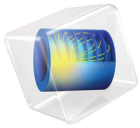


ECAD Import Module

User's Guide

6.4

COMSOL

ECAD Import Module User's Guide

Version: COMSOL 6.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: www.comsol.com/support/learning-center

Part number: CM023601

Contents

Chapter 1: Introduction

Chapter 2: Importing and Exporting ECAD Files

Chapter 3: Programming and Command Reference

1

Introduction

Welcome to the ECAD Import Module. This *User's Guide* details features and functionality available with the product.

This introductory chapter is About This Guide.

About This Guide

This documentation covers the ECAD Import Module.

About the ECAD Import Module

This product extends the functionality of COMSOL Multiphysics by enabling modeling using imported ECAD design files.

The import capabilities cover the *GDS II*, *ODB++®*, and IPC-2581 file formats and include the creation of 3D geometry during the import operation. The export capabilities cover OASIS file format.

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

	If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to other modules (as long as you have a license), application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes all core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to many of the features on the GUI. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information

about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context*) *help*.

Win	To open the **Help** window: • In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1. • On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1. • From the **File** menu, click **Help** (?). • In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac Linux	To open the **Help** window: • In the **Model Builder** or **Physics Builder** click a node or window and then press F1. • In the main toolbar, click the **Help** (?) button. • From the main menu, select **Help>Help**.

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu select **Help>Documentation** ( ).

Mac Linux	To open the **Documentation** window: • Press Ctrl+F1. • In the main toolbar, click the **Documentation** () button. • From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each application includes documentation with the theoretical background and step-by-step instructions to create a model application. The applications are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual applications as a template for your own modeling and applications. In most models, SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the application and its properties, including options to open it or a PDF document.

	The Application Libraries Window in the *COMSOL Multiphysics Reference Manual.*

Opening the Application Libraries Window

To open the **Application Libraries** window ():

Win	• In the **Home** toolbar, click **Windows** and select **Application Libraries**. When the toolbar is compressed, you sometimes find it under **Layout>Windows**. • From the **File** menu select **Application Libraries**. To include the latest versions of model examples, from the **File>Help** menu select () **Update COMSOL Application Libraries**.

Mac Linux	From the **File** or **Windows** menu select **Application Libraries**. To include the latest versions of model examples, from the **Help** menu select () **Update COMSOL Application Libraries**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and case number is sent to you by email.

COMSOL WEBSITES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/videos
Support Knowledge Base	www.comsol.com/support/knowledgebase

2

Importing and Exporting ECAD Files

ECAD file formats have been designed to transfer information necessary for the manufacturing of printed circuit boards (PCBs), integrated circuits (ICs), and microelectromechanical systems (MEMS) devices. You can import data from ECAD files to generate geometry for simulation. During the import of an ECAD file, geometry objects are generated based on the 2D layouts and stackup information found in the files.

In this section:

Importing PCB File Formats

In this section:

Import Settings for IPC-2581 and ODB++ Files

To add an **Import** node, from the **Home** or **Geometry** toolbar, click **Import** ( ). In the **Source** section of the **Settings** window, set the type of file to import to **ECAD file**. You can also skip this step as the type of the selected file is automatically recognized. Click **Browse** to locate the file to import, or enter the path to the file. Before clicking the **Import** button consider to review and configure the import options, especially the layer thickness information, since in many cases the IPC-2581 files and ODB++ archives do not include all necessary information to construct a 3D geometry. If the content of the file have been changed which would affect the listed layers, nets or components in the settings, you can click **Refresh Data from File** ( ) button to get the new contents.

For PCB files, you can click the **Preview** ( ) button in the **Settings** window toolbar to show the preview of the file content in the **Graphics** window. In the **Preview** settings you can also control which layers and nets to add to import (Controlling The Layer Stackup).

The length unit in the file is detected and displayed in the **Settings** window. To use the unit in the file as the length unit for the geometry sequence select the **Update geometry unit** checkbox. The checkbox is selected by default if the **Import** node is added as the first node in the geometry sequence. The length unit of the geometry sequence is set to `inch` when the length unit `mil` is detected in the selected file.

LAYERS

Grouping of Geometries

To determine the extrusion method used for generating the 3D geometry from the 2D layouts select one of the available alternatives from the **Grouping of geometries** list in the Layers section of the **Settings** window:

- **All**. This alternative is available only in 3D. The import algorithm extrudes in one operation all imported copper and dielectric layers into a single 3D geometry object. From each copper layer, the imported symbols, such as lines, pads, and surfaces, are

combined before extrusion. The copper and dielectric layers become domains in the resulting object. To be able to connect the layers, the algorithm requires that the 2D layouts fulfill certain rules. If the import fails, switch to the **By layer** grouping option.

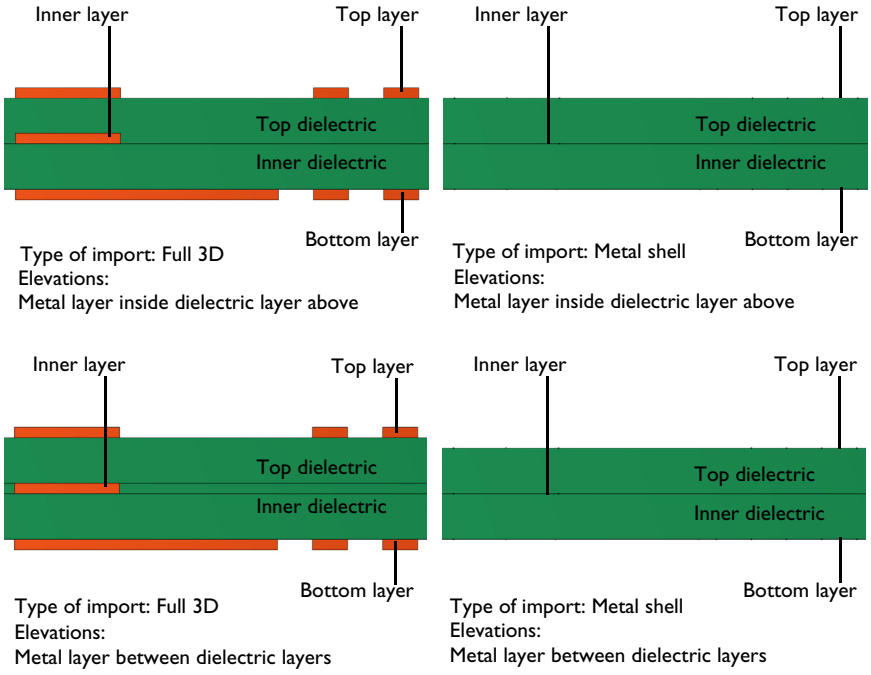
- **By layer**. Use this alternative to extrude each layer separately. The imported symbols, such as lines, pads, or surfaces, from the copper layers are combined before extrusion. Several geometry objects, one for each imported copper and dielectric layer, are generated upon import.
- **No grouping**. Use this alternative to import and extrude individually each symbol from the copper layers. The import results in separate objects for imported symbols and dielectric layers. You can use this option if you want to manually delete certain objects after import, but it is recommended to do this only for relatively simple geometries.

When using the **No grouping** alternative you have the option to import line symbols as curve objects by selecting the **Ignore line width** checkbox.

Note: Using the ignore line width option you can sometimes be useful to construct geometry objects from the information on other layers than copper and dielectric. To do this configure an Import node to import the layer using the **No grouping** method, and with the **Ignore line width** checkbox selected. Also, from the **Elevations** list select **Manual**, and set the elevation to an appropriate value, but keep the default zero layer thickness. After the import use geometry operations to convert the imported curve objects to surface objects, and extrude those to 3D components.

Copper Layers

The **Type of import** list specifies whether copper layers are extruded or not by the import.



The **Full 3D** option imports all copper layers with a thickness according to the table in the **Layers** section of the **Settings** window. The layer elevations are determined according to **Elevations**, see in the next section Controlling The Layer Stackup. With the **Metal shell** option copper layers are imported as embedded boundaries between dielectric regions. As illustrated in the figure above, switching to **Metal shell** changes

the elevation of dielectric layers when **Elevations** is set to **Metal layer between dielectric layers**.

For 3D imports, when the geometry grouping **All** and the import type **Metal shell** are used together, an isolated copper layer cannot be imported if the import also includes another solid (dielectric) layer. In this case use the **By layer** grouping, or add two Import features to the geometry sequence, one to import the copper layer, and a second one for the solid layers.

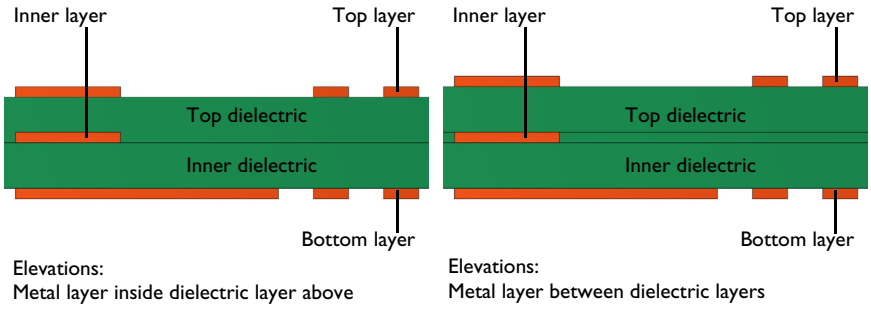
Controlling The Layer Stackup

The import can read stackup information from the PCB file, such as the thickness for copper and dielectric layers. The layer information is displayed in the table in the **Layers** section of the **Settings** window. Sometimes the layer thickness is not included in the export from the ECAD program, so the layers get the default zero thickness. Therefore it is recommended that these values are checked before importing.

To decide how elevations are calculated for the imported layers, select one of the options for **Elevations**:

- Select **Metal layer inside dielectric layer above** to position interior copper layers inside the dielectric layer immediately above the copper layer. The layer elevations in the table are calculated automatically from the layer **Thickness** values. Using this option together with **Type of Import** set to **Full 3D**, the total thickness of the board is calculated as the sum of the dielectric layer thicknesses, plus the thickness of the top and bottom copper layers.
- Select **Metal layer between dielectric layers** to position interior copper layers between the dielectric layers immediately below and above the copper layer. The layer elevations in the table are calculated automatically from the layer **Thickness** values. Using this option together with **Type of Import** set to **Full 3D**, all copper layers, including the interior copper layers, contribute to the total thickness of the circuit board, which is the sum of the thicknesses of all copper and dielectric layers. Extra dielectric layers are created for each copper layer at their respective z-extents and with the same shape as other dielectric layers to pad the gap between the main dielectrics. These are called in-layer dielectric.
- Select **Manual** to enable the manual repositioning of layers in the stackup. The **Elevation** column appears in the table where you can enter values for the lower z positions of the layers. Any changes to the **Elevation** column in the table are lost

when you switch to one of the other two options for **Elevations**. The **Manual** option is available only when **Grouping of geometries** is set to **By layer** or **No grouping**.



When **Grouping of geometries** is set to **By layer** with the **Metal layer between dielectric layers** option for calculating elevations, the checkbox **Unite metal layer and in-layer dielectric** appears and is selected by default. If checkbox is selected, unites each copper layer with their respective in-layer dielectric. If checkbox is cleared, you get the copper and in-layer dielectric as separate objects.

Select the **Show names from file** checkbox to display the layer names as included in the PCB file. This is useful after assigning new layer names that are used in the COMSOL application.

The table contains the following columns:

- The **Name** column displays the layer names from the file. You can edit the entries to give more descriptive names to the layers. The names that appear here are used to name the resulting geometry objects and the selections when the **Layer selections** checkbox is selected under **Selections of Resulting Entities**.
- The **Name in file** column is visible only when you select the **Show names from file** checkbox above the table. The **Name in file** column always displays the layer name from the PCB file, even after editing the **Name** column.
- The **Type** column declares the type of layers. Depending on their type, the import treats layers differently during import. For example, the geometry from a layer of type **Metal** is extruded only if the **Type of import** is set to **Full 3D**. Layers of type **Dielectric** are extruded, when selected in the **Import** column, according to the thickness and elevation data in the table. A union of the objects from the **Outline** layer is used as the PCB outline to determine the extent of the dielectric layers. The **Drill** layer type means that the objects in the layer define drilled holes in the PCB. ODB++ files may sometimes contain copper layers with the context set to MISC.

These are displayed as layers of type **Metal misc** in the table, and they are by default not selected for import, similarly to layers of type **Other**.

- The values in the **Thickness** column are used as the extrusion distances for the layers. The layer elevations are calculated based on these values unless the **Elevation** column is displayed. The values in the layer **Thickness** column can always be changed prior to import.
- The values in the **Elevation** column control the lower *z* positions of the layers. By adjusting the values you can adjust the position of layers in the stackup. The **Elevation** column is only displayed when **Elevations** is set to **Manual**.
- The **Import** column. Clear the checkbox for layers that do not need to be imported. You can also control which layers are imported by using the Preview.

To save the layers table to a text file, click the **Save to File** ( ) button under the table. To load layer table information from a text file, click the **Load from File** ( ) button under the table. Only the editable data in the table is affected when loading from file. Layer data in the file that does not match the existing layers in the table is ignored. To reset the thickness and elevation values in the table to the last read data from the PCB file click the **Reset Thickness and Elevation** ( ) button. To select the **Import** checkbox in the table for all copper and dielectric layers click the **Select All Metal and Dielectric Layers for Import** ( ) button. Click the **Clear All Imports** ( ) button to clear all checkboxes in the **Import** column.

Dielectric Layers

When the PCB file includes the outline of the PCB board in the file the dielectric regions are generated to the shape of the board. Even if the dielectric layers are selected for import in the **Layers** table, the import and extrusion of the corresponding objects or domains can be turned off by clearing the **Import dielectric regions** checkbox.

NETS TO IMPORT

When importing PCBs, the available nets from the file are displayed in a list where you can choose which nets to import. When you click **Import** ( ), under the source section, only the nets that are selected in the **Nets to import** tree will be imported. For PCB files that contain a large number of nets it may help to filter the nets list to display only nets that you have selected to import by selecting the **Only list nets to import** checkbox. You can also enter any text in the **Filter** text box to only display the net nodes that match the filter. To easily select or clear all nets for import, click the **Select All** ( ) button to select all checkboxes and **Clear All** ( ) button to clear all checkboxes that are currently displayed in the list. A specific part of the PCB will only be imported if

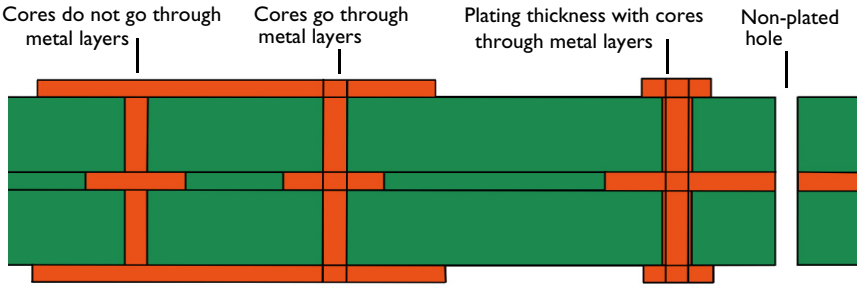
both it's layer and net are selected to be imported. You can also control which nets are imported by using the Preview.

DRILL HOLES

Most printed circuit boards have three types of drilling holes: Plated Through Holes, Non-Plated Through Holes, and Vias. Choose from the **Drill layer handling** list (available only when the **Grouping of geometries** is **By Layer**) how to handle the drill holes specified in the file:

- **Modify layer objects** (default) – This option unites/subtracts the drill layers with/from the layers it intersects. When this is selected, the following checkboxes appear:
  - **Create domains for the cores of vias** (selected by default). When this is selected, the cores of the vias become domains. When this is cleared, the cores of the vias become void.
  - **Via cores through metal layers** (cleared by default). When this is selected, the vias will go through the metal layers. When this is cleared, the vias will not affect the metal layers.
  - **Create domains for the cores of plated holes** (selected by default). When this is selected, the cores of the plated holes will become domains. When this is cleared, the cores of the plated holes will become void.
  - **Plated hole cores through metal layers** (selected by default). When this is selected, the plated holes will go through the metal layers. When this is cleared, the plated holes will not affect the metal layers.
  - **Create domains for non-plated holes** (cleared by default). When this is selected, the non-plated holes will become domains. When this is cleared, the non-plated holes will become void. Note that non-plated holes always go through all the layers.

- **Create separate objects** — This option creates one object for each metal and dielectric layer the drill layers intersects. When this is selected, the following checkboxes appear:
  - **Via cores through metal layers** (cleared by default). When cleared, you get no objects for intersections of vias with metal layers.
  - **Plated hole cores through metal layers** (selected by default). When cleared, you get no objects for intersections of plated holes with metal layers.



Specify the plating thickness of the plated holes and vias in the field **Plating thickness** (default 0) in 3D. When you set a positive thickness value, each drill hole will be surrounded by a cylindrical layer with the given thickness, except for non-plated drill holes. The plating will always be united with or become a part of the drill layer object. You will get no extra object for the plating if you choose **Create separate objects**. Note that the plating thickness is also available if **Grouping of geometries** is set to **No grouping**, in which case the platings become separate objects from their respective cores.

PADS

To more easily apply chosen physics at component-pad connections, you have the possibility to create separated pads from the trace and to identify them by component and pin number.

When grouping of geometries is set to **By layer** or **No grouping**, it is possible to choose to create only selected pads by selecting **From table** (selected by default) option or **All**, which creates all available pads. If **From table** is chosen, a **Components to add** list is shown, with all available components. Select components that you want to create pads for and click the **Add** (+) button to add those components down to the table. In row in the table, in the **Component** column, you specify one or more components for which the same pin numbers should be imported. It also supports entries such as `R*` to

consider all components whose names start with `R`. The **Pins** column lists the pin numbers from which pads should be created, for the chosen components in the same row. It is also possible to write `*` to mean all pins for those components. Values in both columns should be separated with a comma or space. Use **Move Up** (↑), **Move Down** (↓) and **Delete** ( ) buttons to organize the table. Click the **Clear Table** ( ) button to clear all added pads from the table. To load pad table information from a text file, click the **Load from File** button ( ) under the table. To save the pads table to a text file, click the **Save to File** button ( ) under the table.

The pads will be assigned an attribute, where the value is `<Component name>/<Pin number>` and has the fields component, pin, net, package, and value. Read more about attributes in Geometry Attributes. The attribute selects the pad on domain level or if doing a 3D import and the layer has thickness 0, on boundary level. The attribute tag is controlled in the text field **Pad attribute tag**.

You can use a Logical Expression Selection (Geometry Sequences) or a Logical Expression feature to create a selection that selects the domains or boundaries of some pads, for example, using the expression `imp1.pad('R1/1') || imp1.pad.component('C2','C5','C7') || imp1.pad.package('SIP6') || imp1.pad.pin('2') || imp1.pad.net('GND')` in the Logical Expression field. See also Geometry Attributes.

COMPONENTS

Simplified 2D or 3D geometries can be generated for components by selecting the checkbox **Import component outlines** (cleared by default).

In a 3D geometry, you can specify whether the component should be represented by the extruded outline or by a surface as a 2D outline. From the **Height of components** list, select **From file** (default) to extrude the 2D component outline to the height specified in the file. Select **Zero** to import the component outline represented by a face.

Component outlines are positioned by default on the top copper layer or below the bottom copper layer. You can specify an offset for the vertical component placement in the **Elevation offset for top components** field for components on the top copper layer and the **Elevation offset for bottom components** field for the components on the bottom copper layer.

The components will be assigned an attribute, where the values are the component names and are assigned to their respective component's domain or surface (depending on if the component is 2D or 3D). This attribute also has the field "package" whose value is the package for each component. The attribute tag is controlled in the text field **Component attribute tag**. If there are multiple import features, their component attributes will be renamed after their respective import features, for example, `imp1.component` and `imp2.component`. If desired, these features could contribute to a single attribute by changing both attribute tags to have the same name, for example, `component`.

You can use a Logical Expression Selection (Geometry Sequences) feature to create a selection that selects the domains or boundaries of some components, for example, using the expression `imp1.component('C2','C5','C7') || imp1.component.package('SIP9')` in the Logical Expression field.

SIMPLIFY AND REPAIR

Interior Boundaries

By default the **Keep interior boundaries** checkbox is cleared to instruct the import to remove all interior boundaries on the imported layers. Interior boundaries on the copper layers are the result of the union of the individual symbols that make up the traces. Ignoring the interior boundaries keeps the geometry complexity to a minimum and can also make the import more robust in some situations.

Copper Text and Objects Outside the Board Boundary

It is common that the copper layouts have text objects that provide information, and are not part of any electrical circuits. Shapes may be also included for example to mark the board boundaries, and these shapes are often positioned outside the board. Such objects may make it more difficult to construct the geometry and are usually of no interest in a physical simulation.

Select the **Ignore text objects** checkbox to skip all objects in a PCB file that have the `TEXT` tag set. To prevent objects outside the board from being imported select the checkbox **Ignore objects outside of board**. Objects that are positioned outside the board, at a distance greater than that specified by the offset in the **Offset from boundary** field, are ignored by the import. The default value for the offset is –0.1 mm, which means that objects that are further into the board than 0.1 mm from the board boundary are imported. The offset should only be taken as an approximation. It is only

checked where the object has its defining points, which are not necessarily the same points that can be seen on an object after it has been imported. For example, a trace is defined by its start and end points, which lie on the centerline of the trace. Similarly, a circle might be defined by four points (south, north, east, and west). Thus, an object might be ignored even if an edge is further inside the board than the specified offset, as long as all the points that define the object are farther away from the boundary than the offset.

Repair

Geometry repair is controlled via the **Repair imported objects** checkbox and the **Repair tolerance** list. Change the **Repair tolerance** from **Automatic** to **Relative** to manually specify a tolerance in the **Relative repair tolerance** field. **Automatic** means a relative repair tolerance of 10^{-6}.

When selected, the geometry repair is active when combining the objects to generate the 2D geometry of each layer, and also when the layers are combined together after extrusion. By increasing the repair tolerance, problems with short edges arising when combining the objects can sometimes be circumvented. A repair tolerance that is too high may however lead to breaking the geometry. As a guideline, the relative tolerance should be kept between 10^{-5} to 10^{-8}. Values at the lower end of this interval may result in faster import at the cost of a higher number of short edges being present in the imported geometry. Read about how to remove short edges in the next section.

Ignore Vertices in Layers

Select the **Ignore vertices with continuous tangent** checkbox (cleared by default), to remove, in each layer, vertices that connect two edges that have tangent directions making an angle less than the **Continuous tangent tolerance**.

Select the **Eliminate short edges** checkbox (cleared by default) to eliminate edges that are shorter than the **Maximum edge length**, by ignoring one or both of their adjacent vertices.

SELECTIONS OF RESULTING ENTITIES

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is

Domain selection, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Layer selections** checkbox to create predefined selections — for domains, boundaries, and objects — in subsequent nodes in the geometry sequence for each imported layer in the PCB file. The boundary selection for all layers include all boundaries of the objects.

Select the **Net selections per layer** checkbox to create domain and boundary net selections for each layer separately. When the checkbox is cleared, only net selections are generated and the intersection selections between net selections and all other selections generated from the import are not created during the import.

To make the resulting entities that the layers consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, or **Boundary selection**. The default is **All levels**, which makes predefined selections available on all applicable levels, suitable for use with materials and physics defined in domains and boundaries. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

When the **Layer selections** checkbox is enabled, one selection will be created for each layer and for each geometric entity level, object, domain and boundary, for example: `SIG1, DIEL_1, GND, DIEL_2, SIG2`.

For each `DRILL` layer, the following selections are available:

- If the **Plating thickness** is zero:
  - When **Create domains for the cores of vias** or **Create Separate objects** is selected: A domain and a boundary selection for the vias, for example, `DRILL.VIA`. Note that boundary selection includes all the boundaries of the vias.
  - When **Create domains for the cores of plated holes** or **Create separate objects** is selected: A domain and a boundary selection for the plated holes, for example, `DRILL.PLATED`. Note that boundary selection includes all the boundaries of the plated holes.
  - When **Create domains for non-plated holes** or **Create separate objects** is selected: A domain and a boundary selection for the non-plated holes, for example,

DRILL.NONPLATED. Note that boundary selection includes all the boundaries of the non-plated holes.

- If the **Plating thickness** is positive:
  - When **Create domains for the cores of vias** or **Create Separate objects** is selected: A domain and a boundary selection for the vias, for example DRILL.VIA.CORE. Domain and boundary selections for the plating thickness of the vias, for example, DRILL.VIA.PLATING. Note that boundary selection includes all the boundaries of the vias.
  - When **Create domains for the cores of plated holes** or **Create separate objects** is selected: A domain and a boundary selection for the plated holes, for example, DRILL.PLATED.CORE. Domain and boundary selections for the plating thickness of the plated holes, for example DRILL.PLATED.PLATING. Note that boundary selection includes all the boundaries of the plated holes.
  - When **Create domains for non-plated holes** or **Create separate objects** is selected: A domain and a boundary selection for the non-plated holes, for example, DRILL.NONPLATED. Note that boundary selection includes all the boundaries of the non-plated holes.

When the checkbox **Unite metal layer and in-layer dielectric** is selected, the selections for all geometric levels are available for metal, for example INNER (which selects the entities corresponding to the metal). For the domain and boundary levels, in-layer dielectric selections are available, for example, INNER.ILDIEL (which selects entities corresponding to the dielectric). Note that the object selection is not available for in-layer dielectric. All boundary selections include all exterior boundaries of the layer.

When the checkbox **Unite metal layer and in-layer dielectric** is cleared, the selections for all geometric levels are available for both metal and dielectric, for example, INNER (which selects the entities corresponding to the metal) and INNER.ILDIEL (which selects entities corresponding to the dielectric).

If there are any pads selected for import, the Pads selection is created which selects the pads on domain level. If doing a 3D import and type of import is **Metal shell** or the layer has zero thickness, the selection is created on a boundary level.

When the checkbox **Import component outlines** is selected, two selections are created for top and bottom components for all geometric levels (object, domain, and boundary), for example, COMP_BOT and COMP_TOP, referring to the bottom and top components, respectively.

Additionally, the import creates domain selection **Metal**, which selects all the metal layers. If there is no drill hole plating, the drill hole cores are also included, or if plating exists, it instead includes the plating domains. It does not include non-plated holes. The domain selection for **Dielectric** selects everything else except the components. The boundary selection **Metal** is always created and all exterior boundaries of the metal domains are selected.

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

NET SELECTIONS

The sections **Domain Net Selections** (in 2D and 3D) and **Boundary Net Selections** (in 3D), contain a table that lists the generated selections for the imported nets. Net selections are generated when importing IPC-2581 and ODB++ files. The tables contain the following columns, ordered from left to right:

- **Name** — This column contains the name of the selection.
- **Keep** — The checkbox in this column is cleared by default, and determines whether the selection will be available for use in geometry features following the **Import** feature.
- **Physics** — The checkbox in this column is selected by default. Provided that also the **Keep** checkbox has been selected, it determines whether the selection will be available in all applicable selection lists in physics and materials settings, for example. If the Import node has been added under the Plane Geometry node of a Work Plane, this column header is titled **3D**. If the Import node has been added to a geometry part, this column header is titled **Instances**.
- **Contribute to** — Select a cumulative selection from the list in this column to make the resulting entities contribute to a cumulative selection. The default, **None**, gives no contribution. Contributing to a cumulative selection does not require that the **Keep** checkbox has been selected.

Click the **New Cumulative Selection** button under the tables to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

The entities in a selection are highlighted in yellow in the **Graphics** window when the selection is clicked in the table. To zoom in on the selection use the **Zoom to Selection** button in the **Graphics** toolbar.

When the checkbox **Net selections per layer** is selected under the **Selection of Resulting Entities** section, additional selection are created, one for each intersection between the net selections and all other selections generated from the import. This checkbox can be selected after the import has been done. For example, for the default behavior of the drill holes handling, importing a board with three copper layers (`SIG1`, `GND`, `SIG2`), two dielectric layers (`SIG1_DIEL`, `GND_DIEL`), and one drill layer (`DRILL1`), that contains a net named `NET`, the following selections are generated: `NET, NET.SIG1`, `NET.GND`, `NET.SIG2`, `NET.SIG1_DIEL, NET.GND_DIEL, NET.DRILL1_PLATED, NET.DRILL1_VIA, NET.Metal`. Empty net selections are not generated; that is, a selection for the net is generated only when the intersection of the net and the layer contains entities. For entities that do not belong to a net, the selection `NONET` is generated.

The selection `NET` is the union of all other selections for this net. If the name of a net is the same as the name of a layer, for example `GND`, the suffix `_N` is attached to the name of the net selection to avoid a name clash between net and layer selections. If pads have been selected for import, the net selections will also include `NET.Pads` selection.

The boundary net selections consists of all the exterior boundaries of the domain selections. Note that edge and vertex selections are not generated for nets.

To view the list of all objects, you can use the **Selection List** window, that you can access from the Geometry toolbar. The top part of the Selection List window lists all geometry objects in the geometry sequence. The bottom part of the window lists all named selections, including the kept net selections.

PREVIEW

For PCB files, you can click the **Preview** ( ) button in the **Settings** window toolbar to show the preview of the file content in the graphics. When clicked, the view is switched to preview mode, where the graphics switches to a 2D scene, and a window **Preview** appears on the right side of the graphics. The preview rendering displays the metal and drill layers and consists of one object for each combination of layer and net, as well as the board outline as a curve object. The rendering shows the 2D geometries without any edges, and they are colored according to the layer they belong to.

The **Preview** settings window includes a section **Layers to Import**, with a table listing the layers that are either metal layers or drill layers. The first column ( ) controls the import of the layer while the second column ( ) controls if the layer is displayed in

the graphics. The third and fourth columns show the color and the name of each layer, respectively. When changing the value of the first column, an import checkbox (☑), it automatically updates the corresponding checkbox in the **Layers** table in the main import settings. This also affects the dielectric layer below, if such exists. The toolbar below the table contains shortcuts for easier manipulation of the table, like **Import All** ( ), **Import None** ( ), **Show All** ( ), and **Hide All** ( ) buttons. Right-clicking a row in a table brings up a context menu with actions that act on the selected rows. The actions are: **Add to Import**, **Remove from Import**, **Hide**, **Show**, and **Show Only This**, and they control the import of show checkboxes of the selected rows. Selecting a row in layers table marks the corresponding graphics objects in red. Additionally, there is a checkbox **Show board outline** located below the table, which controls if the board's outline is shown in the graphics.

The **Nets to Import** section contains a table equivalent to that in the **Layers to Import** section, but acting on nets instead of layers. This table is also filterable, controlled with **Only list nets to import** and **Only list visible nets** checkboxes and a **Filter** text field. The toolbar actions below the table only act on the rows currently displayed in the table. Selecting a row in the table will mark the corresponding graphics objects blue. At the top of the section is the checkbox **Select which nets to import in graphics**. When this is selected you enter a mode where the objects that are selected in the graphics correspond to the objects that will be imported, that is, an object is selected if both its layer and net will be imported. In this mode, clicking in the graphics directly affects the **Nets to Import** checkboxes, but only if the clicked object has its layer selected for import. Selecting a row in the table when in this mode marks the corresponding objects red, and those objects that are blue become green.

Hovering the mouse over an object will highlight it in red, but selecting it will select all objects belonging to the same net. The graphics context menu always contains the actions **Select Box** ( ), **Zoom Box** ( ), **Zoom Extents** ( ), **Copy Image** ( )and **Help** (?). And if some objects are selected and the checkbox **Select which nets to import in graphics** is cleared, it also contains the actions **Add to Import**, **Remove from Import**, **Hide**, **Show Only This**, as well as **Zoom to Selection**. The preview specific actions will act on the settings for the nets that the selected objects belong to.

File Formats for Printed Circuit Boards (PCBs)

The IPC-2581 and OBD++ file formats can handle most of the information needed to manufacture a PCB. For generating a geometry for simulation, the import

functionality can read some of this information, such as the layout of copper layers, the layout for the vias, the board outline, and layer stackup information.

	Importing PCBs - Generating Shell Traces
	Application Library path: **ECAD_Import_Module/Tutorials/pcb_import_shell**
	Importing PCBs - Creating Component Domains
	Application Library path: **ECAD_Import_Module/Tutorials/pcb_import_components**
	Importing PCBs - Working with Nets
	Application Library path: **ECAD_Import_Module/Tutorials/pcb_import_nets**

An ECAD file for PCBs may also include layers with 2D layouts that specify, for example, the component outlines on the board. You can also import these layers. In a file, the 2D layouts consist of shapes, also called symbols, which build the geometry of copper traces and pads. A large number of symbols are specified by the supported PCB formats — lines, circles, rectangles, and surfaces to name a few.

Preparing PCB Geometries for Simulation

As a first step, it is always beneficial to remove all features from the geometry that are not important for the simulation. This is usually best done in the ECAD software before importing the geometry into COMSOL. Most ECAD or EDA programs support design rule checks (DRC), which test the entire layout and check that all features (vias, conductors, and components) are separated according to certain rules. With such checks, the layout is free from overlapping vias and conductors touching other conductors or vias. This also ensures that the special extrude functionality of the ECAD import works appropriately. If the file contains such design-rule violations, the extrude operation might fail and issue an error message stating that it could not handle the topology of the layout.

The best approach to handle such problems is to perform a DRC with your ECAD software and produce new layout files. If this is not possible, import the layout in 2D and try to identify the problematic features.

When importing to 3D, the import code can extrude the 2D layouts created from the imported data. Alternatively, you can let the import generate surface objects or import

the layers onto a work plane in 3D. This can be useful if you want to modify the imported 2D layouts before extruding them.

Before meshing the geometry created from imported ECAD files, it can be helpful to eliminate short edges during the import. Do this by selecting the checkboxes **Ignore vertices with continuous tangent** and **Eliminate short edges** in the import settings. Note that you can also remove short edges from the geometry by using the automatic Geometry Cleanup or by manually adding a Remove Details feature; see Remove Details and Geometry Cleanup in the *COMSOL Multiphysics Reference Manual*.

The imported geometry often consists of objects with very high aspect ratios, which are hard to mesh with a tetrahedral mesh generator. As a result, it is usually necessary to use interactive meshing of the imported geometry in a by-layer fashion. Assume that the top and bottom layers are metal layers. All metal layers can often be meshed using swept meshing, but dielectric layers usually cannot be meshed that way because the source and target boundaries do not look the same. Begin by meshing from the bottom or top layer, starting with a triangular or quadrilateral boundary mesh. Then, mesh layer by layer, where each metal layer gets a swept mesh, and each dielectric layer (with vias) gets an unstructured tetrahedral mesh. When starting with a quadrilateral boundary mesh, the swept mesher generates hex elements, and pyramid elements are automatically inserted when the tetrahedral mesher generates the mesh for the dielectric layers. Finally, if there is a surrounding air domain mesh it using the tetrahedral mesher.

Another possibility is to create the geometry for meshing and physics by forming the assembly of the metal and dielectric objects generated by the import. This can be controlled by the **Form Union/Assembly** node in the Geometry branch of the model tree. When forming an assembly, identity pairs connect the touching faces between the layers. To generate the mesh for the individual layers, use the swept mesher. By this method it can be easier to resolve details on the copper layers while limiting the number of element layers in the extrusion direction.

	Importing PCBs - Generating Shell Traces Application Library path: **ECAD_Import_Module/Tutorials/pcb_import_shell**
	*Importing PCBs - Creating Component Domains* Application Library path: **ECAD_Import_Module/Tutorials/pcb_import_components**
	Importing PCBs - Working with Nets Application Library path: **ECAD_Import_Module/Tutorials/pcb_import_nets**

Importing and Exporting MEMS and IC File Formats

In this section:

Import Settings for GDS-II Files

To add an **Import** node, from the **Home** or **Geometry** toolbar, click **Import** ( ). In the **Import** section of the **Settings** window, set the type of file to import to **ECAD file**. You can also skip this step as the type of the selected file is automatically recognized by the code. Click **Browse** to locate the file to import, or enter the path to the file. Before clicking the **Import** button configure the layer thickness, since the GDS-II files do not include the necessary layer thickness and elevation information to construct a 3D geometry. Click the **Refresh Data from File** button ( ) to reload the changes that were made in the PCB file, while keeping the chosen import options in the settings window.

For the length unit in the file µm is always displayed, since GDS-II files do not specify a unit. To use the displayed unit as the length unit for the geometry sequence select the **Update geometry unit** checkbox. The checkbox is selected by default if the **Import** node is added as the first node in the geometry sequence.

CELL SELECTION

Data contained in GDS-II files is organized in hierarchical structure consisting of cells. Select a *cell* name from the **Cell to import** list to limit the import to the selected cell beneath the top cell in the hierarchy. Use the default choice in this list to import the top cell. By default all subcells below the selected cell are imported. To import only one subcell type, select the cell from the **Filter by subcell** list. The difference between selecting a cell and filtering the top cell by the same cell is that the latter alternative includes all instances of the selected cell, whereas the first case only includes the selected instance of the cell.

GROUPING OF GEOMETRIES

The import operation can import each layer into its own geometry object, or import each object from the layers in the file into its own geometry object. To determine which method to use select one of the available alternatives from the **Grouping of geometries** list:

- **By layer**. Use this alternative to combine the imported objects from each layer into a single geometry object. Several geometry objects, one for each imported layer, are output by the import.
- **No grouping**. Use this alternative to import each object from the layers into individual geometry objects. To construct the geometry objects the import performs a union of all the segments.

With the **Type of import** setting the extrusion of layers can be switched on or off. The **Full 3D** option imports all layers with a thickness, as specified in the **Layers to import** table. With the **Metal shell** option layers are imported as one surface object per layer (with grouping **By layer**), or one surface object per imported object (with grouping **No grouping**).

CONTROLLING THE LAYER STACKUP

Since layer stackup information is not included in GDS-II files, the layers are assigned a default zero thickness in the **Layers to import** table. These values can be edited before importing the GDS-II file.

Select the **Manual control of elevations** checkbox to position the layers in the z direction. When **Manual control of elevations** is not selected, the z positions of the layers are calculated automatically from the layer **Thickness** values.

Select the **Show names from file** checkbox to display the layer names as included in the GDS-II file. This is useful after assigning new layer names that are used in the COMSOL application.

If the datatype record is used in the file for grouping of objects on a layer you can select the **Split by datatype** checkbox to treat geometry objects of the same datatype as a separate layer. The parent layer is then split into as many separate layers as there are datatypes defined. For example, a layer that contains objects that are grouped into two datatypes will be split into two separate layers after selecting the checkbox. The name of the new layers is obtained by appending the datatype to the name of the parent layer.

Note: When the **Split by datatype** checkbox is selected the import will generate the geometry only for those layers that contain objects with the datatype record set in the file.

The **Layers to import** table contains the following columns:

- The **Name** column displays the layer names from the file. You can edit the entries to give more descriptive names to the layers. The names that appear here are used to name the resulting geometry objects and the selections when the **Layer selections** checkbox is selected under **Selections of Resulting Entities**.
- The **Name in file** column is visible only when you select the **Show names from file** checkbox above the table. The **Name in file** column always displays the layer name from the GDS-II file, even after editing the **Name** column.
- The **Type** column declares the type of layers. GDS-II files do not specify layer types, and the layer type is always set to **Metal**.
- The values in the **Thickness** column are used as the extrusion distances for the layers. Layer elevations are also calculated based on these values when **Manual control of elevations** is not enabled. The values in the layer **Thickness** column can always be changed prior to import. Note that GDS-II files do not contain layer thickness (nor elevation) data.
- The values in the **Elevation** column control the lower *z* positions of the layers. By adjusting the values you can adjust the position of layers in the stackup. The **Elevation** column is only displayed when **Manual control of elevations** is enabled.
- The **Import** column. Clear the checkbox for layers that do not need to be imported.

To save the layers table to a text file, click the **Save to File** button ( ) under the table. To load layer table information from a text file, click the **Load from File** button ( ) under the table. Only the editable data in the table is affected when loading from file. Layer data in the file that does not match the existing layers in the table is ignored. To reset the thickness and elevation values in the table to the last read data from the GDS file click the **Reset Thickness and Elevation** button ( ). To select the **Import** checkbox in the table for all layers click the **Select All Metal and Dielectric Layers for Import** button ( ). Click the **Clear All Imports** button ( ) to clear all checkboxes in the **Import** column. To reload the layer information from the GDS file to the table click the **Reload Layers** button ( ). This is useful in case the GDS file has changed since the last time it was read, and includes additional layers. By clicking **Reload Layers** you can load the

information for the new layers into the table, while keeping any edits you have done in the table for layers that are also found in the new file. Note that layers that are not found in the new file are removed from the table.

INTERIOR BOUNDARIES

By default the **Keep interior boundaries** checkbox is cleared to instruct the import to remove all interior boundaries on the imported layers. This keeps the geometry complexity to a minimum and can also make the import more robust in some situations.

ARC RECOGNITION

Recognition of arcs and straight lines can significantly reduce the complexity of imported layouts from GDS files. With the **Recognize arcs** set to **Automatic**, all polygon chains that represent arcs are identified and replaced with more efficient curve objects, and polygon segments that lie on a single straight line are recognized and joined into a single straight segment.

With **Recognize arcs** set to **Manual**, the following settings will help you fine tune the process of the merging of segments into a single circular arc:

- **Minimum angle between segments**: this parameter prohibits merging of two adjacent polygon segments to a circular arc if the angle between them is less than the provided number.
- **Maximum angle between segments**: this prohibits merging of two adjacent polygon segments to a circular arc if the angle between them is greater than the provided number.
- **Maximum curvature deviation**: this prohibits merging a polygon segment to the constructed circular arc if that would (relatively) change the curvature by more than the provided number.
- **Maximum length deviation**: this prohibits merging of two adjacent polygon segments to a circular arc if the relative difference between the lengths is greater than the provided number.
- **Maximum deviation from circle**: the allowed maximum relative error between the polygon segments and the constructed circular arc.

In manual arc recognition mode, the **Find straight lines** checkbox controls whether to convert several polygon segments that lie on a straight line into a single straight segment. This option uses the number in the **Minimum angle between segments** field to determine if a group of segments lie on the same straight line.

Short polygon segments that are difficult to eliminate using arc recognition can usually be removed as described in the section Ignore Vertices in Layers.

REPAIR

Geometry repair is controlled via the **Repair imported data** checkbox and the **Repair tolerance** list. Change the **Repair tolerance** from **Automatic** to **Relative** to manually specify a tolerance in the **Relative repair tolerance** field. The geometry repair allows to repair incorrectly drawn objects, such as polygons with (small) gaps.

If selected, the geometry repair is also active when combining the imported objects in a cell or layer. By increasing the repair tolerance, problems with short edges arising when combining the objects can sometimes be circumvented.

IGNORE VERTICES IN LAYERS

Select the **Ignore vertices with continuous tangent** checkbox (cleared by default), to remove, in each layer, vertices that connect two edges that have tangent directions making an angle less than the **Continuous tangent tolerance**.

Select the **Eliminate short edges** checkbox (cleared by default), to eliminate, by ignoring one or both of their adjacent vertices, edges that are shorter than the **Maximum edge length**.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Layer selections** checkbox to create predefined selections — for domains, boundaries, and objects — in subsequent nodes in the geometry sequence for each imported layer in the GDS file. To also make all of one of the resulting entities that the layers consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, or **Boundary selection**. The default is **All levels**, which makes predefined selections available on all applicable levels, suitable for use with materials and physics defined in domains and boundaries. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Cell selections** checkbox to create predefined selections — for domains and boundaries — in subsequent nodes in the geometry sequence for each imported cell in the GDS file. To also make all of one of the resulting entities that the cells consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, or **Boundary selection**. The default is **Domain selection**, if available, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Exporting OASIS

With a license for the ECAD Import Module you can export 2D geometry objects to the OASIS (`*.oas`) format. To do this:

- right-click the **Geometry** node and select **Export** (), or
- in the **Geometry** toolbar click **Export** ().

Then, in the **Export** window, the **File type** list, select **OASIS (`*.oas`)**. Use the **Browse** () button to choose the filename, or enter a filename including the path in the **Filename** field.

Next, select **Export selected objects** to export only chosen geometry objects or select **Export entire finalized geometry** to export the resulting geometry of a Form Union or Form Assembly operation. Note that **Export entire finalized geometry** is not available when exporting from 3D components and the **Selection** list will only show geometry objects that come from work plane features.

To export the geometry to the selected file, click the **Export** ( ) button. A confirmation message appears in the Messages window.

ADVANCED

When exporting to the **OASIS** file format the **Approximation detail** list is available with the following options:

- **As in Graphics preference Detail** (default). This takes the rendering mesh from the Graphics window, which is controlled by the Graphics preference Detail.
- **Coarse**. This uses a rendering mesh with detail Coarse.
- **Normal**. This uses a rendering mesh with detail Normal.
- **Fine**. This uses a rendering mesh with detail Fine.

When exporting from 3D components the exported file will contain corresponding 2D geometry objects in the local coordinate system of the corresponding work plane. In this case a table with each work plane will appear. The table consists of three columns representing:

- **Work plane** - It contains the tags of all work planes in the geometry. If no object from work plane is selected in the **Selection** list, the corresponding row will be grayed out.
- **Layer number** - The number assigned to each layer in the file. You can edit this number.
- **Layer name** - The name assigned to each layer. The default names use the corresponding feature labels and they can be edited.

During export the selected objects that belong to a specific work plane will be saved in the file, and they will be assigned to the layer specified for that work plane in the table. It is possible to assign multiple work planes to the same layer by entering the same layer number in multiple rows. Rows with the same Layer number must also have the same Layer name.

The GDS-II File Format

The GDS-II file format is commonly used for mask layout production used in the manufacturing process of semiconductor devices and MEMS devices. The file is a binary file, containing information about drawing units, geometry objects made of polygons, and object drawing hierarchy. The drawing hierarchy is made up of a library of cell definitions, where each cell can be instantiated (drawn several times) with scaling, translation, mirroring, and rotation. It is also possible to repeat a cell as an array of drawn objects. This is very useful for mask layouts of integrated circuits, which

often consist of millions of transistors. There are usually only a few transistor configurations present on the layout, and each transistor configuration only has to be defined once. You can configure the import to include only selected cells, see Cell Selection. Geometry objects in the cells belong to different layers which represent different steps in the manufacturing process. Geometry objects can also be assigned data types that are sometimes used to group together objects.

FILE EXTENSION

The file extension of the GDS-II format is usually `.gds`, and the ECAD import requires it to be so, otherwise it cannot identify the file as a GDS-II file. If the file has a different extension, it must be changed to `.gds` before importing the file.

SUPPORTED FEATURES

There are several record types in a GDS-II file that are of no interest in a geometry import and these are ignored. There are also a few record types that actually could be imported as a geometry object, but are also ignored. One such example is the Text record, which produce a lot of mesh elements and is usually of no interest in a simulation. Below is a list of the supported record types:

- Boundary: a closed polyline object
- Box: a box object
- Path: a path with a thickness
- Sref: an instance of a cell that can be translated, rotated, scaled, and mirrored
- Aref: an n-by-m array of Sref objects
- Element: specification of a cell

Preparing the Geometry

The GDS-II format does not contain layer stackup information, such as thickness and elevation, this has to be supplied before import to create a 3D structure by extrusion. For extruding the layers, the import algorithm performs several extrude operations to generate one geometry object per layer.

Several layers on the same height is common for semiconductor layouts, where the fabrication process includes deposition followed by etching and then redepositing of a different layer. To handle such a process scheme, after the import, you can do etching by removing a layer from other objects, by using the **Difference** button from the **Boolean Operations** submenu in the **Geometry** toolbar. You can find the various import options described under the section Grouping of Geometries.

3

Programming and Command Reference

In this section you find detailed COMSOL API reference information for the geometry features in the ECAD Import Module.

In this section:

Commands

Import of ECAD Files

PURPOSE

Import geometry objects from an ECAD file in 2D and 3D using the ECAD Import Module

SYNTAX

```
model.geom(<tag>).feature().create(<ftag>,"Import");
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property);
model.geom(<tag>).feature(<ftag>).getStringMatrix("layerprop");
model.geom(<tag>).feature(<ftag>).importData();
```

DESCRIPTION

Use `model.geom(<tag>).feature().create(<ftag>,"Import")` to create a geometry import feature.

When the property `filename` is set to a file recognized as an ECAD file, the property `ecadtype` is automatically initialized to either `gds`, `ipc2581`, or `odb++`. The following properties are available:

TABLE 3-1: VALID PROPERTY/VALUE PAIRS.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`contangletol`	double	`5 or 5[deg]`	Continuous tangent tolerance. Only used if `ignorecontvtx` is on.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.
`createselection`	on | off	`off`	Create selections.
`ecadtype`	`gds` | `ipc2581` | `odb++` | `unknown`	`unknown`	Type of ECAD file. A read only property that is automatically initialized from the property `filename`.
`elevation`	String[]	Empty	Z-position of layers.

TABLE 3-1: VALID PROPERTY/VALUE PAIRS.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`elimshortedg`	`on` | `off`	`off`	Eliminate short edges by ignoring vertices in layers. Only used when type is ecad.
`filename`	String	Empty	Filename.
`fileunit`	String	Empty	The length unit in file. Read only.
`grouping`	`all` | `layer` | `none`	`layer`	The grouping of the imported layers, where `all` returns one single object, and `layer` gives you one object per layer. For the import of GDS files the use of `all` is not recommended. In 2D, `all` is the same as `layer`.
`height`	String[]	Empty	Thickness of layers.
`ignorecontvtx`	`on` | `off`	`off`	Ignore vertices with continuous tangent in layers. Only used when type is ecad.
`importlayer`	String[]	Empty	The entries in the array govern if the corresponding layer is to be imported. Each entry value is either "on" or "off".
`importtype`	`full3d` | `shell`	`full3d`	Determine if metal layers are imported as solid or faces in 3D.
`intbnd`	`on` | `off`	`off`	Keep interior boundaries on layers.
`layername`	String[]	Empty	The user names for the layers. Empty string implies the name is taken from the ECAD file.
`maxedglen`	double	0	Maximum edge length. Only used if `elimshortedg` is on.
`repairgeom`	`on` | `off`	`on`	Repair imported objects.

TABLE 3-1: VALID PROPERTY/VALUE PAIRS.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`repairtol`	double	`1e-5` (for GDS) `1e-6` (for ODB++ and IPC-2581	Repair tolerance, relative to size of union of imported objects. Setting this property will automatically set `repairtoltype` to relative and repairgeom to on.
`repairtoltype`	`auto` | `relative`	`auto`	Determine if repair tolerance is set automatically or manually by the `repairtol` property.
`sellayer`	`on` | `off`	`on`	Create layer selections.
`sellayershow`	`all` | `dom` | `bnd` | `off`	`all`	Show layer selections in physics, materials, and so on, and in part instances (when `sellayer` is on). This property is not available in a work plane's Plane Geometry.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	The highest available entity level except `obj`; usually `dom`.	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`updategeomunit`	Boolean	`true or false`	The default is `true` if the feature is the first in the geometry sequence, otherwise `false`. If `true`, the geometry sequence unit will be updated to the `fileunit` value when performing the import.

The file specified by `filename` can be of any of the following formats:

TABLE 3-2: SUPPORTED FILE FORMATS.

FILE FORMAT	FILE EXTENSIONS	ECADTYPE
GDS	`.gds`	`gds`
IPC-2581	`.cvg,.xml`	`ipc2581`
ODB++®	`.zip, .tar, .tgz, .tar.tgz`	`odb++`

The imported objects are represented using COMSOL's geometry modeler.

The property `ecadtype` determines which properties are available, see Table 3-2 for supported types. The property `ecadtype` is read only, and it is automatically initialized from the property `filename`, according to Table 3-2.

The property `importlayer` determines which layers to import. It is a string array of the same length as the number of layers, holding the strings `on` or `off`. The property is initialized with a valid default when setting the property `filename`.

To inspect available layers in a file, use the method

```
model.geom(<tag>).feature(<ftag>).getStringMatrix("layerprop");
```

It returns a `String[][]`, with the same number of rows as the number of layers, that contains information about each layer.

The property `height` determines the height of all layers. It is a string array of the same length as the number of layers, holding string representations of the layer height, for example `1[mm]`. The property is initialized with a valid default when setting the property `filename`.

The property `elevation` determines the Z-position of all layers. It is a string array of the same length as the number of layers, holding string representations of the layer elevation. Read more about how this property is initialized with a valid default for the supported formats in the following sections:

- Import of ODB++ and IPC-2581 Files,
- Import of GDS Files.

If `selresult` is set to `on`, a selection is created for all resulting entities of each type (object, domain, boundary, edge, and point), for use in the geometry sequence. To access the object selection, use `model.geom(`*<tag>*`).selection(`*<ftag>*`)`, where *<tag>* is the geometry tag and *<ftag>* is the feature tag. To access the other selections, use `model.geom(`*<tag>*`).selection(`*<ftag>*`.`*<lvl>*`)`, where *<tag>* is the geometry

tag, *<ftag>* is the feature tag, and *<lvl>* is one of `dom`, `bnd`, `edg`, or `pnt`. If, in addition, `selresultshow` is set to a value other than `off`, all or some of these selections appear for use outside the geometry sequence. To access these selections, use `model.selection(`*<tag>*`_`*<ftag>*`_`*<lvl>*`)`, where *<tag>* is the geometry tag, *<ftag>* is the feature tag, and *<lvl>* is one of `dom`, `bnd`, `edg`, or `pnt`.

If `sellayer` is set to `on`, a selection is created for all resulting entities of the types object, domain, and boundary, of each layer, for use in the geometry sequence. To access the object selections, use `model.geom(`*<tag>*`).selection(`*<ftag>*`_`*<otag>*`)`, where *<otag>* is a tag derived from the name of the imported layer. *<otag>* is derived by replacing space and dot characters with underscore characters and removing other characters that are not numbers or uppercase or lowercase English characters (A–Z and a–z). Additionally, if required to make *<otag>* unique, `_`*<m>* is appended, where *<m>* is an integer. To access the other selections, use `model.geom(`*<tag>*`).selection(`*<ftag>*`_`*<otag>*`.dom`) or `model.geom(`*<tag>*`).selection(`*<ftag>*`_`*<otag>*`.bnd`), where *<otag>* is a tag derived from the name of the imported layer. If, in addition, `sellayershow` is set to a value other than `off`, all or some of these selections appear for use outside the geometry sequence.

To access these selections, use `model.selection(`*<tag>*`_`*<ftag>*`_`*<otag>*`_dom`) or `model.selection(`*<tag>*`_`*<ftag>*`_`*<otag>*`_bnd`), where *<otag>* is a tag derived from the name of the imported layer.

IMPORT OF ODB++ AND IPC-2581 FILES

Properties that are only available when `ecadtype` is `ipc2581`, `odb++`.

TABLE 3-3: VALID PROPERTY/VALUE PAIRS FOR IPC-2581 AND ODB++.

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
`compattrtag`	String	`featureTag .component`	Component attribute tag
`compbottomoffset`	double	`0`	Elevation offset for bottom components (only in 3D)
`compheight`	`fromfile` | `zero`	`fromfile`	Height of components (only in 3D)
`comptopoffset`	double	`0`	Elevation offset for top components (only in 3D)
`dielectric`	`on` | `off`	`on`	Import dielectric regions
`drillhandling`	`separate` | `modify`	`modify`	Drill layer handling

TABLE 3-3: VALID PROPERTY/VALUE PAIRS FOR IPC-2581 AND ODB++.

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
elevationtype	metalabove | metalbetween | manual	metalabove	Select how elevations should be determined.
ignorelinewidth	on | off	off	Import line symbols as curve objects by ignoring the line width during import
ignoreoffset	double	-0.1[mm]	Offset used when ignoreoutsideboard is on. The file unit is used if no unit is specified.
ignoreoutsideboard	on | off	off	Ignore objects for which the defining points fall outside the board boundary incremented with the offset defined by ignoreoffset
ignoretext	on | off	on	Ignore objects marked as text in the files
importcomponents	on | off	off	Import component outlines
importnet	String[] with on/off values	all on	Select which nets to import. Values for checkboxes are on or off
netnames	String[]		Names of nets. Read only
nonplatedholedom	on | off	off	Create domains for non-plated holes
padattrtag	String	featureTag .pad	Pad attribute tag. It is created when padstocreate is all or when is set to table that is not empty
padcomponents	String[]		Corresponds to the column Components in the GUI table
padpins	String[]		Corresponds to the column Pins in the GUI table
padstocreate	table | all	table	Specify how to choose which pads should be created
platedholedom	on | off	on	Create domains for the cores of plated holes
platedholethrough	on | off	on	Plated hole cores through metal layers

TABLE 3-3: VALID PROPERTY/VALUE PAIRS FOR IPC-2581 AND ODB++.

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
platingthickness	double	0	Drill hole plating thickness
selnetcontributetobnd	String[]	Empty	Tags of cumulative selection to contribute to (or none to not contribute), for boundary net selections
selnetcontributetodom	String[]	Empty	Tags of cumulative selection to contribute to (or none to not contribute), for domain net selections
selnetkeepbnd	on | off	false	Keep boundary net selections
selnetkeepdom	on | off	false	Keep domain net selections
selnetnamebnd	String[]	Empty	Names of boundary net selections (read only)
selnetnamedom	String[]	Empty	Names of domain net selections (read only)
selnetperlayer	on | off	off	Create intersection selections between net selections and all other selections
selnetshowbnd	on | off	off	Show boundary net selections in physics, materials, and so on; in part instances; or in 3D from a plane geometry
selnetshowdom	on | off	off	Show domain net selections in physics, materials, and so on; in part instances; or in 3D from a plane geometry
selnettagbnd	String[]	Empty	Tags of boundary net selections (read only, hidden in GUI)
selnettagdom	String[]	Empty	Tags of domain net selections (read only, hidden in GUI)
showlayer	String[] with on/off values	all on	Show or hide layer in preview
shownet	String[] with on/off values	all on	Show or hide nets in preview
showoutline	Boolean	true	Show or hide the board outline in preview

TABLE 3-3: VALID PROPERTY/VALUE PAIRS FOR IPC-2581 AND ODB++.

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
`unitemetalanddiel`	`on` | `off`	`on`	Unite metal layer and in-layer dielectric
`viadom`	`on` | `off`	`on`	Create domains for cores of vias
`viathrough`	`on` | `off`	`off`	Via cores through metal layers

The property `elevationtype` controls how imported layers are positioned in the *Z* direction. When `elevationtype` is `metalabove` or `metalbetween`, the imported layers are stacked on top of each other with no gaps, so the *Z*-positions are computed from the `height` property. When `elevationtype` is `manual` and grouping is set to `layer` or `none`, the imported layers are positioned in the *Z* direction as specified by the `elevation` property. To switch to manual control of elevations it is recommended to first set `elevationtype` to `manual`, then set the values for the `height` and `elevation` properties.

The property `elevation` is initialized with a valid default on these occasions:

- when setting the property `filename`,
- when setting the property `elevationtype` to `metalabove` or `metalbetween`,
- when setting the property `elevationtype` to `manual`,
- when `elevationtype` is `metalabove` or `metalbetween`, and the value of the `height` property is changed,
- when `elevationtype` is `metalabove` or `metalbetween`, and the value of the `importtype` property is changed.

To access the values of different attributes created during import, enter

```
model.component(<ctag>).geom(<tag>).attribute(<attrtag>).
values();
```

where *<attrtag>* is a tag of created attribute, *padattrtag* for pad attribute tag, or *compattrtag* for component attribute tag.

To access the values of different attribute fields created during import, enter

```
model.component(<ctag>).geom(<tag>).attribute(<attrtag>).
fieldValues(<value>,<field>);
```

where *<attrtag>* is a tag of created attribute, *padattrtag* for pad attribute tag, or *compattrtag* for component attribute tag. *<value>* corresponds to the value found for the specified attribute, and *<field>* corresponds to the field name. For example,

in case of components, the value would be `C1`, or in the case of pads, `C2/1`, where `C2` represents the component connected to the pad and `1` represents the pin number. The available field for the components attribute is `package`, while the available fields for the pads attribute are `component`, `pin`, `net`, and `package`. They give the information on which component the pad is connected, to which pin, which package, and which net it belongs to, respectively.

You can use the net selection tag as a key when setting array elements in the other properties by using `set(property,`*`<ntag>`*`, value)`, where *`<ntag>`* is a tag derived from the name of the net selection.

To access net selections, use `model.selection(`*`<tag>`*`_`*`<ftag>`*`_`*`<ntag>`*`_dom`) or `model.selection(`*`<tag>`*`_`*`<ftag>`*`_`*`<ntag>`*`_bnd`), where *`<ntag>`* is the tag of the net selection.

IMPORT OF GDS FILES

If the file type is GDS, you can create additional selections for each imported cell by using the properties `selcell` and `selcellshow`.

Properties that are only available when `ecadtype` is `gds`.

TABLE 3-4: VALID PROPERTY/VALUE PAIRS FOR GDS.

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
`arcdistancetol`	double	`0.3`	Tolerance for segment length deviation.
`arcmaxangle`	double	36	Maximum angle for segments.
`arcminangle`	double	`3.6`	Minimum angle for segments to part of an arc.
`arcmaxcircledev`	double	`1e-3`	Maximum allowed point deviation from a circle, relative to the circle radius.
`arcradiustol`	double	`0.4`	Tolerance for arc curvature.
`cell`	String	Empty	Name of cell to import. The top cell is imported by default.
`cellfilter`	String	Empty	Name of cell to include. Leave empty to include all cells below the cell selected by the `cell` property.
`findarcs`	`auto` | `manual` | `off`	`auto`	Control arc recognition.

TABLE 3-4: VALID PROPERTY/VALUE PAIRS FOR GDS.

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
findlines	on | off	on	Turn on straight line recognition. Only available when findarcs is manual.
manualelevation	on | off	off	Manual control of elevations.
selcell	on | off	off	Create cell selections.
selcellshow	all | dom | bnd | off	dom	Show cell selections in physics, materials, and so on, and in part instances (when selcell is on). This property is not available in a work plane's Plane Geometry.
splitbydatatype	on | off	off	Treat datatypes as separate layers

The property manualelevation controls how imported layers are positioned in the *Z* direction. When manualelevation is off, the imported layers are stacked on top of each other with no gaps, so the *Z*-positions are computed from the height property. When manualelevation is on and grouping is set to layer or none, the imported layers are positioned in the *Z* direction as specified by the elevation property. To switch to manual control of elevations it is recommended to first set manualelevation to on, then set the values for the height and elevation properties.

The property elevation is initialized with a valid default on these occasions:

- when setting the property filename,
- when setting the property manualelevation to off,
- when manualelevation is off, and the value of the height property is changed,
- when manualelevation is off, and the value of the importtype property is changed.

When findarcs is auto or manual, line segments are combined to form arcs. If findarcs is manual, the properties arcdistancetol, arcradiustol, arcminangle, arcmaxangle, and findlines can be used to tune the arc recognition algorithm, otherwise the algorithm tries to determine optimal parameters.

Exporting Geometry to OASIS (2D)

To export the finalized geometry to a file, enter

```
model.component(<ctag>).geom(<tag>).exportFinal(<filename>);
```

where *<filename>* is a string for a file system path or a file location URI referencing a file version in a Model Manager database.

To export selected geometry objects to a file, first select the objects to export using

```
model.component(<ctag>).geom(<tag>).export().selection().
set(<objnames>);
```

Set a file format using

```
model.component(<ctag>).geom(<tag>).export().setType(<format>);
```

where *<format>* is "`oasis`", the OASIS (`*.oas`) file format.

Export the file by entering

```
model.component(<ctag>).geom(<tag>).export(<filename>);
```

ADVANCED SETTINGS FOR OASIS FILES

Set curve approximation detail for OASIS export by using

```
model.component(<ctag>).geom(<tag>).export().
setOasisApproxDetail(<detail>);
```

where *<detail>* is `String` set to `graphics` (default), `coarse`, `normal`, `fine`.

Get curve approximation detail for OASIS export by using

```
model.component(<ctag>).geom(<tag>).export().
getOasisApproxDetail();
```

The method returns a `String`.

Get the Work plane column in Layers table for Oasis export in 3D (read only) using

```
model.component(<ctag>).geom(<tag>).export().
getLayerWorkPlanes();
```

The method returns a `String[]`.

Set the Layer number for a work plane in Layers table for OASIS export in 3D using

```
model.component(<ctag>).geom(<tag>).export().
setLayerNumber(<index>,<layer>);
```

where `<index>` is the index number of the row in the table and `<layer>` is the Layer number you want to assign to that row.

Get the Layer number column in Layers table for OASIS export in 3D using

```
model.component(<ctag>).geom(<tag>).export().getLayerNumbers();
```

The method returns an `int[]` value.

Set the Layer name for a work plane in Layers table for OASIS export in 3D using

```
model.component(<ctag>).geom(<tag>).export().
setLayerName(<index>,<name>);
```

where `<index>` is the index number of the row in the table and `<name>` is a `String` of the Layer name you want to assign to that row.

Get the Layer name column in Layers table for OASIS export in 3D using

```
model.component(<ctag>).geom(<tag>).export().getLayerNames();
```

The method returns a `String[]`.