT Tiles list. **See** "[Selecting Tiles](#)" and "[Viewing Tile Information in the Tile List](#)" in this chapter for information.

1. On the **Home** tab, in the **Tiles** group, click **AT Tiles**. The aerotriangulation tiles are colorized according to their aerotriangulation status:

   - **White** tile – Unprocessed
   - **Purple** tile – Aerotriangulation completed
   - **Yellow** tile – Aerotriangulation merged
   - **Orange** border – Aerotriangulation failed (retrying)
   - **Red** border – Aerotriangulation failed
   - **Blue** border - <80% Success

   Note: In the AT Tiles legend, the colored squares (e.g. ▣) indicate colored tiles, while the colored lines (e.g. ▬) indicate a colored border.

2. If you hover anywhere on the tile, the tile's name and number of photos in the tile are displayed at the top left corner of the 3D Window.

3. In the **3D Window**, click the tile you want to review, or select the tile in the **Tiles List**. The Tile tab is displayed from which you can navigate to the tile, display its original and calculated camera positions and sparse point cloud, and view build reports. **See** "[Navigating to a Tile](#)", "[Displaying Original Camera Positions for Tiles](#)", "[Displaying Calculated Camera Positions for Tiles](#)", "[Viewing an Aerotriangulation Tile's Sparse Point Cloud](#)" and "[Viewing a Tile's Build Report](#)" in this chapter for information.

Viewing Reconstruction Tiles

To view the reconstruction tiles:

Note: This information is also available in the Reconstruction Tiles list. **See** "[Selecting Tiles](#)" and "[Viewing Tile Information in the Tile List](#)" in this chapter for information.

1. On the **Home** tab, in the **Tiles** group, click **Reconstruction Tiles**. The reconstruction tiles are colorized according to the last build processing step completed for the tile.

   ▪ **Yellow** tile – Aerotriangulation is complete.

   ▪ **Pink** tile – All steps through point cloud creation are complete.

   ▪ **Dark Blue** tile – All steps through model creation are complete,

   ▪ **Light Blue** tile – All steps through model texturing are complete.

   ▪ **Green** tile – All steps through project output are complete.

   ▪ **Orange** border – An error occurred in the processing step that follows the last successfully completed step and a retry mechanism will be applied soon (e.g., if a dark blue colored tile has an orange border, this indicates that the 3D model creation step completed successfully but that model texturing failed and will be retried).

   ▪ **Red** border – Tile failed.

   ▪ **Dark Red** border – Insufficient photo coverage for reconstruction.

   Note: In the Reconstruction Tiles legend, the colored squares (e.g. ⬛) indicate colored tiles, while the colored lines (e.g. ▬) indicate a colored border.

Reconstruction Tiles

2. If you hover anywhere on the tile, the tile's name and number of photos in the tile are displayed at the top left corner of the 3D Window.

3. In the **3D Window**, click the tile you want to preview, or select the tile in the **Reconstruction Tiles** list. The Tile tab is displayed from which you can navigate to the tile, display its original and calculated camera positions and reconstruction tile results, view build reports, and export and import the tile's model. **See** "Navigating to a Tile", "Displaying Original Camera Positions for Tiles", "Displaying Calculated Camera Positions for Tiles", "Displaying Reconstruction Tile Results", and "Viewing a Tile's Build Report" in this chapter and the "Importing and Exporting Models" chapter for information.

Selecting Tiles

Tiles can be selected either in the 3D Window or from the AT or Reconstruction Tile Lists to perform various commands on them, e.g., jump to tile, show calculated information, or show AT or reconstruction tile results.

Reconstruction Tiles List

To select tiles:

1. In the **3D Window**, click the tile you want to select, or select the tile(s) in the **Reconstruction Tiles** list. You can multi-select tiles using the Tile List commands. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information.

2. If you want to save your selection, right-click in the **Reconstruction Tiles** list, and select **Save Selection**.

3. To clear your selection, right-click, and select **Clear Selection.**

4. To load a tile's .*select* file with selection information, right-click, and select **Load Selection**. **See** "Tile Menu" in the "Using the Project Tree and Item List" chapter for more information.

Viewing Tile Information in the Tile List

The AT and reconstruction tiles lists provide information about each tile, including: photo count, build processing status, and error status. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter for more information.

To view tile information:

▪ In the **Project Tree**, select the **AT Tiles** or **Reconstruction Tiles** group. The tile list displays all the tiles in the selected group, and the following information:

Property	Description
Selection	Select the check box to select the tile in the 3D Window. You can select multiple tiles to perform actions on multiple tiles.

Name	Name of the tile (Read-only).
Photo Count	Number of photos in the tile (Read -only).
Photo Calculated	Number of photos in AT tile for which PhotoMesh successfully calculated camera position (only AT) (Read -only).
Calculated %	Percentage of photos in AT tile for which position information was calculated: Photos Calculated/Photo Count (only AT) (Read -only).
Build Status	**For AT tiles:** ▪ Unprocessed ▪ AT completed ▪ AT merged **For reconstruction tiles:** ▪ AT complete ▪ All steps through point cloud creation are complete ▪ All steps through model creation are complete ▪ All steps through model texturing are complete ▪ All steps through project output are complete (Read -only) **Note:** The tiles in the 3D Window are also colorized based on build status. **See** "Viewing Aerotriangulation (AT) Tiles" and "Viewing Reconstruction Tiles" in this chapter for information.
Error Status	Indicates if there was an error in creating the tile: Failed, Failed (retrying), No coverage (only reconstruction) or <80% success (only AT) (Read -only).
User Status	Click the drop-down arrow, and select the applicable status, indicating what actions still need to be performed on the tile, e.g., rebuild, editing or external retouch. You can also set the user status from the **Tile** tab > **User Status**. User status flags can be displayed in the 3D Window on each of the tiles to indicate at a glance the current status of each tile. **See** "Setting and Displaying User Status Flags and Legend" in this chapter for information. You can customize the name and color associated with each user status in the Options menu. **See** "Setting PhotoMesh Options" in the "Basic Concepts" chapter for more information. ▪ **For AT Tiles**: OK, Requires Rebuild, Requires Tie/Control Points. ▪ **For Reconstruction Tiles**: OK, Requires Rebuild, Requires Editing, Requires External Retouch. **Note:** User status can also be set from the Tiles tab (Tiles tab > User Status).

Comments	Free text field where you can add any applicable information about the tile. You can add comments from the **Tile** tab > **User Status**.

Navigating to a Tile

To fly or jump to a tile:

1. In the **3D Window**, click the tile you want to select, or select the tile(s) in the tile list. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information.

   Note: If multiple tiles are selected, navigation is to the last selected tile.

2. On the **Tile** tab, in the **View** group, click **Fly to Tile/ Jump To Tile**.

Displaying Original Camera Positions for Tiles

After the aerotriangulation step has begun, you can display photos' original camera positions.

To display photos in the original camera positions defined in the project:

1. In the **3D Window**, click the required tile, or select the tile(s) in the **AT Tiles** list. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information. The Tile tab is displayed.

2. On the **Tile** tab, in the **Original Photos** group, toggle on **Original**.

3. On the **Tile** tab, in the **Original Photos** group, in the **Color Code** drop-down, select one of the following options for color coding of the photos' camera symbols:

   ▪ **Status** – Camera symbols are color coded based on photo status (calculated/ not calculated).

   ▪ **Collection** – Camera symbols are color coded according to collection, with a random color assigned to each collection's camera symbol.

   Note: These settings can also be accessed from the View tab.

Displaying Calculated Camera Positions for Tiles

Calculated camera positions for photos can only be displayed after the aerotriangulation step is complete.

Note: When calculated camera positions are displayed, you can also view calculated property information in the Photos list. **See** "Calculated Camera Position Properties" in the "Photo Management" chapter for information.

To display photos in the camera positions calculated through aerotriangulation:

1. In the **3D Window**, click the required tile, or select the tile(s) in the **AT Tiles** list. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information. The Tile tab is displayed.

2. On the **Tile** tab, in the **Calculated Photos** group, toggle on **Calculated**. In the 3D Window, the photos are displayed in the camera positions calculated through aerotriangulation, and calculated positioning information is included in the Photos list. **See** "Photos List" in the "Photo Management" chapter for information.

3. On the **Tile** tab, in the **Calculated Photos** group, in the **Color Code** drop-down, select one of the following options for color coding of the photos' camera symbols in the 3D Window:

   ▪ **Status** – Camera symbols are color coded based on status (calculated/ not calculated).

   ▪ **Collection** – Camera symbols are color coded according to collection, with a random color assigned to each collection's camera symbol.

- **Median Error** – Camera symbols are color coded according to median error (px):
  - Green: px < 0.5
  - Orange: 0.5 < px < 1
  - Purple: px > 1
- **Reconstruction Group** – Camera symbols are color coded according to reconstruction group (i.e., photo blocks created in the AT process), with a random color assigned to each reconstruction group's camera symbol. This information is significant in identifying the largest group (i.e., the group PhotoMesh will perform reconstruction on) and determining which additional photos can be connected to this group using tie points and other methods, for optimal output.
- **Number of Matches** - Camera symbols are color coded according to the number of photo matches:
  - Red: matches <= 2
  - Orange: 2< matches <= 4
  - Green: matches > 4

Viewing an Aerotriangulation Tile's Sparse Point Cloud

A sparse point cloud is a representation of all photo features that were automatically found in two or more project photos, with their computed positions.

To view an AT tile's sparse point cloud:

1. In the **3D Window**, click the required tile, or select the tile(s) in the **AT Tiles** list. You can multi-select tiles using the tile list commands. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information. The Tile tab is displayed.
2. On the **Tile** tab, in the **Sparse Point Cloud** group, click **Show Point Cloud**.
   Note: The sparse point cloud may take some time to display the first time it is shown or after making changes to the project.
3. Select the **Point Size** in pixels.
4. Select the color coding option for classifying the point cloud's points:
   - **Single color** – A single color is used for all points.
   - **Num. tied photos** – Points are color coded according to the number of photos in which this point was found.
     - **Orange** – Two matches
     - **Green** – Three or more matches

Displaying Reconstruction Tile Results

To display point cloud, 3D model, or texture results for a tile:

1. In the **3D Window**, click the required reconstruction tile, or select the tile(s) in the **Reconstruction Tiles** list. You can multi-select tiles using the tile list commands. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information. The Tile tab is displayed.
2. On the **Tile** tab, in the **Show** group, toggle on any of the following options:
   - Point Cloud

- Model – Shows only the mesh model
- Textured Model
- Orthophoto – If RGB color bands were selected for the orthophoto, they are displayed. Otherwise, only the first band is displayed in grayscale mode. **See** "[Selecting the Collections and Bands to Include in the Build Outputs](#)" in this chapter for information.

Note: Each of these options is enabled only if the build step for this data was completed.

Setting and Displaying User Status Flags and Legend

You can set and display the user status of AT and reconstruction tiles. By default the user status options are used to indicate what actions still need to be performed on the tile, e.g., rebuild, editing or external retouch, but they can be customized as required from the Options menu. After setting user status, flags can be displayed in the 3D Window on each of the tiles to indicate at a glance the current status of each tile.

To set and display user status flags:

1. In the **3D Window**, click the required tile, or select the tile(s) in the **Reconstruction Tiles** or **AT Tiles** lists. You can multi-select tiles using the tile list commands. **See** "[Selecting Items Using the Item List](#)" in the "Using the Project Tree and Item List" chapter and "[Selecting Objects in the 3D Window](#)" in the "Basic Concepts" chapter for more information. The Tile tab is displayed.

2. On the **Tile** tab, in the **User Status** group, click the drop-down arrow and select the applicable status, indicating what actions still need to be performed on the tile, e.g., rebuild, editing or external retouch. **See** "[Viewing Tile Information in the Tile List](#)" in this chapter for more information.

   Note: You can customize the color and name associated with each user status in the Options menu. **See** "[Setting PhotoMesh Options](#)" in the "Basic Concepts" chapter for more information.

3. On the **View** tab, in the **Reconstruction Tiles** group, click the arrow under **User Status Flags**, and select **Show User Status Flags**.

4. If you want to show a user status legend that indicates what AT and reconstruction tile user status is associated with each color flag, on the same tab, click the arrow under **User Status Flags**, and select **Show User Status Legend**.
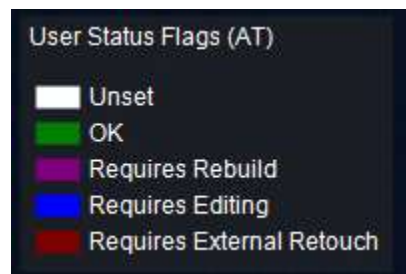
User Status Legend

Setting a Reconstruction Tile's Display Style

There are three options for the display style of selected reconstruction tiles:

- 2D rectangle
- 3D box

▪ 3D filled box

To set the display style for selected reconstruction tiles:

▪ On the **View** tab, in the Reconstruction Tiles group, click the arrow under **Selection Style** and select the style you want. If you want to display a frame around reconstruction tiles only when selecting a tile, and not when hovering over it, click the arrow under **Selection Style** and select **Hide Tiles Hover**.

Viewing a Tile's Build Report

The Build report contains statistical and graphical information about the build process. The first section of the report displays summary information:

▪ Basic project information including: project name, number of photos, and total input gigapixels.

▪ Basic aerotriangulation information including: AT area, number of aerotriangulation tiles, calculated photos, and mean median error. The mean median error is the average of the median reprojection errors for all photos in the collection.

▪ Basic ground control point and check point information

▪ Basic reconstruction information including: output area, number of reconstruction tiles, and output coordinate system.

▪ Performance information including: build and fuser time.

This summary section is followed on each report tab by detailed information and graphs relating to the respective tab:

▪ **Aerotriangulation** – Aerotriangulation details per photo collection (e.g. mean median error, focal length, and principal X/Y). **See** "Collection Property Sheet" in the "Photo Management" chapter for information about the properties displayed. Information is only displayed in this section if an aerotriangulation tile was selected.

  Note: If a report is generated while aerotriangulation is in progress, a message is displayed in the aerotriangulation section that the aerotriangulation is not yet complete.

▪ **(Build Process) Steps** – Details about each build step: steps involved, tiles produced, processing time, and active fusers. When a reconstruction tile is selected, details display about the tile building steps (point cloud, model, texturing). When an aerotriangulation tile is selected, details display about the aerotriangulation.

▪ **Fusers** – Statistics for each fuser regarding processing time for each build step and each tile in each step.

To view a build report:

1. Do any of the following:

   Note: The same build report opens in each of these options but with different filter values selected.

   ▪ On the **Home** tab, in the **Review** group, click the arrow under **Build Report**, and click one of the options: **AT Report**, **Steps Report**, or **Fusers Report**.

   ▪ Select a tile in the **3D Window** or in the Tile List, and then on the **Tile** tab, in the **View** group, click **Build Report**.

- In **PhotoMesh Build Manager**, click the **Build Report** button.
- In **PhotoMesh Build Manager**, right-click a fuser, and select **Fuser build report**.
- In **PhotoMesh Build Manager**, right-click a tile, and select **Tile build report**.

2. At the top of the **Build report**, select a report type. This determines what detailed information and graphs display in the section following the summary section.

   - AT (Aerotriangulation)
   - Steps (Build)
   - Fusers

3. The detailed information and graph data can be filtered to display information only about tiles with warnings or errors or about specific build process steps, fusers, or aerotriangulation tiles. Click **Filter** and, select one of the following:

   - All
   - With warnings
   - With errors
   - User selection

4. If you choose **User selection**, the Select dialog is displayed:

   a. Select the required step, fuser or AT tile (depending on the report type being viewed). Use CTRL and SHIFT to multi-select. Then click **OK**.

   b. If you want to change your selection, click **Select** to reopen the Select dialog.

Select Dialog Box

223
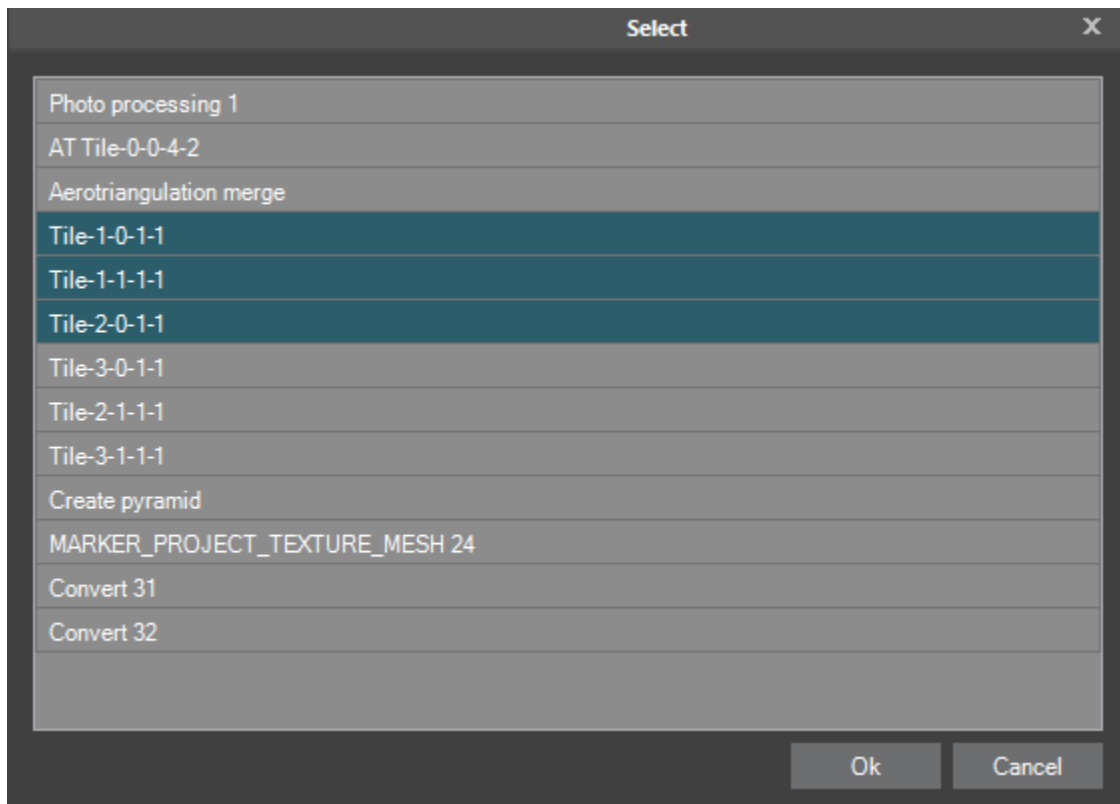
5. If you want to expand or collapse parts of the report to make it easier to find and view only the information you want, click **Expand/Collapse All** at the top of the report.

6. If you want to export the Build Report as an HTML file, at the top of the report, click **Export**. Then in the **Export Build Report** dialog box, browse to the required file location, and click **Save**.

7. If you want to print the report, at the top of the report, click **Print**.

Viewing an Aerotriangulation Tile's Detailed Report

View information about each photo included in a selected aerotriangulation tile.

To view detailed aerotriangulation information:

In the **3D Window**, click the required aerotriangulation tile, or select the tile(s) in the **AT Tiles** list. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter and "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for more information. The Tile tab is displayed. **See** "Selecting Tiles" in this chapter for more information.

1. On the **Tile** tab, in the **View** group, click **Detailed Report**. The Aerotriangulation Detailed Report is displayed showing two tables:

   ▪ Inputted and calculated aerotriangulation details **per photo collection** (e.g., mean median error, focal length, and principal X/Y). Note that this table displays the same information as in the tile's build report for aerotriangulation, with the addition of input data for each property and principal point data.

   ▪ Inputted and calculated positioning and orientation values **for each of the photos** (organized by collection) in the selected tile.

Detailed Aerotriangulation Report

2. If you want to export the Detailed Report as a .txt file, click **Export**. Then in the **Export Aerotriangulation Report** dialog box, browse to the required file location, and click **Save**.

Exporting Tiles

Selected AT or reconstruction tiles with all their information (e.g., user status, comment, photo count) can be exported to an SQLite file, and then imported into TerraExplorer or standard geospatial applications. This allows you to easily share information about the project's data coverage. The exported tiles can also be reviewed in TerraExplorer or other geospatial applications, and user status and comments added, and then reimported to PhotoMesh. **See** "Importing Tiles" in this chapter for information.

To export a tile:

In the **Project Tree**, select the required **AT Tiles** or **Reconstruction Tiles** group.

In the **AT Tiles** list or **Reconstruction Tiles** list, select the required tile(s), right-click and select **Export Selected Tiles**. The Export Tiles dialog box is displayed. **See** "Selecting Tiles" in this chapter for more information.

Browse to the required folder, and click **Save**.

Importing Tiles

After exported tiles were reviewed in TerraExplorer or other standard geospatial applications, and user status and comments were added, the tiles can be reimported to PhotoMesh. **See** "Exporting Tiles" in this chapter for information.

To import a tile:

In the **Project Tree**, select the required **AT Tiles** or **Reconstruction Tiles** group.

In the **AT Tiles** list or **Reconstruction Tiles** list, right-click and select **Import Tile Status**. The Import Tile Status dialog box is displayed.

Browse to the required folder, and click **Open**.

Working with Water Body Polygons

Water bodies are difficult to model accurately because it is hard to identify and match features due to the uniformity and non-static nature of water. In order to enable PhotoMesh to model and texture a water body successfully, you can define the required water body by drawing or importing a corresponding polygon. PhotoMesh uses this polygon to generate a flat surface over the user-defined water body, and automatically reconstruct any above-water elements – boats, piers, small islands without requiring special user markings. Water bodies can be either of the following:

- **Leveled** – A constant water level (i.e., altitude) is set for all points in the water body polygon, to create level water bodies such as seas, lakes, ponds, and pools. This is especially useful for creating a level water body on top of a non-flat terrain or mesh, or for creating a flat water body based on an imported polygon with variable altitude values.

- **Non-Leveled** (Sloped) – PhotoMesh segments the water body polygon into multiple flat reconstruction tiles, each of which has a different elevation value based on the average elevation within the specific tile. This option should be selected to create an elevation gradient for water bodies such as rivers, with non-uniform water levels.

Imported polygon layers are each saved separately in the Project Tree, while new polygons that you manually create are saved into the "Water - Manually Created" group under the Retouch Layers group in the Project Tree. If you open a legacy project with water polygons, the water polygons are converted to the new feature layer format during the load. The project must then be saved to save the changes.

Creating Water Body Polygons

To create a water body polygon:

1. On the **Home** tab, in the **Add** group, click the arrow under **Retouch**, and select either of the following:
   - Create New Water Body
   - Create Leveled Water Body

   Note: You can also create a water polygon by selecting **Draw Water Body Polygon** or **Draw Leveled Polygon** in the Polygon list:

2. With the cursor in the **3D Window**, click to add the first point of the polygon. Click again to add additional points of the polygon until you have marked the entire shape. You must place at least three points to define the polygon.

   Note: It is sometimes easier to draw an accurate water polygon over a textured mesh model. After adding the model, rebuild, setting the **Build Start From** field to **Auto**, so that the required steps are automatically selected.

   Note: When creating a water body polygon, the lines of the polygon should not overlap each other.

   Note: It is recommended to keep the number of polygon points to the minimum required to define the shape. Adding unnecessary points may adversely affect performance.

3. Right-click to finish the polygon. The polygon name and water average elevation can be modified in the Water Polygon list. **See** "Water Polygon List" in this chapter for information. If you want to edit the polygon, **see** "Editing a Water Body Polygon's Shape" and "Setting Water Body Polygon Properties" in this chapter for information.

Importing Water Body Polygons

To import a water polygon shapefile or KML layer:

1. On the **Home** tab, in the **Add** group, click the arrow under **Retouch**, and select **Import Water Layer**. The Import Water Body Polygon Layer dialog box is displayed.

2. Browse to the water polygon layer you want to import, and click **Open**. If you want to edit the polygons, **see** "Editing a Water Body Polygon's Shape" and "Setting Water Body Polygon Properties" in this chapter for information.

Exporting Water Body Polygons

To export a water polygon layer to a shapefile:

1. Save the project, and then on the **Home** tab, in the **Add** group, click the arrow under **Retouch**, and select **Export Water Layer**. The Export Water Body Polygons dialog box is displayed.

2. Browse to the folder in which to save the layer, type a **File name**, and click **Save**.

Editing a Water Body Polygon

Note: After modifying a water body polygon, all affected tiles (e.g., tiles that now are intersected by the polygon) require rebuilding from the point cloud step. PhotoMesh will automatically determine what processing is required following the water polygon change, by rebuilding the project from "Auto".

To move a water polygon or modify its shape:

▪ In the Project Tree, select the required water polygon layer, and then in the Polygon List, select the required polygon, and click **Edit Water Body** . **See** "Editing Polylines and Polygons" in this chapter for information on editing polygons.

Selecting Water Body Polygons

Polygons can be selected from the Water Polygon list to perform various commands on them, e.g., jump to a polygon, level/unlevel, or set average elevation.

To select water body polygons:

▪ In the **3D Window**, click the polygon you want to select, or select the polygon in the **Water Polygon** list. You can multi-select tiles using the Water Polygon List commands. **See** "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter and "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter for more information.

Navigating to a Water Body Polygon

To fly or jump to a water body polygon:

1. In the **Project Tree**, select the **Water (Manually Created)** group or a **Water (Imported)** layer .
2. In the **Polygon List**, right-click the required polygon, and select **Fly To/Jump To**.

Setting Water Body Polygon Properties

Water body polygon data can be viewed and modified in the water polygon property sheet and Polygon List. In the Water Polygon List, you can set average elevation of the water polygons, as well as polygon visibility and inclusion in the project. **See** "About the Item List" in the "Using the Project Tree and Item List" chapter for information on working with the Water Polygon List.

To display and modify water body polygon properties:

1. In the **Project Tree**, click the **Water (Manually Created)** group or a **Water (Imported)** layer.
2. For group/layer properties: in the property sheet, modify the property value as required. **See** "Water Polygon Layer Property Sheet" in this chapter for more information.
3. For individual polygon properties: In the **Water Polygon List**, select the check boxes of the required polygons, and modify their properties as required. **See** "Water Polygon List" in this chapter for more information.

    Note: You can also search the list for the required polygons. **See** "Selecting Items Using the Item List" in the "Basic Concepts" chapter for information.

Water Polygon Layer Property Sheet

The properties in the water polygon layer property sheet apply to all polygons in the layer.

Property	Description
Position	
Altitude Offset	Altitude offset from the current altitude value applied to entire water body layer.
Coordinate System	Coordinate system of all the water polygons in the layer (Read-only for manually created polygons).
General	
Name	Name of the water body layer in the Project Tree.
Excluded	Select the check box to exclude the water body layer from the project. In the Project Tree, the layer is displayed in strikethrough font.
Original Source Path	Original source path from where the water body layer was imported (Read-only).
Fill Color	Fill color for all the water body polygons in the layer.

Leveled	Select the check box to set a constant water level (i.e., altitude) for all points in all the water body polygons in the layer, to create level water bodies such as seas, lakes, ponds, and pools. This is especially useful for creating level water bodies on top of a non-flat terrain or mesh, or for creating flat water bodies based on imported polygons with variable altitude values. If the check box is cleared, PhotoMesh segments the water body polygons into multiple flat reconstruction tiles, each of which has a different elevation value based on the average elevation within the specific tile. This option should be selected to create an elevation gradient for water bodies such as rivers, with non-uniform water levels.

Water Polygon List

In the **Water Polygon** list, you can view and set certain polygon properties, as well as set polygon visibility and inclusion in the project. **See** "[About the Item List](#)" in the "Using the Project Tree and Item List" chapter for information on working with the Polygon list.

Property	Description
Selection	Select the check box to select the polygon in the 3D Window. You can select multiple polygons to perform actions on multiple polygons.
Excluded	Select the check box to exclude the polygon from the project. The polygon continues to display in the 3D Window.
Visible	Select the check box to show the polygon in the 3D Window.
Name	Name of the water polygon.
Leveled	Select the check box to set a constant water level (i.e., altitude) for all points in the water body polygon, to create level water bodies such as seas, lakes, ponds, and pools. This is especially useful for creating a level water body on top of a non-flat terrain or mesh, or for creating a flat water body based on an imported polygon with variable altitude values. If the check box is cleared, PhotoMesh segments the water body polygon into multiple flat reconstruction tiles, each of which has a different elevation value based on the average elevation within the specific tile. This option should be selected to create an elevation gradient for water bodies such as rivers, with non-uniform water levels. If a water polygon is leveled, all its nodes are set to the elevation of the first node drawn. **Note:** If you want to level or unlevel multiple polygons, multi-select the required polygons. Then right-click in the **Water Polygons** list, and select **Level/Unlevel**. Unleveling allows you to set different elevations for each polygon node. **See** "[Selecting Water Body Polygons](#)" in this chapter for more information.
Average Elevation	The average elevation of the water polygon. When this value is edited, PhotoMesh proportionally adjusts the elevation value of

each of the individual nodes, so that the ratio between node elevations is still maintained.

Note: If you want to set the average elevation of multiple polygons, multi-select the required polygons. Then right-click in the **Water Polygons** list, and select **Set Average Elevation**. You can also modify the average elevation of multiple polygons by a specified delta. To do so, multi-select the required polygons. Then right-click in the **Water Polygons** list, and select **Set Elevation Delta**. **See** "Selecting Water Body Polygons" in this chapter for more information.

Deleting Water Body Polygon Layers

To delete a polygon layer:

▪ In the **Project Tree**, right-click the required polygon layer, and select **Delete**.

Deleting Selected Water Body Polygons

To delete a selected polygon from a layer:

1. In the **Project Tree**, select the layer.
2. In the **Polygon** list, right-click the required polygon, and select **Delete**.

    Note: You can also select the required polygon(s) in the **3D Window**, and then right-click in the **Polygon** list, and select **Delete**. Use CTRL-click or SHIFT-click to multi-select polygons in the 3D Window.-

Editing Polylines and Polygons

Water body polygons and retouch polygons and polylines are edited using the commands in the mini toolbar. This toolbar is displayed whenever the polygon/polyline is selected.

To edit a polygon/polyline:

1. Select the polygon/polyline, and then do either of the following:
    ▪ Move the polygon/polyline
    ▪ Modify the polygon/polyline shape
2. If you want to select a different object from the one currently being edited, click **Select another object** .
3. If you want to align your polygon/polyline to a surface, click **Align to Surface** . The polyline/polygon's nodes are aligned to the surface.

To move a polygon/polyline:

1. Click the **Edit Object icon** .
2. Do any of the following:

- If you want to set position based on mouse **and** snap the polygon/polyline to another object, click **Snap Based on Snapping Options** icon [icon] (Shortcut key: 4). **See** "Setting Snapping Options" in this chapter for information.
- If you want to set position based on mouse without snapping the polyline/polygon to another object, click **Set Position Based on Mouse** [icon] (Shortcut key: 3).
- If you want to move polygon/polyline in the XY plane, click **XY Plane** [icon].
- If you want to move the polygon/polyline in the Z plane, click **Z Axis** [icon].

3. Click the polygon/polyline, and drag it to the required location/altitude. Release the left mouse button to set the polygon/polyline at the current location/altitude. If snapping to an object, move and snap the polygon/polyline to an edge or vertex of any existing object.

To modify a polygon/polyline's shape by moving its nodes:

1. Click **Edit Nodes** [icon].
2. Do any of the following:
   - If you want to set position based on mouse **and** snap the polygon/polyline to another object, click **Snap Based on Snapping Options** icon [icon] (Shortcut key: 4). **See** "Setting Snapping Options" in this chapter for information.
   - If you want to set position based on mouse without snapping the polygon/polyline to another object, click **Set Position Based on Mouse** [icon] (Shortcut key: 3).
   - If you want to move a node in the XY plane, click **XY Plane** [icon].
   - If you want to move a node in the Z plane, click **Z Axis** [icon].
3. In the **3D Window**, place the cursor over one of the polyline/polygon's nodes. The node turns blue.
4. Click the node and drag it to the required location in the XY/Z Plane. Release the mouse button to set the node at the current location.

Setting Snapping Options

You can set how a node or object snaps to other objects.

To set snapping options:
1. Select a single object for editing.
2. On the top bar of the object's property sheet or mini toolbar, click **Set Snapping Options** [icon].
3. Toggle on any of the following options:
   - **Node Snapping** – Snap object or node to nodes of other objects or features.
   - **Line Segment Snapping** – Snap object or node to line segments of other objects or features.
   - **3D Snapping** – Smart snap object or node to dynamically-identified edges and corners of any element in the 3D World.
   - **Directional Snapping** – Constrain the object line segments currently being drawn to pre-defined angle increments. The angle constraint can either be absolute, i.e., relative to true North or relative to the previous line segment.
   - **Segment Distance on Edit** – Show segment lengths while editing a polygon/polyline.

230

Retouching a Mesh Layer

Mesh imperfections, such as bumps, irregular surfaces, or floating artifacts, can be flattened, filled, and cleaned up in PhotoMesh using the Manual Retouch tool. This tool allows you to mark the surfaces for flattening and floating artifacts for removal. The output of the Manual Retouch tool is a polygonal feature "retouch layer" with attribute information that defines what retouch action should be performed in each area: clean, flatten, and retexture. When rebuilding the project, PhotoMesh uses the manual retouch layer to add a set of constraints to the reconstruction algorithms. Most of the constraints defined apply to the mesh model, and the modified model is then retextured using the project's photos. There is also an option in the Manual Retouch tool to set a specific file texture, e.g., water or grass, for a marked area, instead of basing texturing on the project's photos.

You can run the manual retouch tool either on the reconstruction tile's 3D textured models or on the final 3DML. All retouch operations are listed in the Manual Retouch list.

To use the Manual Retouch tool:

Note: The shortcut keys only work when keyboard focus is on the retouch toolbox or the 3D Window.

1. Do either of the following:
   - In the **3D Window**, click the required reconstruction tile. Then on the **Tile** tab, in the **Show** group, toggle on **Textured Model**.
   - In the **Project Tree**, in the **Outputs** folder, under the required Build folder, select the **3DML output**.

2. On the **Home** tab, in the **Add** group, click the arrow under **Retouch**, and select **Open Manual Retouch Toolbox**. The Manual Retouch Toolbox is displayed.

   Note: You can also open the Retouch tool by clicking **Manual Retouch** under the **Retouch Layers** group in **the Project Tree**.

   Note: You can also turn on global selection, and then select a manual retouch polygon/polyline in the 3D Window. **See** "Selecting Objects in the 3D Window" in the "Basic Concepts" chapter for information.
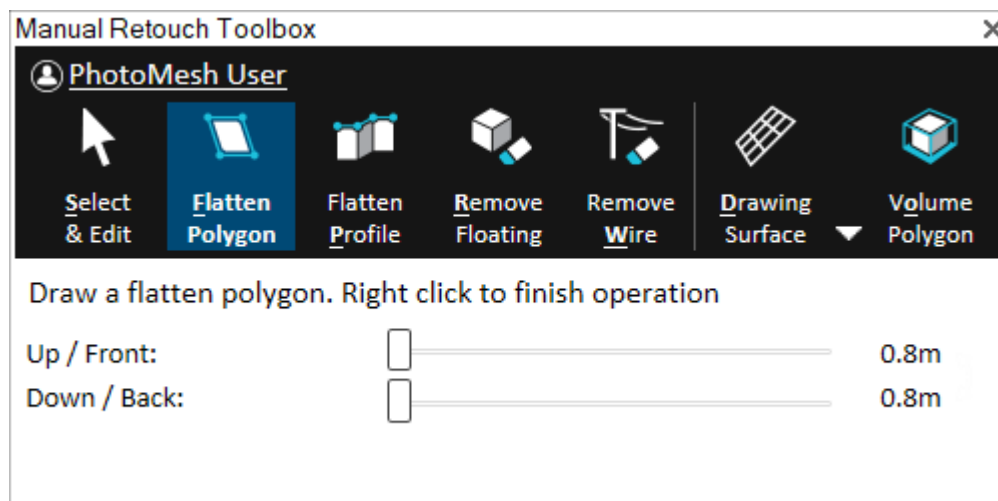
Manual Retouch

3. If you want to tag your edits with your name, click the user name link, and enter your user name. This information is saved in the layer's username attribute. When multiple users work on the same retouch

layer, this attribute enables you to identify the particular user that last worked on each retouch polygon.

4. If you want to create a surface on which to draw your retouch polygon or polyline, e.g., for a model that is not flat or missing corners, click the arrow next to **Drawing Surface** (Shortcut key: d) and select one of the following options:

- **Show Last** – Display the last surface that was created.

- **Edit** – Open the surface property sheet to edit altitude, x, y, yaw, pitch, and roll properties, and use the toolbar to edit the surface in the 3D Window. **See** "Editing Polylines and Polygons" in this chapter for information.

  - Click one of the other manual retouch commands and draw a polygon or polyline whose nodes are all lying in the same plane. Selecting **Set Position Based on Mouse** or **Snap Based on Snapping Options** in the polygon property sheet before drawing makes it easier to draw the polygons on a vertical plane.

- **Horizontal** – The surface is aligned horizontally to the area under the cursor.

- **Vertical** – The surface is aligned vertically to the area under the cursor.

- **3 Points** – The surface is aligned to the polygon created by the three points you draw.

5. Perform one of the following retouch operations:

- **Flatten polygon** (Shortcut key: f) – Remove all elements inside a defined volume, and add a flat surface in the area of the 2D polygon. For e.g., filling holes and flattening walls.

  i. Click **Flatten Polygon**.

  ii. Draw a polygon around the area for flattening. Right-click to complete.

- **Flatten profile** (Shortcut key: p) – Create multiple flatten polygons by defining a profile line (usually on a roof top) and the base elevation. A flatten polygon is created for each segment of the profile line (from the line to the base elevation). This method is useful for creating retouch polygons in hard to access places, e.g., between buildings.

  i. Click **Flatten Profile**.

  ii. Draw the profile line, and then right-click to complete.

  iii. Use the property sheet's toolbar to edit the polygon. **See** "Editing Polylines and Polygons" in this chapter for information. Close the mini toolbar when finished editing.

  iv. Place the "X" on the model to define the base elevation.

- **Remove floating** (Shortcut key: r) – Remove all elements inside a defined volume. This option is generally used to remove floating elements that aren't connected to the ground.

  i. Click **Remove Floating**.

  ii. Click any point on the floating artifact. PhotoMesh automatically creates a polygon that includes the selected point.

- **Remove wire** (Shortcut key: w) – Remove thin elements such as power lines by drawing a polyline that marks the wire to be removed. A separate floating polygon is created for each segment of the polyline.

  i. Click **Remove Wire**.

  ii. Draw a polyline along the wire to remove, and then right-click to complete.

6. To select the polygon for editing after it is drawn, click **Select and Edit** (Shortcut key: s), and then select the retouch object for editing. Use the mini toolbar to edit the polygon. **See** "Editing Polylines and Polygons" in this chapter for information.

7. To adjust the length to extrude the retouch polygon upwards or frontwards and downwards or backwards in the direction of the polygon's normal, select the polygon and in the **Manual Retouch Toolbox**, slide the **Front buffer** and **Back buffer** sliders. If a polygon's elevation angle to the ground is less than 20 degrees, the polygon is extruded vertically. If the polygon points were created in a clockwise orientation, then the front is the side facing the user.

8. If you want to add water or grass texture to your flatten polygon (created via "Flatten Polygon" or "Flatten Profile"), select the polygon, and then in the **Manual Retouch Toolbox**, select a **Texture file**. Enter a value for the texture tile size. If you want to remove the added texture, select the first entry which is blank in the **Texture file** list.

9. To select how to display the drawn polygon, click the **Volume** icon (Shortcut key: o):

   - **Volume Colorize** – Mark the area for retouching by colorizing all parts of the mesh contained within the extruded polygon.
   - **Volume Polygon** – Mark the area for retouching by displaying 3D polygons based on the extruded polygon.
   - **Volume No –** Mark the area for retouching only with a 2D polygon.

Modifying Retouch Polygons

All retouch operations can be viewed and edited from the Manual Retouch list. **See** "Selecting Items Using the Item List" in the "Using the Project Tree and Item List" chapter for more information.

Selection	User Name	Operation	Front Buffer	Back Buffer	Texture
✓	Jon	Clean	2	1	
	Max	Clean	2	1	
	Jen	Clean	1	1	
	Angie	Clean & Fill	1	1	
✓	Jon	Clean & Fill	2.8	1	
	Max	Clean & Fill	2.8	1	
	Jon	Clean & Fill	2.8	1	
	Jon	Clean & Fill	2.2	0.2	
	Jen	Clean & Fill	2.2	0.2	
	Max	Clean & Fill	2.2	0.2	
	Max	Clean & Fill	2.2	0.2	
	Max	Clean	2.2	1	
	Angie	Clean & Fill	2.2	1	
	Angie	Clean & Fill	2.2	1	

Manual Retouch List

To modify a retouch operation:

1. In the **Project Tree**, select the **Manual Retouch** layer, and then in the **Manual Retouch** list, select the required row.

2. Perform any of the following actions:

   - **Fly To/Jump To** – Right-click the required row, and select **Fly To/Jump To**.
   - **Delete Polygon** – Right-click the required row, and select **Delete**.

233

▪ **Edit Polygon** – Right-click the required row, and select **Edit**. The retouch polygon is selected in the 3D Window and a mini toolbar is displayed. **See** "[Editing Polylines and Polygons](#)" and "[Marking a Mesh Layer for Retouching](#)" in this chapter for information on modifying the polygon shape/position and properties. Some properties can be modified directly in the Manual Retouch list.

Exporting and Importing a Retouch Layer

You can export retouch layers with selected retouch polygons to an SQLite file. The layer can be modified in TerraExplorer and reimported to PhotoMesh.

To export a retouch layer:

1. In the **Project Tree**, select the **Manual Retouch** layer, and then in the **Manual Retouch** list, select the retouch polygons that you want to export, right-click and select **Export**. The layer is exported with the selected retouch polygons as an SQLite file.
2. Browse to the required folder, and click **Save**.

To import a retouch layer:

1. In the **Project Tree**, select the **Manual Retouch** layer, and then in the **Manual Retouch** list, right-click and select **Import**.
2. Browse to the required SQLite file, and click **Open**. The layer is imported to the project.

Exporting Aerotriangulation Results

You can export your aerotriangulation results either as multiple tiles or as a single unified SQLite or XML file.

To export your aerotriangulation results:

1. On the **Home** tab, in the **Review** group, click the arrow under **Build Report** and select **Export AT**. The Export Aerotriangulation Results dialog box is displayed.
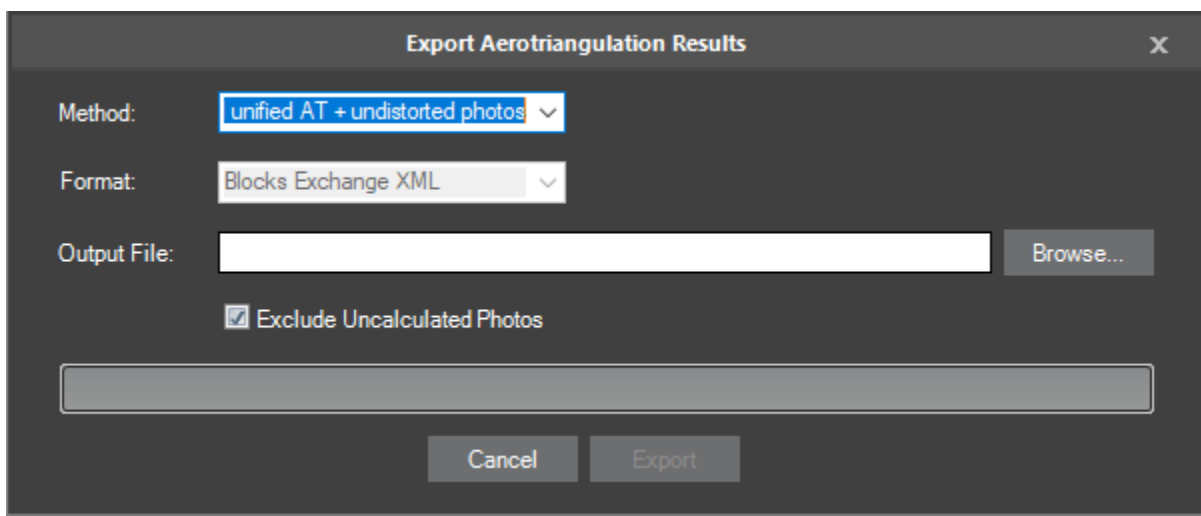
Export Aerotriangulation Dialog Box

2. In the **Method** field, select how to export the AT results:

- Unified AT – Create a single, unified file from all AT tiles for export of AT information into a single set of AT results for each photo and camera. The unified AT includes the average AT results for photos/cameras that participate in more than one AT tile .

- Multiple AT Tiles – Export each AT tile to a separate file.

- Create a unified AT (see description above) + generate undistorted photos based on the distortion parameters calculated in the AT process.

    Note: When photos are loaded into a project, most include distortions resulting from camera lens properties. In the build's photo preparation step, PhotoMesh calculates the parameters necessary to undistort the photos, and then uses these parameters to properly perform feature matching and other AT sub-steps. If this option is selected, these parameters are also used to create and save undistorted versions of all the project's photo files.

3. Select a **Format**:
   - Blocks Exchange XML – An open exchange format for import/export of AT data.
   - SQLite - Skyline internal format for import/export of AT results.

4. In the **Output File** field, click **Browse**, and then in the **Save Aerotriangulation Results** dialog box, do the following:

   a. Browse to the folder in which you want to save the AT results.

   b. Type the **File Name** for your results, and click **Save**.
      Note: If the results are being exported as multiple tiles, the naming system for each of the tiles is as follows: File Name_AT_Tile-[*Tile* #], e.g., FileName_ AT_Tile-0-0-1-1.xml, FileName_ AT_Tile-0-1-1-1.xml, etc.

5. If you want to **exclude uncalculated photos** from the exported AT results, select the check box. If uncalculated photos are not excluded, they are exported with original position and orientation values.

6. Click **Export**.

3DML Display Style Options

Several style options are available for displaying the mesh model output. The mesh model can include a wireframe and be textured or solid color.

To select a mesh layer display style:

- On the **View** tab, in the **Mesh Layer** group, select the arrow under **Display Style** and select a style:

    - Texture
    - Texture + wireframe
    - Solid color
    - Solid color + wireframe
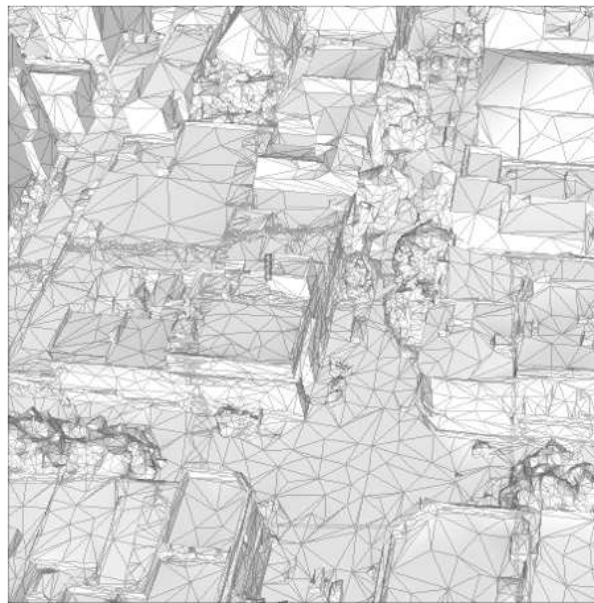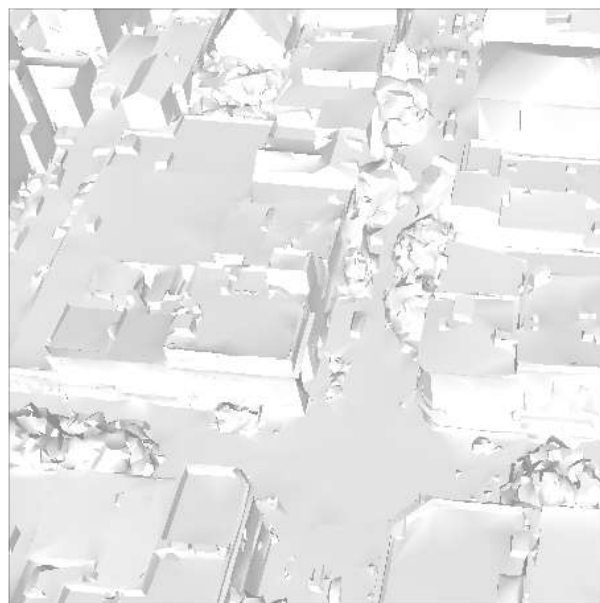
Managing Build Versions

To manage your build versions:

1. On the **Home** tab, in the **Build** group, click **Versions Manager**. The Build Versions Manager is displayed.

Build Versions Dialog Box

2. If you want to load a different version than the current one, select the version and click **Load**. If the current version was not saved, a dialog box is displayed asking if you want to save it before the selected version is loaded. Click **Yes** to save.

3. If you want to delete a version, select the version and click **Delete**.

237

CHAPTER 18   IMPORTING AND EXPORTING MODELS

About Importing and Exporting Models

After a 3D mesh model is created in Skyline PhotoMesh, certain tiles may require editing, e.g. to remove a particular building that is being torn down, flatten surfaces and correct certain imperfections such as bumps or irregular surfaces, or edit textures. These edits, whether they involve mesh or texture retouching, can be performed in external software, such as Autodesk® MeshMixer, by exporting the required OBJ models and then reimporting them to PhotoMesh in order to rebuild the project. When edits are complete, you can reimport precisely the data you want to import, all in one step:

- Only textures, using the models generated by PhotoMesh before the edits

- Only models (in which case retexturing is required in PhotoMesh)

- Both models and textures

You can import a single model or a folder with multiple model files, in which case each model file is read into its respective tile as long as its original file name is maintained.

Exporting a Point Cloud

To export a point cloud:
1.   On the **Home** tab, in the **Tiles** group, click **Reconstruction Tiles**.
2.   Click the tile whose point cloud you want to export. You can use CTRL-click or SHIFT-click to multi-select tiles. The Tile tab is displayed.
3.   On the **Tile** tab, in the **Export** group, click **Point Cloud**. The Export Point Cloud dialog box is displayed.
4.   Browse to the required location, and click **Save**.

Exporting a Model

To export a model:
1.   On the **Home** tab, in the **Tiles** group, click **Reconstruction Tiles**.
2.   Click the tile whose model you want to export. You can use CTRL-click or SHIFT-click to multi-select tiles. The Tile tab is displayed.
3.   On the **Tile** tab, in the **Export** group, click **Model**. The Export Model Folder dialog box is displayed.
4.   Browse to the required location, and click **Save**.
   Note: When reimporting an exported model file after editing in external software, the model file must have the original file name with which it was exported to enable PhotoMesh to update the respective tile based on the edited model.

Importing a Model

To import a single model:
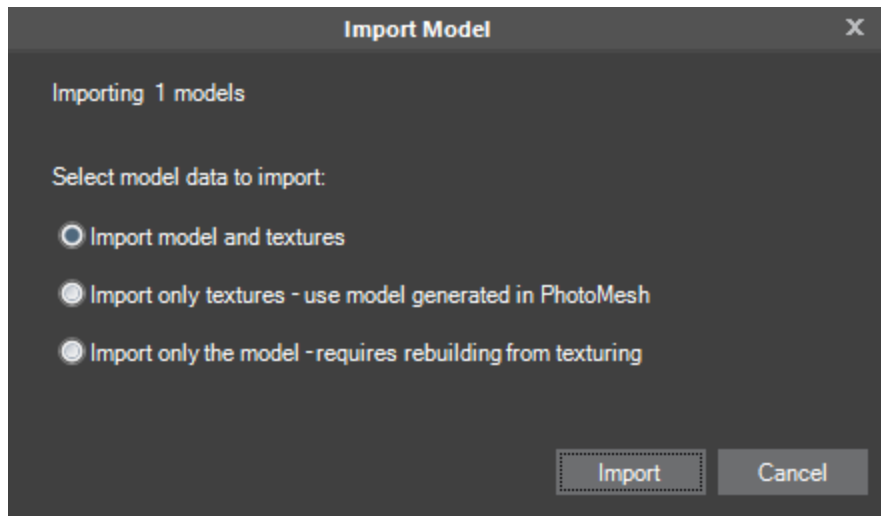1.   On the **Home** tab, in the **Tiles** group, click **Reconstruction Tiles**.
2.   Click the tile for which you want to import a model. The Tile tab is displayed.
3.   On the **Tile** tab, in the **Import** group, click **Model**. The Import Model dialog box is displayed.
4.   Browse to the required model and click **Open**. The Import Model dialog box is displayed.

Import Model Dialog Box

5. Select the model data to import and then click **Import**:
   - Import model and textures
   - Import only textures – use model generated in PhotoMesh
   - Import only the model – requires rebuilding from texturing
6. After reimporting, rebuild using the "Build" command starting from "Auto", to have PhotoMesh automatically determine what processing must be performed to complete the project. **See** "Rebuilding" in the "Building" chapter for information.

To import a folder with multiple models:
1. On the **Home** tab, in the **Add** group, click **Import Models**. The Import Model Folder dialog box is displayed.
2. Browse to the folder that contains the edited models and click **Select Folder**. The Import Model dialog box is displayed.

   Note: The model files must have the original file names with which they were exported to enable PhotoMesh to update the respective tiles based on the edited models.
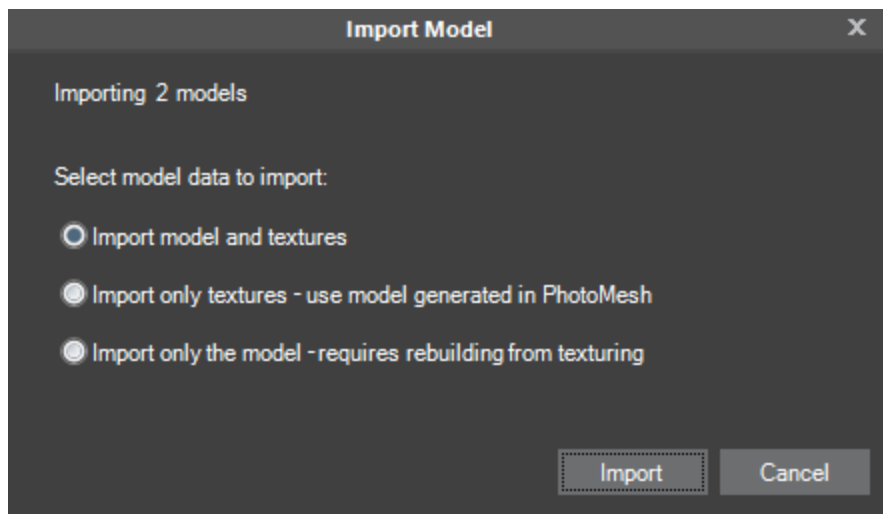
Import Model Dialog Box

3. Select the model data to import and then click **Import**:

- Import model and textures
- Import only textures – use model generated in PhotoMesh
- Import only the model – requires rebuilding from texturing

4. After reimporting, rebuild using the "Build" command starting from "Auto" and building until "Output", to have PhotoMesh automatically determine what processing must be performed to complete the project. **See** "Rebuilding" in the "Building" chapter for information.

CHAPTER 19 ADDING TERRAIN OBJECTS AND LAYERS

About Terrain Objects and Layers

Terrain objects and layers can be added to a PhotoMesh project to enable you to visualize how the model will appear in TerraExplorer's 3D World.

You can add any of the following objects and layers:

- Locations

- Imagery and Elevation Layers

- 3D Mesh Layers

- Terrain Objects (Modify Terrain and Hole on Terrain)

Locations

A location defines the geographical coordinates of a point on the terrain as well as the position from which the point is to be viewed.

To create a new location in the current camera position:

- On the **Tools** tab, in the **Objects and Layers** group, click **Create Location**. The location of interest is set to the terrain coordinate at the center of the 3D Window. If there is no terrain at the center of the view (the camera is facing the sky), the location of interest is set to a distance of two times the current altitude of the camera above the terrain. The location is added to the Project Tree in the Objects and Layers section.

  Note: When you create a location, the point of interest is set only to the terrain coordinate of the center of the 3D Window and is not affected by any 2D or 3D models in front of the terrain.

Imagery and Elevation Layers

Raster layers can be added to your project from any of the following:

- Files on your own computer or on your local network.

- A database that is served over a network by one of these GIS servers:
  - Web Map Service
  - Web Map Tile Service
  - Oracle Spatial Database
  - ECW Image Web Server
  - TerraGate Server
  - SkylineGlobe Server (**See** the "Working with SkylineGlobe Server" chapter for information)
  - ArcSDE Raster Server

See the TerraExplorer User Guide for more information.

To add an imagery/elevation source to a project:

1. On the **Tools** tab, in the **Objects and Layers** group, click **Imagery Layer/Elevation Layer**. The Load Imagery/Elevation Layer dialog box is displayed.

2. Browse to the required file and select it, or select the server type and enter the required information. **See** the TerraExplorer User Guide for further information.

3. Click **Open**.

   Note: If the source file is geo-referenced, it is located accordingly on the terrain. Otherwise, it is placed at the center of the 3D Window. You can then manually edit its position, or reproject it.

4. After the layer has been placed on the terrain, you can change its properties using the property sheet. **See** "Imagery and Elevation Layer Parameters" in the TerraExplorer User Guide for further information.

3D Mesh Layers

3D Mesh Layers (3DML) can be loaded from 3DML files on your own computer or local network or from a database that is served over a network by SkylineGlobe 3D Mesh service.

To load a mesh layer:

1. On the **Tools** tab in the **Objects and Layers** group, click **3DML**. The Browse for 3DML Layer dialog box is displayed.

2. Browse to the required file, and click **Open** or select the required server type, and click **Select**. Then enter the information required for the selected server. **See** the "Working with SkylineGlobe Server" chapter for information.

3. After the layer has been placed on the terrain, you can change its properties using the property sheet. **See** "3D Mesh Layer Parameters" in the TerraExplorer User Guide for further information.

Terrain Objects

The following terrain objects can be added to a PhotoMesh project to enable you to visualize how the model will appear in TerraExplorer's 3D World:

- Modify terrain

- Hole on terrain

Modifying the Terrain

Using the Modify Terrain tool, you can modify the elevation values in an area of the terrain by drawing a polygon in the 3D Window and then setting the Modify Terrain object's properties in its property sheet. The surface of the terrain is raised or lowered according to the elevation values of the polygon's points. Different elevation behaviors allow replacing, cropping above or cropping below the original values of the terrain. The interior of the polygon can be flat or irregular in shape.

To modify the terrain:

1. On the **Tools** tab, in the **Objects and Layers** group, click **Modify Terrain**. The Modify Terrain property sheet is displayed.

2. With the cursor in the **3D Window**, click to add the first point of the polygon.

3. Click again to add additional points of the polygon until you have marked the entire shape. You must place at least three points to define the polygon.

   Note: When creating a Modify Terrain object, the lines of the polygon should not overlap each other.

> **Note**: It is recommended to keep the number of polygon points to the minimum required to define the shape. Adding unnecessary points may adversely affect performance.

4. Right-click to finish the object.

5. Continue to edit the object's parameters and shape, or close the property sheet to finish the operation. **See** "Modify Terrain Property Sheet" in the [TerraExplorer User Guide](#) for further information.

Cutting a Hole on Terrain

Cutting a hole on the terrain removes the terrain texture while maintaining the original contours of the terrain. The objects or feature layers created on the terrain in the area of the hole are drawn on the original contours of the terrain.

To cut a hole on the terrain:

1. On the **Tools** tab, in the **Objects and Layers** group, click **Hole on Terrain**. The Hole on Terrain property sheet is displayed.

2. With the cursor in the **3D Window**, click to add the first point of the hole.

3. Click again to add additional points of hole until you have marked the entire shape.

4. Right-click to finish the hole.

> **Note**: It is recommended to keep the number of hole points to the minimum required to define the shape. Adding unnecessary points may adversely affect performance.

5. Continue to edit the hole or close the property sheet to finish the operation. **See** "Hole on Terrain Property Sheet" in the [TerraExplorer User Guide](#) for further information.

CHAPTER 20 ANALYSIS TOOLS

About Analysis Tools

The Tools tab provides you with a set of TerraExplorer analysis tools that help you validate photo parameters.

| Tool	Description
Distance	Displays the aerial distance, elevation difference, horizontal distance and slope angle between two or more points in the 3D Window.
Area	Measure the following for a defined area on the terrain: ▪ **Horizontal plane** – Displays the measurement of the horizontal projection of an area in the 3D Window. ▪ **Terrain surface** – Displays the measurement of the marked area while taking into account terrain contours.
Query	Provides data about any point or object in the 3D World.

Distance Tool

The Distance tool allows you to measure the aerial distance, elevation difference and slope between two or more points. The aerial distance is the actual distance between the points.

To use the Distance tool:

1. On the **Tools** tab, in the **Measure** group, click **Distance**. The Distance Measurement dialog box is displayed with the following options regarding display of distance information in the **3D Window**:

   Note: The aerial, horizontal, and vertical distance information are calculated and displayed in the Distance Measurement **dialog box** even if the respective option was not selected

   ▪ **Show Aerial Distance** – Show a yellow line marking the aerial distance, i.e., the actual distance between points, in the 3D Window.

   ▪ **Show Horizontal Distance** – Show a white line marking the horizontal distance between points in the 3D Window.

   ▪ **Show Vertical Distance** – Show a purple line marking the vertical distance between points in the 3D Window.
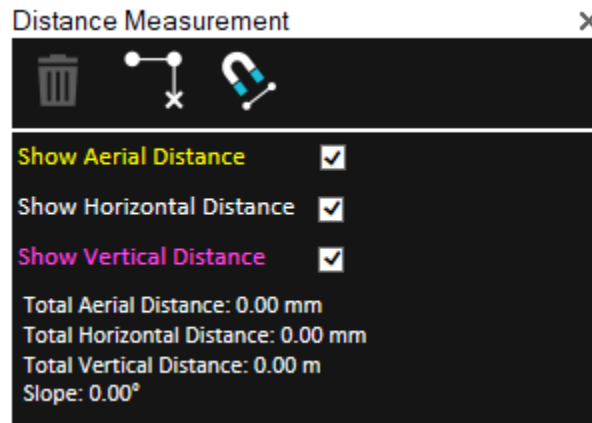
Distance Measurement Dialog Box

2. If you want to snap the measurement points to an object's edge or vertex, click **Snap** [icon]. All edges, i.e., intersections of two plane faces, and vertices, i.e. intersections of two polylines are automatically detected.

3. In the **3D Window**, click any point on the terrain or on an object to define the start point of the measurement. A yellow line extends from the start point. Drag the cursor to the next point and click again. Repeat to add as many segments as required. If you want to delete the last point added, click

   Delete Point [icon]. Right-click to finish the measurement.

The Distance Measurement dialog box displays the following measurements:

Measurement	Description
Total Aerial Distance	The aerial distance, i.e., the actual distance between the points. This measurement is the sum of the aerial distances between the points as you progress from the start point to the endpoint.
Total Horizontal Distance	The horizontal distance between the points.
	This measurement is the sum of the horizontal distances between the points as you progress from the start point to the endpoint.
Total Vertical Distance	The difference in elevation between the points.
	If you have marked more than two points, this measurement does not take into account the elevation values of the middle points.
Slope	The slope of the line between the first and last points.
	If you have marked more than two points, this measurement does not take into account anything other than the elevation at the start point and the elevation at the endpoint.

4. If you want to delete the measurement polyline in the 3D Window, click **Delete** [icon].

5. To perform additional distance measurements, repeat steps 2-3 above.

6. To close this tool, click the X on the top right corner of the dialog box.

Area Tool

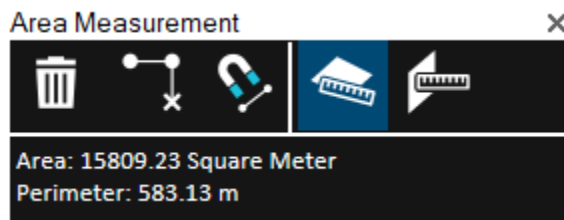Using the Area tool, you can calculate area on the terrain or on an arbitrary plane.

To use the Area tool:

1. On the **Tools** tab, in the **Measure** group, click **Area**. The Area Measurement dialog box is displayed.

Area Measurement Dialog Box

2. Click either of the following commands:

   ▪ **Horizontal plane area (2D measurement)** - Calculate area on terrain. The area measured is the horizontal projection of the area you have outlined, even if some or all of your selected area encompasses mountainous terrain.

   ▪ **3D plane area** - Calculate area on an arbitrary plane. The polygon's orientation is determined by the first three points selected.

3. If you want to snap the measurement points to an object's edge or vertex, click **Snap** . All edges, i.e., intersections of two plane faces, and vertices, i.e. intersections of two polylines are automatically detected.

4. Draw a polygon in the 3D Window that defines the borders of the region being measured:

   ▪ Place the polygon points in the **3D Window** by clicking in the desired locations. A red line extends from the start point. If a valid selection area was drawn, the line turns white after the third click, indicating that you can then right-click and obtain the required measurement. A minimum of three points must be selected. If you want to delete the last point added, click **Delete Point.**

   Note: If the boundary line of the measured area crosses itself, it turns red, and a dialog box is displayed informing that the polygon is invalid.

5. Right-click to finish the measurement. The Area dialog box displays area and perimeter measurements.

6. If you want to delete the area polygon in the 3D Window, click **Delete** .

7. To perform additional area measurements, repeat steps 2-5 above.

8. To close this tool, click the X on the top right corner of the dialog box.

Information Query Tool

The Query tool provides you with data about any point or object in the 3D World. For a point on the terrain, location and elevation information is provided, while for an object, the query also displays the object's perimeter and area. If the object selected was loaded from a layer with attribute information, the object's attributes are also displayed.

To use the information query tool:

1. On the **Tools** tab, in the **Measure** group, click **Query**.

2. Move the cursor over any point in the **3D Window**. When you move the cursor over an object, the object is highlighted.

3. Click the left mouse button on the point or object for which you require information.

   ▪ In the Message Bar, the longitude, latitude and altitude are displayed.

   ▪ The Query Results dialog box is displayed, showing the longitude, latitude, and elevation. When an object is selected, general information about the object and attribute information, if available, is displayed.

4. With the Query tool selected, you can click again on any point in the terrain to find its coordinates. Repeat for as many points as you wish.

On the **Tools** tab, in the **Measure** group, click **Query** again to close the Information Query.

CHAPTER 21 PHOTOMESH AUTOMATION

About PhotoMesh Automation

PhotoMesh provides you with REST APIs for automating PhotoMesh project and queue tasks.

- **PhotoMesh API** - REST API to automate the production phases, e.g., load photo files or list, set AT and reconstruction areas, begin and manage a build, and create new build versions.

- **ProjectQueue API** - REST API to create a project queue, and then activate, monitor, and manage the queue processing.

Creating a Project Queue

Processing of the project queue is activated from the PhotoMesh Queue Builder. After the project queue JSON file is loaded, the relevant fields in the Queue Builder are automatically populated based on the information in the loaded file. These fields can be edited in the Editor and additional projects can be added to the queue. When the queue is ready, you activate queue processing, and then PhotoMesh begins performing the build tasks according to their order in the queue. During the build process, the builds included in the queue are monitored and managed from the Projects Queue Manager. **See** "Managing the Queue" in this chapter for more information

To create a queue:

1. From the **Start** page, open the **Automation Dashboard** and in the **Project Queue** section, click **Project Queue**. The Projects Queue Editor is displayed. **See** "Automation Dashboard" in the "Basic Concepts" chapter for information.

2. Click **Load from File**, and browse to the project queue JSON file you created. **See** "Creating a Project Queue JSON File" in this chapter for information. The file is loaded, and the relevant fields are automatically populated based on the file information. If you want to start with a blank JSON file, and enter all the values manually, click **New projects template**.

3. Enter/modify the required information for your file type:

Field	Description
Name	Free text describing the project.
Action	The action you want to perform in PhotoMesh. The following are the possible options: - Create (and Build) New Project - New Build - New Build – Copy AT - Rebuild - Pause - Script (Python or JavaScript)
Project Path	Type the full path to: - The PhotoMeshXML project file that should be loaded if performing a new build or rebuilding,

	▪ The PhotoMeshXML project file that should be created if creating and building a new project. ▪ A script file if the action selected is script.
Build From	The step from which you want to start the build. Select "Auto" to have PhotoMesh automatically determine what processing must be performed to complete the project, avoiding unnecessary rebuilding.
Build Until	The step until which you want to build.
Inherit from Build	The name of the build version from which to inherit the project's base build parameters. This property is applied only when performing a New Build or – New Build-Copy AT.
Preset	If the preset file is saved under the application's AppData or in the application files' Presets folder, enter the name of the preset files to be loaded by PhotoMesh, separated by a comma. Otherwise enter the full path of each preset, separated by a comma. Click **Select** to select one or more presets from a list, and then click **OK**, or type the full path to the file.The available preset files are in the Presets folder located under the folder from where PhotoMesh is running. Only the file name should be typed, without the file extension, e.g., "Draft_mode" or "Standard_GPS".
Max Pool Fusers	Maximum number of pool fusers to run.
Max AWS Fusers	Maximum number of Amazon Web Services (AWS) fuser instances to run.
AWS Startup Script	The startup script PhotoMesh should use when launching AWS fuser instances. This field is only displayed if Max AWS Fusers was set > 0.
AWS Configuration Name	The name of the configuration JSON file PhotoMesh should use to determine which AWS instances to launch. Only the file name should be passed, without the file extension. This field is only displayed if Max AWS Fusers was set > 0.
AWS Configuration JSON Path	The path to the folder where the AWS Configuration Name JSON file is saved (default is ./PhotoMesh/AWSConfigurations). This field is only displayed if Max AWS Fusers was set > 0.
Collections	The source of the project's photo information. This property is only displayed when Action was set to Create New Project. The following options are available: ▪ Load Folder ▪ Load Video ▪ Load List (XML, Excel)
Collection XXX	This property is only displayed when Action was set to Create New Project: ▪ If Collections was set to Load Folder – Enter the name and full path to each of the project's collections. ▪ If Collections was set to Load Video – Enter the name and full path to each of the project's collections (containing the video

file).

Excel	Full path to the photo list.
Properties	Full path to the collectionProperties.xml file (optional). There are no restrictions on the naming of the collection properties XML file, or on the location where this file is saved. This property is only displayed when Action was set to Create New Project and Collections was set to Load Folder.
Focal35	Equivalent focal length in millimeters for a 35 mm film camera. This property is only displayed when Action was set to Create New Project and Collections was set to Load Video.
Taken Every	Sampling interval, i.e., every how many seconds a frame should be captured. This property is applied only when loading a video. This property is only displayed when Action was set to Create New Project and Collections was set to Load Video.
Seconds Start	Type the start time from which video frame extraction should begin. This property is only displayed when Action was set to Create New Project and Collections was set to Load Video.
End	Type the end time for video frame extraction. This property is only displayed when Action was set to Create New Project and Collections was set to Load Video.

4. Repeat steps 2- 3 for each project you want to add.

5. Click **Remove** to remove a project.

6. Click **Move up** and **Move down** to change the order of the projects in the queue.

7. Click **Add to Queue**.

Managing the Queue

The project queue is monitored and managed from the Projects Queue Manager. The Manager provides dynamic information on the status of the projects in the queue and enables you to start/stop the projects in the loaded queue, and change fuser allocation. **See** "Projects Queue Editor" in this chapter for more information on loading a queue file.

To manage the projects in your queue:

1. From the **Start** page, open the **Automation Dashboard** and in the **Project Queue** section, click **Project Queue**. The Projects Queue Editor is displayed. See "Automation Dashboard" in the "Basic Concepts" chapter for information.

2. Click **Queue Manager**. The Projects Queue Manager is displayed, with the left side displaying information on each of the projects in the queue. Click **Move up** and **Move down** to change the order of the projects in the queue.

3. If you want to remove any of the projects from the queue, click the **Remove** link above the project name. If you want to clear the entire queue, click **Remove All** at the top of the page.

4. Click **Build** to start building the projects in the queue.

5. If you want to abort the current build project, and all other builds in the queue, click **Abort & Stop**.

6. If you want to change the maximum number of AWS or pool fusers for the queue, enter the required number in the appropriate field, and click **Change**.

PhotoMesh API

PhotoMesh provides a REST API for PM Production to automate the production phases, e.g., load photo files or list, set AT and reconstruction areas, begin and manage a build, and create new build versions.

To view interactive documentation of the PhotoMesh REST API:

- From the **Start** page, open the **Automation Dashboard** and in the **PhotoMesh API** section, click **Reference Guide**. **See** "Automation Dashboard" in the "Basic Concepts" chapter for information.

    Note: You can also access it from Resources >PhotoMesh API. **See** "PhotoMesh Resources" in the "Introduction" chapter for information.

Queue API

PhotoMesh provides a REST API to create a project queue and then activate, monitor, and manage queue processing. The project queue is created using the **add** API method, in which a project queue JSON file is passed that lists all the tasks that should be performed for each of the projects in the project queue. **See** "Creating a Project Queue JSON File" in this chapter for more information.

To view technical and interactive documentation of the Queue REST API:

- From the **Start** page, open the **Automation Dashboard** and in the **Project Queue** section, click **Reference Guide**. **See** "Automation Dashboard" in the "Basic Concepts" chapter for information.

To use PhotoMesh Queue API:

- From the **Start** page, open the **Automation Dashboard** and in the **Project Queue** section, click **Project Queue**. **See** "Automation Dashboard" in the "Basic Concepts" chapter for information.

Creating a Project Queue JSON File

The project queue JSON file lists all the tasks that should be performed for each of the projects in the project queue. It also includes photo information and build and fuser settings for each of the queue's projects. The steps below describe the file structure and syntax.

To create a projects queue JSON file:

1. Create a JSON object that holds a "projects" array whose objects are each of the projects to be included in the queue.
2. Create project objects that are composed of name/value pairs separated by commas, which set the build tasks to perform, build and fuser settings, and photo information. The following name/value pairs should be included:

Field	Description
Mandatory Parameters	
action	Determines what action is performed. The following are the possible values: 0 – Create New Project (Create project, load photos and photo information, and then build the project), 1 – New Build, 2 – New Build-Copy AT, 3 – Rebuild, 4 – Pause, 5 – Script (Python or JavaScript).
projectPath	The full path to: - The PhotoMeshXML project file that should be or created if creating and building a new project (action =

251

	0).
	▪ The PhotoMeshXML project file that should be loaded if performing a new build or rebuilding (action = 1, 2, or 3).
	▪ A script file if the step is script (action = 5)
buildFrom	The step from which to start the build. The following are the possible values: 0 – Auto, 1 – Data Preparation, 2 – AT, 3 – Point Cloud, 4 – Model, 5 – Texturing, 6 – Output.
buildUntil	The step until which to perform the build. The following are the possible values: 0 – Auto, 1 – Data Preparation, 2 – AT, 3 – Point Cloud, 4 – Model, 5 – Texturing, 6 – Output.
preset	The name of the preset file to be loaded by PhotoMesh. Click **select** to select a preset or type the full path to the file. The available preset files are in the Presets folder located under the folder from where PhotoMesh is running. Only the file name should be passed, without the file extension, e.g., "Draft_mode" or "Standard_GPS".
Specific Parameters	
sourceType	The source type of the project's photo information. This parameter is applied only when creating a new project (action = 0). The following are the possible values: 0 – Load Folder, 1 – Load Video, 2 – Load List (Excel or XML file).
sourcePath	This parameter is applied only when creating a new project (action = 0):
	▪ If creating a new project by loading a folder of photos (sourceType = 0), the sourcePath array contains the name and full path to each of the project's collections, and an optional "properties" value with the full path to the collectionProperties.xml file. When using the API, there are no restrictions on the naming of the collection properties XML file, or on the location where this file is saved.
	▪ If creating a new project by loading a video (sourceType = 1), the sourcePath array contains the name and full path to each of the project's collections (containing the video file), focal length 35, Sampling interval (frameEvery), and start and end times for video frame extraction.
	▪ If creating a new project by loading an Excel or XML photo list file (sourceType = 2), the sourcePath array contains the full path to the photo list.
inheritBuild	The name of the build version from which to inherit the project's base build parameters. This property is applied only when performing a New Build or – New Build-Copy AT (action = 1 or 2).
Fusers Parameters	

MaxLocalFusers	The maximum number of pool fusers to use for the build.
MaxAWSFusers	The maximum number of Amazon Web Services (AWS) fuser instances to launch for the build.
AWSFuserStartupScript	The startup script PhotoMesh should use when launching AWS fuser instances. This parameter is required if MaxAWSFusers was set > 0.
AWSBuildConfigurationName	The name of the configuration JSON file PhotoMesh should use to determine which AWS instances to launch. Only the file name should be passed, without the file extension. This parameter is required if MaxAWSFusers was set > 0.
AWSBuildConfiguration.JsonPath	The path to the folder where the AWSBuildConfigurationName JSON file is saved (default is ./PhotoMesh/AWSConfigurations). This parameter is required if MaxAWSFusers was set > 0.

Example 1: New Project from Folder

```
{
  "projects": [
{
      "comment": "New project from folder",
      "action": 0,
      "projectPath": "N:\\PM
Projects\\ProjectFolder\\ProjectName.PhotoMeshXML",
      "buildFrom": 1,
      "buildUntil": 6,
      "inheritBuild": "",
      "preset": "PhotoMesh Default",
      "workingFolder": "N:\\WorkingFolder",
      "MaxLocalFusers": 10,
      "MaxAWSFusers": 7,
      "AWSFuserStartupScript"script",
      "AWSBuildConfigurationName": "AT_MultiCoreCPU",
      "AWSBuildConfigurationJsonPath": "",
      "sourceType": 0,
      "sourcePath": [
        {
          "name": "RGB",
          "path": "X:\\RGB",
          "properties": "X:\\RGB.xml"
        },
        {
```

```
        "name": "NIR",
        "path": "X:\\NIR",
        "properties": "X:\\NIR.xml"
      },
      {
        "name": "Thermal",
        "path": "X:\\THERMAL",
        "properties": "X:\\Thermal.xml"
      }
    ]
  }
 ]
}
```

Example 2: New Project from Video

```
{
  "projects": [
{
      "comment": "New project from video",
      "action": 0,
      "projectPath": "N:\\PM
Projects\\ProjectFolder\\ProjectName.PhotoMeshXML",
      "buildFrom": 1,
      "buildUntil": 6,
      "inheritBuild": "",
      "preset": "PhotoMesh Default",
      "workingFolder": "N:\\WorkingFolder",
      "MaxLocalFusers": 10,
      "MaxAWSFusers": 7,
      "AWSFuserStartupScript":  "script",
      "AWSBuildConfigurationName": "AT_MultiCoreCPU",
      "AWSBuildConfigurationJsonPath": "",
      "sourceType": 1,
      "sourcePath": [
        {
          "name": "Collection 1",
          "path": "X:\\PMProjects\\Washington\\PMCollections",
          "focal35": 24,
        "frameEvery": "00:00:05",
          "startTime": "00:00:00",
```

```
            "endTime": "01:00:00"
        }
      ]
    }
  ]
}
```

Example 3: Rebuild Project

```
{
  "projects": [
{
      "comment": "Rebuild project",
      "action": 3,
      "projectPath": "N:\\PM
Projects\\ProjectFolder\\ProjectName.PhotoMeshXML",
      "buildFrom": 1,
      "buildUntil": 6,
      "preset": "PhotoMesh Default",
      "workingFolder": "N:\\WorkingFolder",
      "MaxLocalFusers": 10,
      "MaxAWSFusers": 7,
      "AWSFuserStartupScript":  "script",
      "AWSBuildConfigurationName": "AT_MultiCoreCPU",
      "AWSBuildConfigurationJsonPath": ""     }
  ]
}
```

Troubleshooting

If you cannot access the PhotoMesh, PhotoMesh Fuser, or Project Queue API, this is generally due to a firewall blocking the required port or missing URL reservations. In this case, you can manually add from the command line a rule to allow the required port through the firewall and configure a URL registration. Below are the steps for each of the API's:

- [PhotoMesh API](#)

- [Fuser API](#)

- [Project Queue API](#)

PhotoMesh API

To open the PhotoMesh API port 8086:

1. Click the **Start** ⊞ button, type "cmd", and then select **Command Prompt** from the list.

2. In the command prompt window type the following and click **Enter**:

```
netsh advfirewall firewall add rule name="PM API WebAPI TCP Port 8086"
dir=in action=allow protocol=TCP localport=8086
```

3. Then type the following and click **Enter**:

```
netsh http add urlacl url=http://*:8086/ SDDL=D:(A;;GX;;;WD)
```

The first line adds a firewall rule named "PM API WebAPI TCP Port 8086" allowing it to bypass the firewall, and the second line assigns the rights for the port.

If you want to change the port of the PhotoMesh API, do the following:

1. In a text editor, open the *PhotoMesh.exe.config* file that is located in the application files folder, and replace the value of the "*API.API URL*" key.

```
<appSettings>
   <add key="API.API URL" value="http://XXXX" />
</appSettings></appSettings>
```

2. Open in a text editor the *.env* file that is located in the application files' **PhotoMesh.js** folder, and replace the value of *PHOTOMESH_EXE_SERVER_PORT*. E.g.,

```
PHOTOMESH_EXE_SERVER_PORT=XXXX
```

PhotoMesh Fuser API

To open the PhotoMesh Fuser API port 8085:

1. Click the **Start** button, type "cmd", and then select **Command Prompt** from the list.
2. In the command prompt window, type the following and click **Enter**:

```
netsh advfirewall firewall add rule name="PM Fuser API WebAPI TCP
Port 8085" dir=in action=allow protocol=TCP localport=8085
```

3. Then type the following and click **Enter**:

```
netsh http add urlacl url=http://*:8085/ SDDL=D:(A;;GX;;;WD)
```

The first line adds a firewall rule named "PM Fuser API WebAPI TCP Port 8085" allowing it to bypass the firewall, and the second line assigns the rights for the port.

If you want to change the port of the Fuser API, do the following:

▪ In a text editor, open the *PMFuserService.exe.config* file located in the application files' **Fuser** folder, and replace the value of *webApiUrl*. E.g.,

```
<add key="WebApiUrl" value="http://*:XXXX/" />
```


Project Queue API

To open the Project Queue API fuser port 8087:

1. Click the **Start** button, type "cmd", and then select **Command Prompt** from the list.
2. In the command prompt window, type the following and click **Enter**:

```
netsh advfirewall firewall add rule name="PM ProjectQueue WebAPI TCP
Port 8087" dir=in action=allow protocol=TCP localport=8087
```

3. Then type the following and click **Enter**:

```
netsh http add urlacl url=http://*:8087/ SDDL=D:(A;;GX;;;WD)
```

The first line adds a firewall rule named "PM ProjectQueue WebAPI TCP Port 8087" allowing it to bypass the firewall, and the second line assigns the rights for the port.

If you want to change the port of the Project Queue API, do the following:

▪ In a text editor, open the *.env* file located in the application files' **PhotoMesh.js** folder, and replace the *port* number.
E.g., PORT=8087 > PORT=XXXX