

## Hughes Circuits Design Rules – Altium Designer

Hughes Circuits, Inc. has provided these recommended design rules for use in your PCB layouts. These design rules are a great starting point for your layout because they will help you to design a PCB that is optimized for manufacturability. These design rules correspond to Hughes Circuits' "Standard" manufacturing level. The design rule file should be imported to your Altium Designer database before beginning placement of components. Depending on the requirements of your specific design, it may be necessary to modify these design rules and/or create additional design rules. Below is a table of the design rules included in the Hughes Circuits, Inc. Altium design rules file. The table contains a description and value of each rule, and the value of the corresponding Altium Designer default rule.

<b><i>Rule Type and Description</i></b>	<b><i>Priority</i></b>	<b><i>Hughes Circuits</i></b>	<b><i>Altium Designer Default</i></b>
---	------------------------	-------------------------------	---------------------------------------

### **Electrical>Clearance**

Polygon Clearance - Spacing between Polygon Pours only, and conductors of different nets.	1	Different Nets Only - 10mil	NONE
General Clearance - Spacing between all conductors of different nets.	2	Different Nets Only - 5mil	Different Nets Only - 10mil

### **Electrical>Short-Circuit**

ShortCircuit - Allows or disallows the possibility of shorts between different nets.	1	Allow Short Circuit - Disabled	Allow Short Circuit - Disabled
--	---	--------------------------------	--------------------------------

### **Routing>Width**

Width - Width of tracks used for routing nets.	1	Min Width - 5mil Preferred Width - 8mil Max Width - 250mil	Min Width - 10mil Preferred Width - 10mil Max Width - 10mil
--	---	--	---

### **Routing>Routing Via Style**

RoutingVias - Size of via pad and hole used for routing nets.	1	Minimum Via Diameter - 22mil Maximum Via Diameter - 32mil Preferred Via Diameter - 26mil  Minimum Via Hole Size - 10mil Maximum Via Hole Size - 20mil Preferred Via Hole Size - 14mil	Minimum Via Diameter - 50mil Maximum Via Diameter - 50mil Preferred Via Diameter - 50mil  Minimum Via Hole Size - 28mil Maximum Via Hole Size - 28mil Preferred Via Hole Size - 28mil
---	---	---	---

<b><i>Rule Type and Description</i></b>	<b><i>Priority</i></b>	<b><i>Hughes Circuits</i></b>	<b><i>Altium Designer Default</i></b>
---	------------------------	-------------------------------	---------------------------------------

#### **Mask>Solder Mask Expansion**

SolderMaskExpansion - Defines the size of the soldermask clearance with respect to the size of the copper pad.	1	Expansion - 0mil	Expansion - 4mil
--	---	------------------	------------------

#### **Mask>Paste Mask Expansion**

PasteMaskExpansion - Defines the size of the pastemask stencil opening with respect to the size of the copper pad.	1	Expansion - 0mil	Expansion - 0mil
--	---	------------------	------------------

#### **Plane>Power Plane Connect**

Via PlaneConnect-Defines connection style of vias only to plane layers of the same net.	1	Connect Style - Direct Connect	NONE
General PlaneConnect-Defines the connection style of all pins to plane layers of the same net.	2	Connect Style - Relief Connect Conductors - 4 Conductor Width - 8mil Air-Gap - 10mil Expansion - 10mil	Connect Style - Relief Connect Conductors - 4 Conductor Width - 10mil Air-Gap - 10mil Expansion - 20mil

#### **Plane>Power Plane Clearance**

Via PlaneClearance -Spacing between via holes and plane layers of different nets.	1	Clearance - 10mil	NONE
General PlaneClearance -Spacing between all through-holes and plane layers of different nets.	2	Clearance - 12mil	Clearance - 20mil

<b>Rule Type and Description</b>	<b>Priority</b>	<b>Hughes Circuits</b>	<b>Altium Designer Default</b>
----------------------------------	-----------------	------------------------	--------------------------------

#### **Plane>Polygon Connect Style**

Via PolygonConnectStyle - Defines the connection style of vias only to polygon pours of the same net.	1	Connect Style - Direct Connect	NONE
General PolygonConnectStyle - Defines the connection style of all pins to polygon pours of the same net.	2	Connect Style - Relief Connect Conductors - 4 90 Angle Conductor Width - 8mil	Connect Style - Relief Connect Conductors - 4 90 Angle Conductor Width - 10mil

#### **Manufacturing>Minimum Annular Ring**

Via MinimumAnnularRing - Defines minimum size of annular ring on routing vias only.	1	Minimum Annular Ring - 5mil	NONE
General MinimumAnnularRing - Defines minimum size of annular ring on plated through holes.	2	Minimum Annular Ring - 10mil	NONE

#### **Manufacturing>Hole Size**

HoleSize - Defines minimum and maximum hole sizes allowed.	1	Measurement Method - Absolute Minimum - 10mil Maximum - 99999mil	Measurement Method - Absolute Minimum - 1mil Maximum - 100mil
--	---	--	---

#### **Manufacturing>Hole To Hole Clearance**

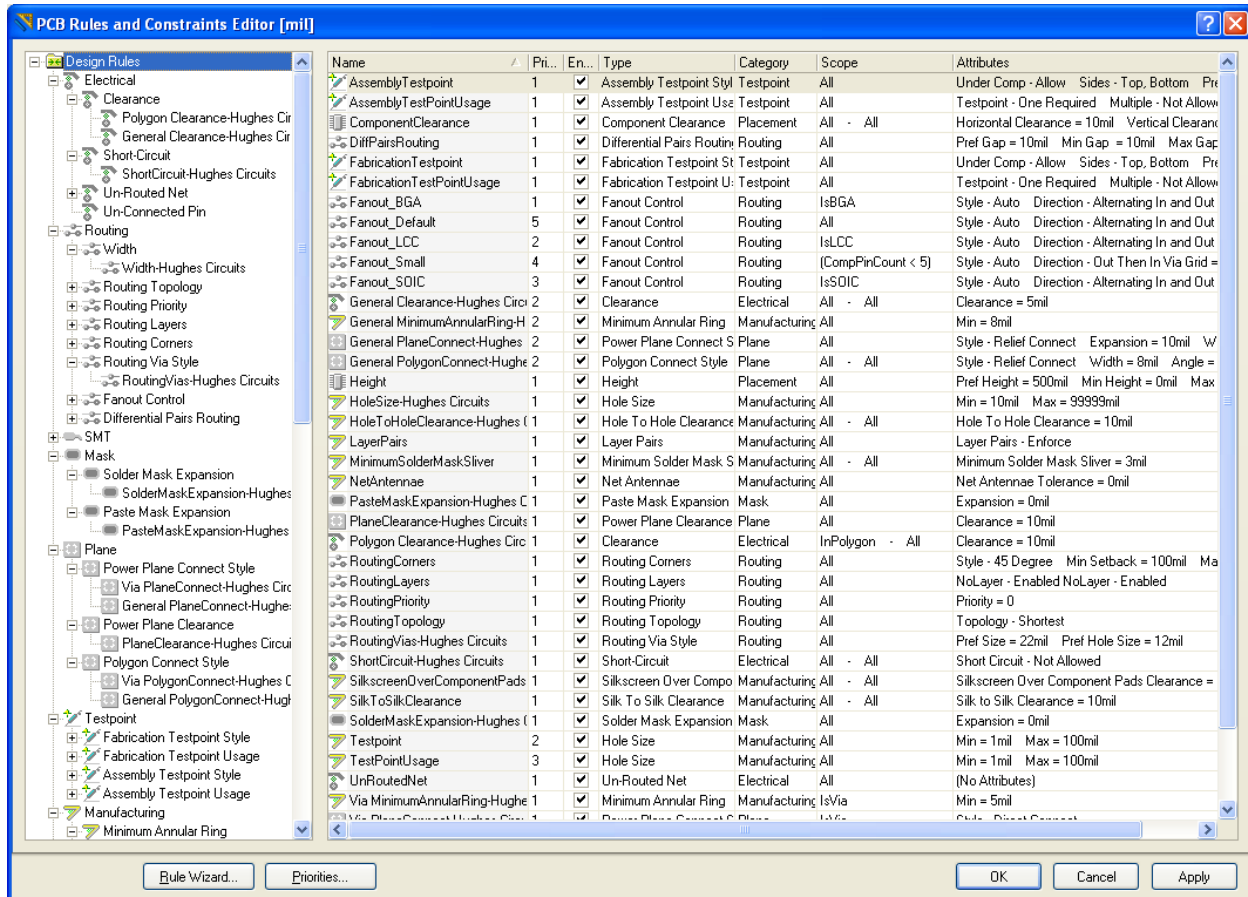
HoleToHoleClearance - Defines minimum distance between any two holes.	1	Allow Stacked Micro Vias - Disabled Hole to Hole Clearance - 10mil	Allow Stacked Micro Vias - Enabled Hole to Hole Clearance - 10mil
---	---	---	--

#### **High Speed>Vias Under SMD**

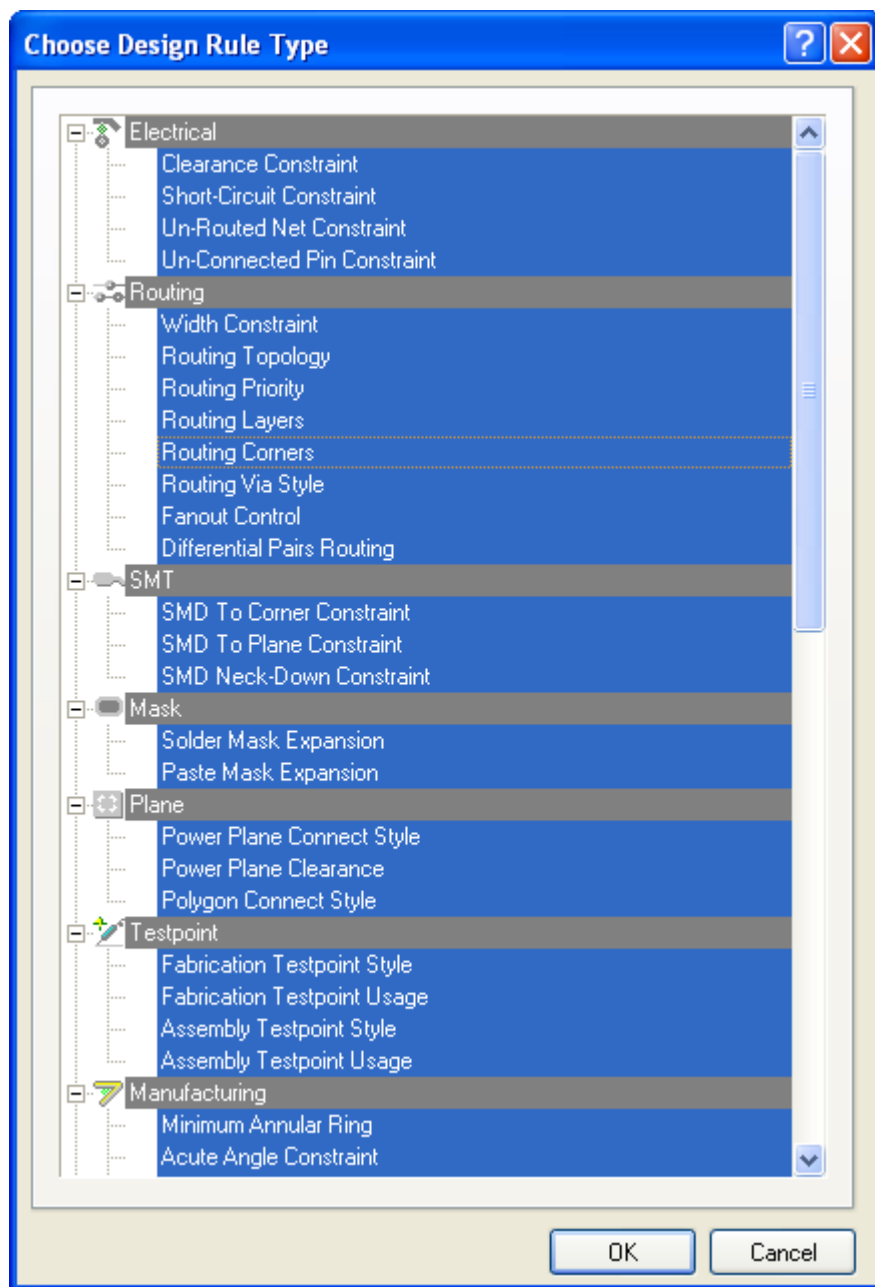
ViasUnderSMD - Allows or disallows the possibility of putting vias inside SMD pads.	1	Allow Vias under SMD Pads - Disabled	Allow Vias under SMD Pads - Disabled
---	---	--------------------------------------	--------------------------------------

## Instructions for Importing Hughes Circuits, Inc. Altium Design Rules

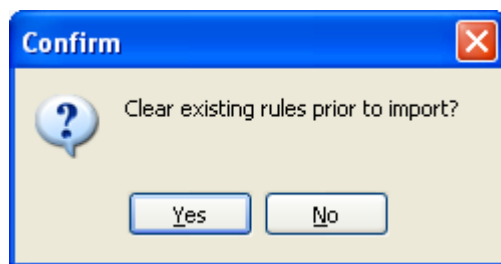
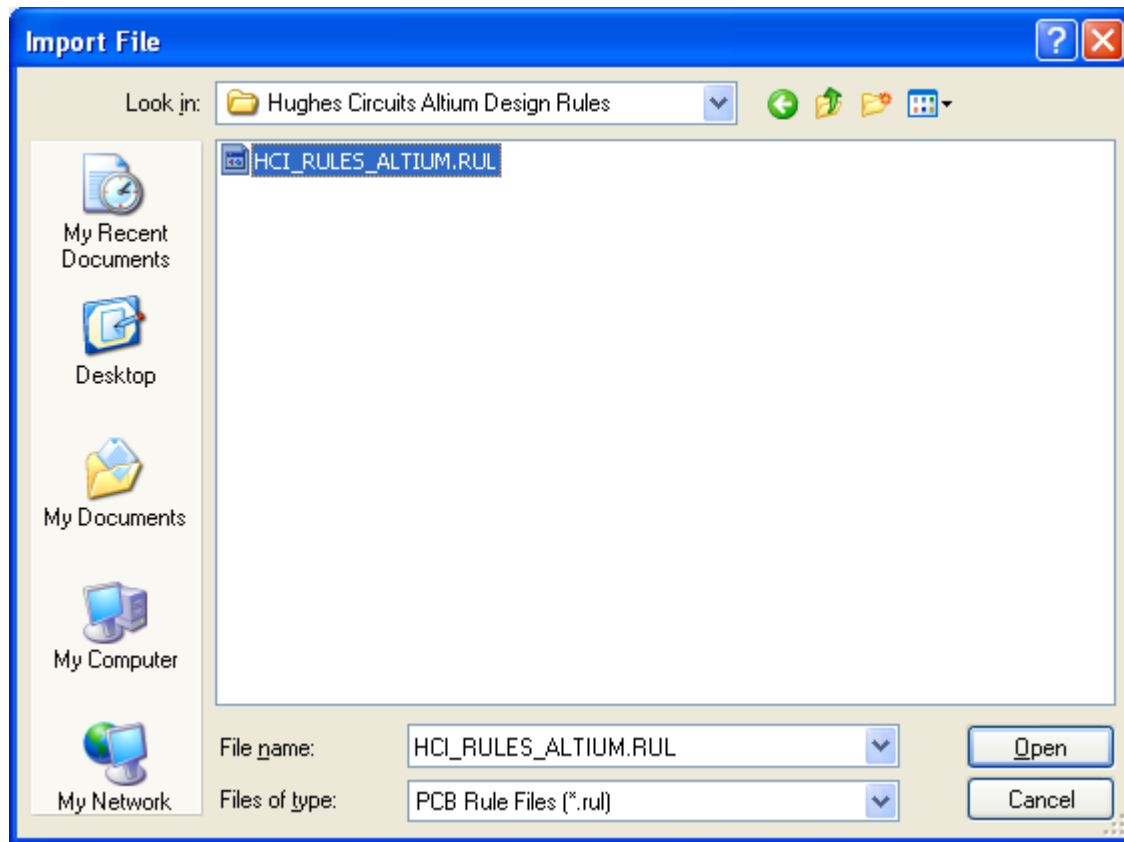
On the main toolbar, select *Design>Rules...*



Right Click anywhere in the left-hand window pane, and choose *Import Rules...* Select the specific rule types you would like to import or simply select all rule types and click *OK*.



Browse to the Hughes Circuits Altium Design Rules file, select *Open*.



Selecting *Yes* will clear out any design rules already entered into your Altium database, leaving you with the Hughes Circuits Design rules and any default Altium rules for which Hughes Circuits does not have a substitute. Selecting *No* will simply add the Hughes Circuits design rules to your existing design rules, giving the Hughes Circuits Design rules the highest priority. Either selection is acceptable.