DOWNSTREAM CAM350 15.0 RELEASE NOTES

Build: 2075
Date: 10/29/2024

Contents

RELEASE SUMMARY

CAM350/DFM 15.0 build 2073 is a minor update to the currently released CAM350 15.0 product. This release includes several customer defect fixes and enhancements as well as language releases for the Japanese and Chinese markets. The new product features in 15.0 are described later in this document. They include a new CAM350 Stencil Kit, a new Auto Align feature and a New Cadence Allegro interface as well as several enhancements.

INSTALLATION AND LICENSING

The installer for the client software (CAM350 15.0 and BluePrint 7.0) will create new folders and you can run both your previous release (CAM350 14.6 and BluePrint 6.6) and your new Release software side by side on the same PC if you wish. There is a new 15.0-7.0 License Manager and License that must be installed. This new License Manager and License will run your 15.0-7.0 software as well as previous releases (ie BluePrint 6.6 and 6.5).

Note: The CAM350 15.0 – BluePrint 7.0 License Manager will NOT run CAM350 12.2 – BluePrint 5.2 product licenses.

For many users your installation should be as simple as this:

1. Run the installation executable
2. Choose "Install or Update Licensing" to install your new License Manager and License File.
    a. If you are an existing customer on maintenance, choose "Install license from media" to install your new license file.
    b. If you are a new user or your license is not found on media, get your new license from DownStream, copy it to your PC and then choose "browse to find License File".
3. Choose "Install DownStream Products" to install the new CAM350 15.0 and BluePrint 7.0 software on your PC.

If you are installing to a Virtual Machine or have any questions, reference our DownStream Installation Guide or contact us at support@downstreamtech.com.

SYSTEM REQUIREMENTS

Your PC should meet or exceed the following requirements:

OS: Windows 10, 11 (**64 bit only**)
Processor: 2GHz or faster
Memory: 8-16GB+
Disk Space: 2GB available, SDD Recommended
Graphics: Discrete graphics card with on-board memory preferred (for best 3D performance)

CAM350 15.0 NEW FEATURES

- ✓ New Stencil Designer
- ✓ New Auto Layer Align
- ✓ New Cadence Allegro Interface
- ✓ Netlist Compare enhancements
- ✓ Design Compare enhancements
- ✓ Licensing support for Nutanix AHV Virtual Machine

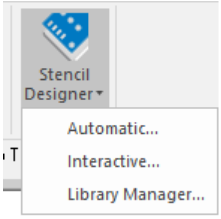
CAM350 15.0 – NEW FEATURES DETAILED

NEW STENCIL DESIGNER

Use the new Stencil Designer to create custom paste stencils sourced from paste mask, solder mask, Top or other layer types. Source a stencil from a selected patterns of flashes or pads or select a footprint. Saved the stencil definition to a stencil library and reuse defined stencils across multiple designs. Use the Application Programming Interface to automate the stencil design process.

EXAMPLE A – In this example, you will use the new Stencil Designer to create stencils for a design using the change shape creation mode.

1. Start CAM350 and choose **File > Open**.
2. Browse to and select the demo file DST Demo 2018.cam from the Demos folder.
3. In the **Layers Display** pane, double click the **Paste Mask Top** layer to make it the only visible layer.
4. On the **View** ribbon, turn off **Top** and **Bottom Parts** (outlines) to hide them.
5. Choose **Design > Stencil Designer**. There are two modes of using Stencil Designer:



Automatic – Create stencils from a library of stencil conversion history.
Interactive – Select pads, flashes or footprints in the design and create stencils. In addition, use Library Manager to manage stencil libraries.

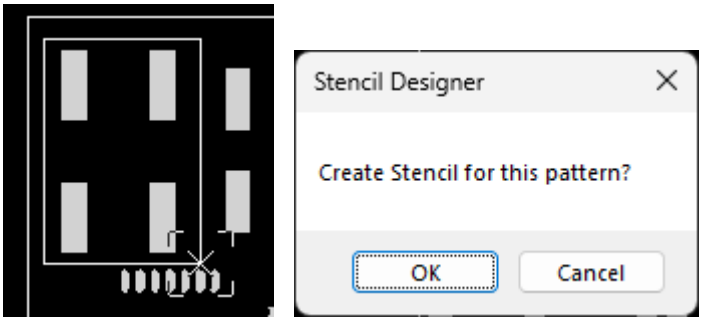
6. For this sample, select **Interactive**. The Select Pattern and Creation Mode dialog appears. Use the dialog to set source layer for selection, mode of selection and creation mode.

7. Set Source Layer to **Paste Mask Top** and set Create Duplicate Layer option disabled.

   The Selection modes are:
   Select Pattern – interactively select pads or flashes, commonly in a pattern.
   Select Footprint – Select a footprint (if present) – must be sourced from a Top or Bottom Layer.
   Select Library – Select a stencil library item and create stencils that match its pattern.

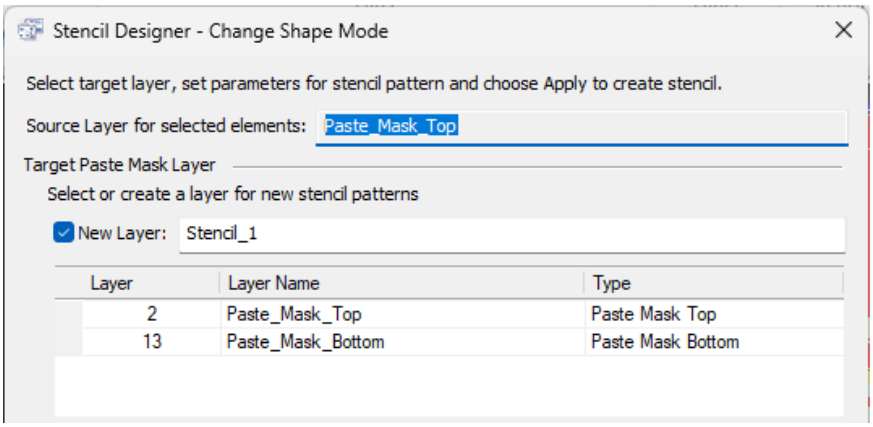
   The Creation Modes are:
   Change Shape – change the shape of the selected pad or flash.
   Offset – keep the shape and shift the stencil shape off of the pad.
   Scale/Resize – keep the shape but shrink or grow its size.
   Heel and Toe – keep the shape and shrink the heel or toe (For PLCCs, SOICs, etc.)

8. Set Selection Method to **Select a Pattern** and set Creation Mode to **Change Shape**. The source layer is set to the active and only visible layer.
9. In the upper left corner of the board, area select the 4 pin oscillator pattern and press right mouse button to accept the selection. The pattern is highlighted.



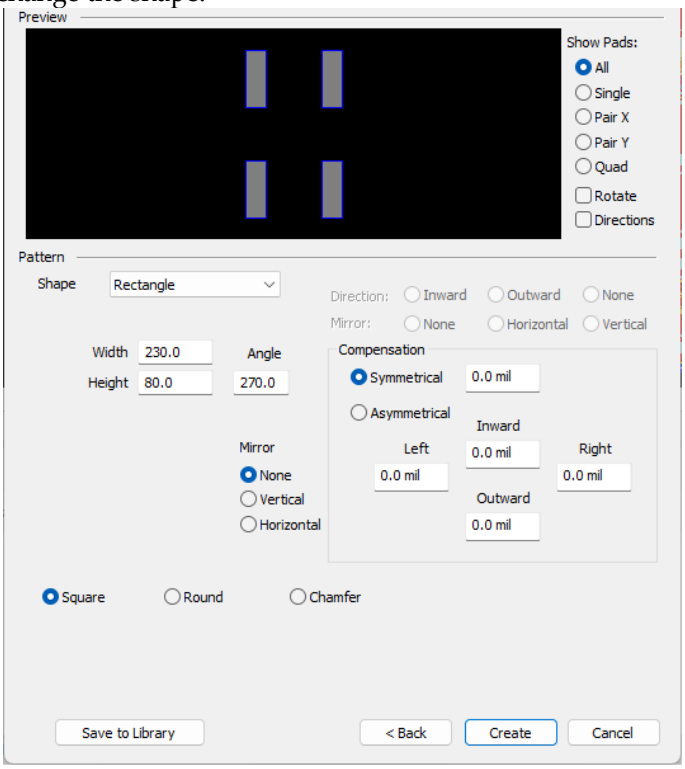
10. Click right mouse button to accept the selection. Click **OK** to start stencil creation. The Stencil Design dialog appears in Change Shape Mode.

The Stencil Design dialog is comprised of two sections. The first area is for selection of the target layer for new stencils. You can choose to create a new layer, or add to an existing Paste Mask layer.

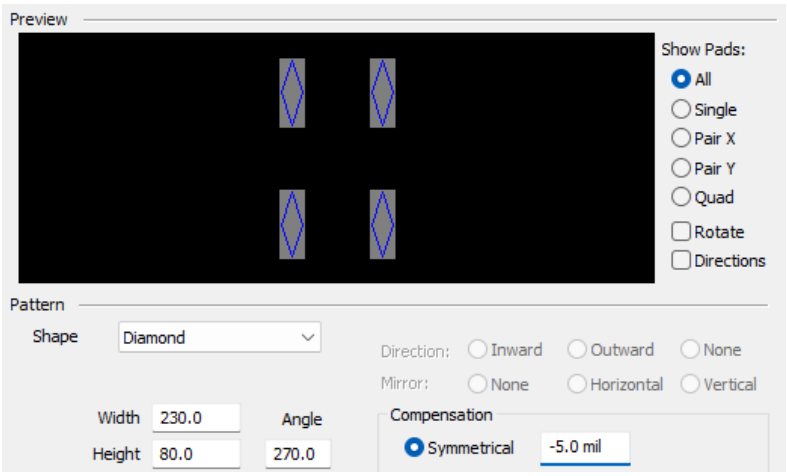


11. For this example, choose New Layer and accept the default name.

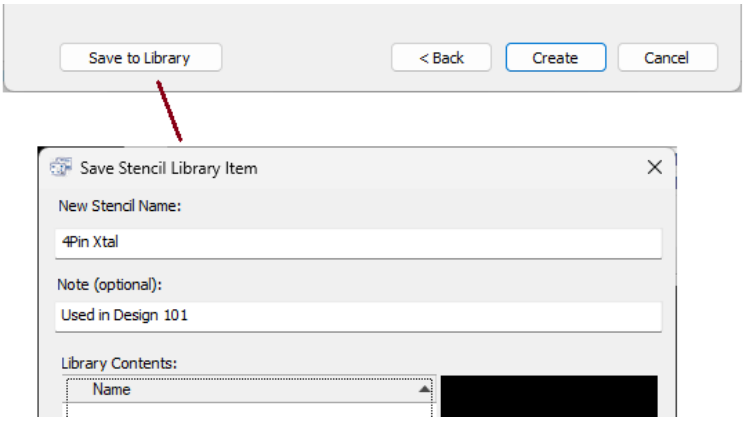
The second area is where you define the new shape and preview the stencil design before you create it. Initially, change shape mode presents a stencil shape identical to the selected pattern. Use the settings to change the shape.



12. Change the shape from Rectangle to **Diamond**.
13. Compensation is for enlarging or shrinking the shape. Set **Symmetrical Compensation** to -5.0 mil (shrink).



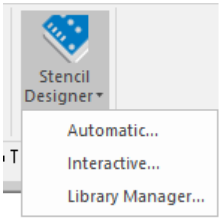
14. Saving to a library is optional. However, it does allow you to reuse stencil design content across designs with similar patterns or footprints. Use **Save to Library** to preserve this definition for use on another design. Enter a Stencil Name and optionally a Note.
15. Be sure Internal Library is selected as the target library. Click **OK** to close the Library Save dialog. The stencil definition is saved to the internal stencil library.



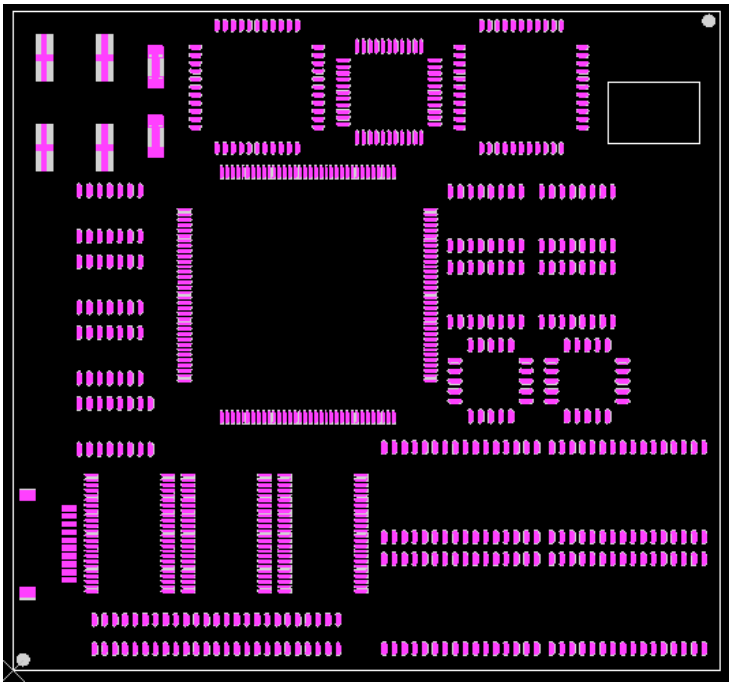
16. In the Stencil Designer dialog, choose **Create**. The Stencil is created and added to the new stencil layer.
17. In the layers pane, double click the new Stencil_1 layer to view the new stencil. Select the Paste Mask Top layer to make it visible also.
18. Experiment on your own with making pattern selections and using the three other creation modes to create stencils.

EXAMPLE B – In this example, you will use the new Stencil Designer to create stencils for a design using the change Automatic creation mode.

1. In CAM350 choose **File** > **Open**. Do not save your last opened file.
2. Browse to and select the demo file DST Demo 2018.cam from the Demos folder.
3. In the **Layers Display** pane, double click the **Paste Mask Top** layer to make it the only visible layer.
4. On the **View** ribbon, turn off **Top** and **Bottom Parts** (outlines) to hide them.
5. Choose **Design > Stencil Designer**.



6. Choose **Automatic**. The Stencil Designer Automatic Creation dialog appears.
7. This dialog has four sections. The source layer selection, target layer selection, stencil library source selection and stencil library processing area.
8. As in the previous example, set **Source Layer** to **Paste Mask Top** and enable **New Layer** as the target layer and leave the default name of **Stencil_1**.
9. In the Library area, select **External Library**. The Open Library dialog appears.
10. Browse to and select the demo library **dst demo.dststlib** and click **Open**. This loads the external library and presents them in the Library Contents column.
11. Each item in the library is a previously created stencil. It is possible some or all of these stencils can be used for this design. Choose **Add All** to add all library items to the column of stencils to process.
12. Click **Create**. The matched stencil library entries are compared to the design and if the patterns match, a stencil is created for each match.



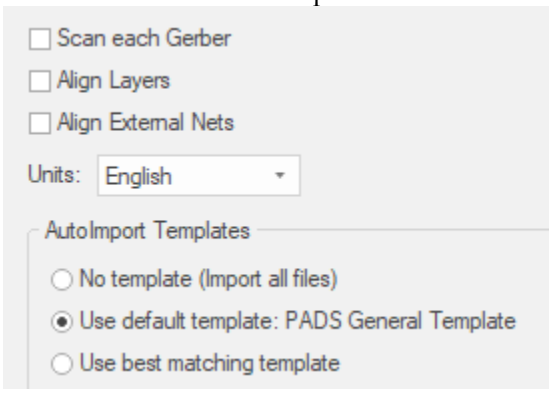
NEW AUTO LAYER ALIGN

Use new automated layer alignment features to align all layers and external netlist during automated import. Alternatively, use the new alignment commands to align content post import.

EXAMPLE A – In this example, we will automatic import a collection of misaligned Gerber, NC Drill and external netlist files and use the new interactive features to align them all.

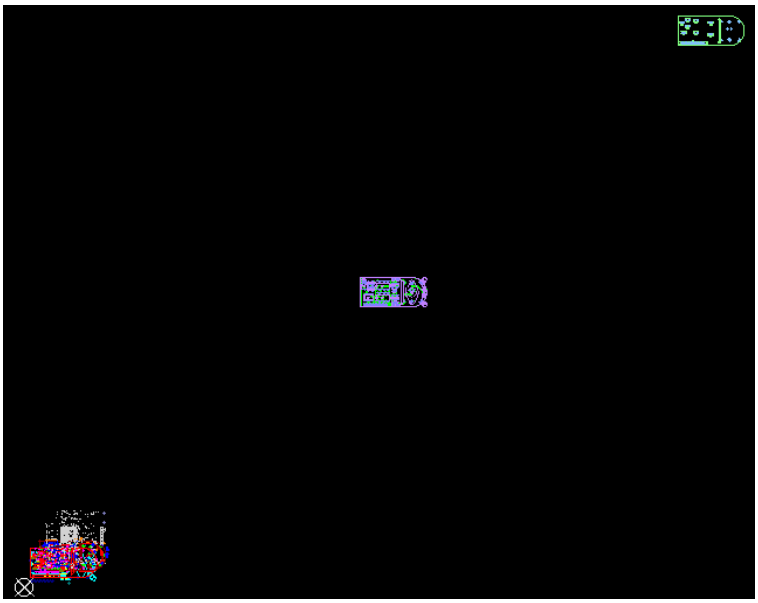
1. Start CAM350 and choose **File** > **Import.**
2. Choose **Automatic Import**.
3. Browse to and select the layer_offset folder.
4. At the bottom of the page there are options to align layers and external netlist. Ignore those for the moment. We will use them in a later step.
5. Set the AutoImport Template to PADS General Template

a.



6. Set AutoImport Templates to **PADS General Template.**
7. For the purpose of this sample, we will skip the remaining pages. Click **Finish**.

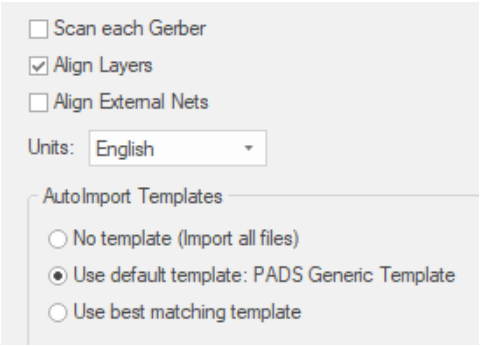
8. Click **No** to decline Netlist Extraction.
9. When the imported content is viewed, you will note all layers were imported with variable origins and as a result are all misaligned.

a. 

10. On the **Tools** ribbon, select **Auto Align Layers**. All imported layers and external netlist are aligned to a common origin.
11. On the **View** Ribbon, select **Zoom All** (or press Home).
12. Choose **File** > **New** to clear the imported data and do not save the file.

EXAMPLE B – In this example, we will automatic import the same collection of misaligned Gerber, NC Drill and external netlist files and use new automated features to align them all.

1. In CAM350, choose **File** > **Import.**
2. Choose **Automatic Import**.
3. Browse to and select the layer_offset folder.
4. At the bottom of the page there are options to align layers and external netlist. Enable **Align Layers**. We will align the netlist in a later step.
5. Set the AutoImport Templates to **PADS General Template.**



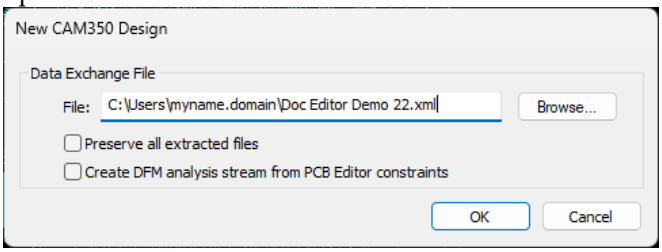
6. For the purpose of this sample, we will skip the remaining pages. Click **Finish**.
7. Click **No** to decline Netlist Extraction.
8. When the imported content is viewed, you will note all imported layers are aligned to a common origin.
9. On the **Analyze** ribbon, select **Netlist Tools** > **Align External Netlist**. The external netlist is aligned to the common design origin.

NEW CADENCE ALLEGRO INTERFACE

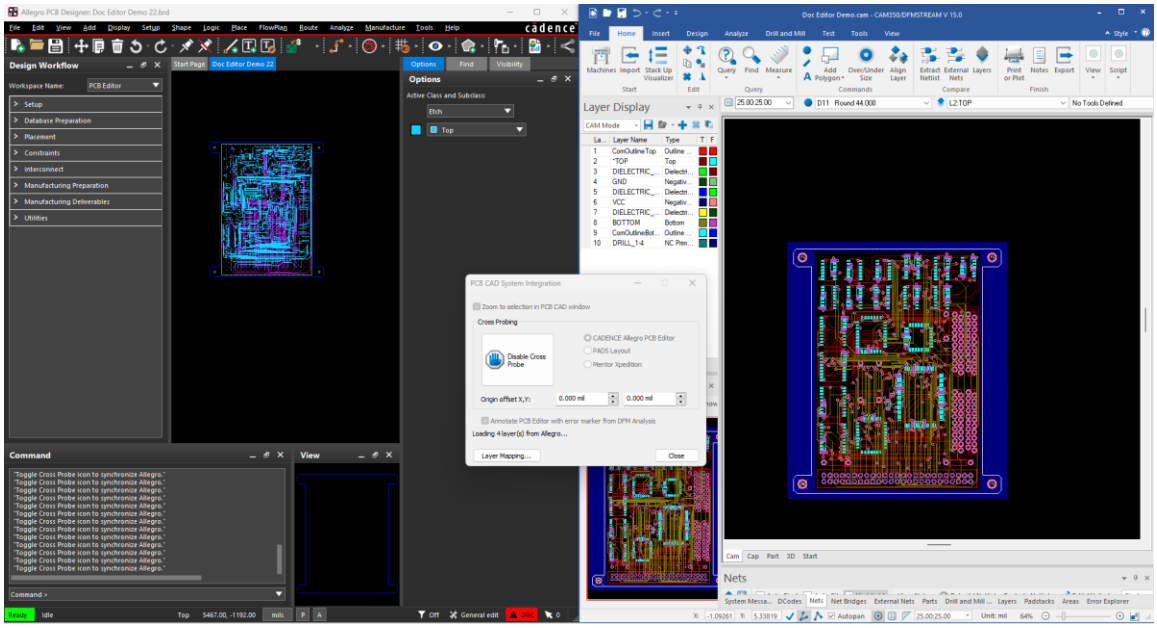
Use the new Cadence Allegro Interface to initiate a new CAM350/DFMStream session directly from the Allegro menu. When selected, an IPC-2581 file is extracted from Allegro, a new session of CAM350 started and the file imported into CAM350. Additionally, crossprobing between CAM350 and Allegro is initiated.

EXAMPLE A – In this example, we start with a design in Allegro, initiate a new session of CAM350, and initiate crossprobing with CAM350.

1. Open "Doc Editor Demo.brd" in Cadence Allegro.
2. In Allegro, select **Tools > New CAM350 Design File**. The New CAM350 Design dialog appears.
3. Accept the default path and filename for the IPC-2581 file, leave other options disabled and click **OK** to proceed.



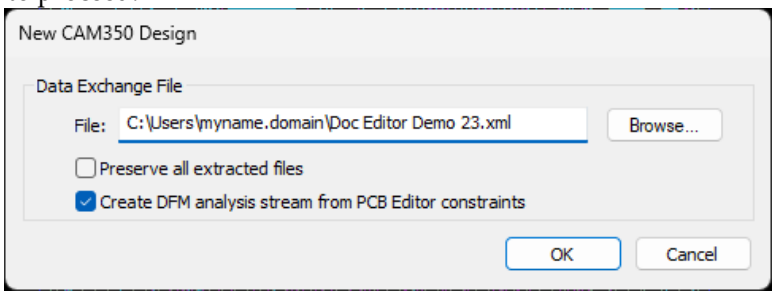
After a few moments, the IPC-2581 is extracted and CAM350 is started. In addition, the data is imported into CAM350 and the System Integration dialog opens with Allegro crossprobing enabled.



4. **Double-click** the **Top** layer to make it the only visible layer in CAM350 and note how the layer display updates identically in Allegro. This is an example of the crossprobing features.

EXAMPLE B – In this example, we start with a design in Allegro, initiate a new session of CAM350, and initiate crossprobing with CAM350. In addition, a custom DFM analysis stream will be derived from the current allegro constraints, and if present, constraint areas. You will then run a DFM Analysis usingthe stream, select analysis errors and crossprobe back to Allegro to see the error location there.

1. Open "Doc Editor Demo.brd" in Cadence Allegro.
2. In Allegro, select **Tools > New CAM350 Design File**. The New CAM350 Design dialog appears.
3. Accept the default path and filename for the IPC-2581 file.
4. Enable the **Create DFM analysis stream from PCB Editor constraints** option.
5. Click **OK** to proceed.



After a few moments, the IPC-2581 is extracted and CAM350 is started. In addition, the data is imported into CAM350 and the System Integration dialog opens with Allegro crossprobing enabled. You will run the custom DFM stream analysis.

1. Click **OK** to proceed.

2. Once CAM350 is started and crossprobing initiated, choose **Analysis** > **Streams Editor**. The Streams Editor pane opens.
3. The list of Active Streams will contain two streams. The first Stream titled as Stream 0 is the default stream. The second stream titled as Doc Editor Demo is the custom DFM Stream.
4. Select the custom stream and note the last two checks. These are custom outer signal layer and inner negative plane checks derived from design data in Allegro.
5. Uncheck Run for Signal 1 and Negative 0 checks to skip these checks that are duplicates of the custom check.
6. Select **Run Selected** > **Inside Board Outline Only** to initiate the analysis. Several errors should be detected. Choose **OK** to Switch to Error Explorer.
7. Close the Stream Editor Pane.
8. Arrange the Allegro and CAM350 applications on your display so that both are visible and sharing screen space.
9. In Error Explorer, select any error in the list and note the reaction of Allegro and CAM350 to center the view in both applications on the selected error. Also note the addition of a CAM350 specific error marker in Allegro.

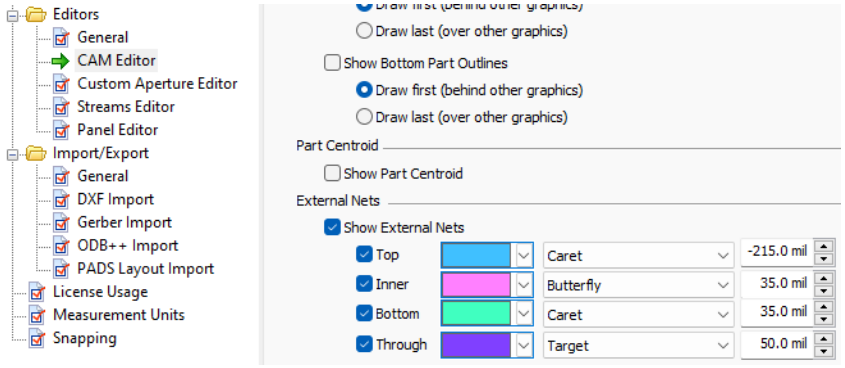
NETLIST COMPARE ENHANCEMENTS

New features were introduced to aid in the exploration of netlist compare errors. Enhancements were made to both Error Explorer and the display of nets involved in netlist compare errors.

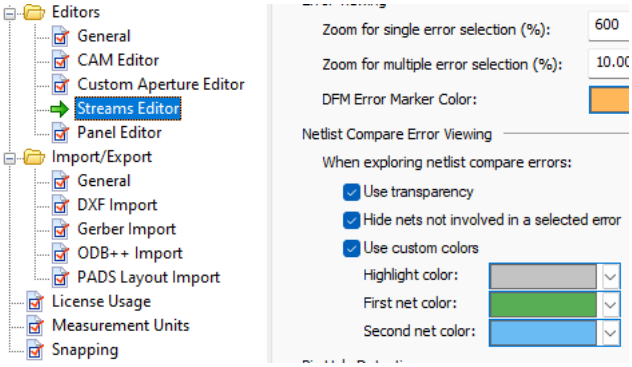
EXAMPLE – In this example, you will open a design that contains netlist errors and use the new features to explore the errors. First, let's review the new options for external nets and netlist compare.

1. Start CAM350 and Browse to and open the **open-shorts.c**am file.
2. Choose **Home** ribbon > **Options**.
3. Select **Editors** > **CAM Editor** options.
4. Scroll down until **External Nets** options are visible.

Here you will note the addition of viewing inner net points in support of newer ODB++ and IPC-2581 netlist standards. Additionally, the default graphics and colors for net points have been updated.



5. Select **Editors** > **Streams Editor** options.
6. Scroll down until the new **Netlist Compare Error Viewing** options are visible.



These new options were added to facilitate improved exploration of netlist compare errors. When you select a netlist error in error explorer, the display is temporarily updated until the error is unselected. The display of nets involved in an error is determined by three new options.

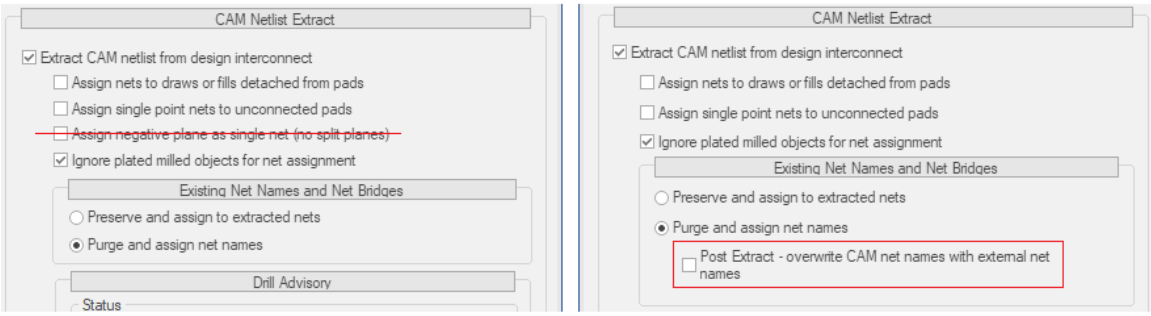
Use transparency – make all non-involved nets transparent.
Hide nets not involved in a selected error – hide all non-involved nets.
Use custom colors – replace current layer and highlight colors for involved nets with the assigned colors.

7. Select **Cancel** to close the Options dialog without making changes.
8. Select the **Nets** pane to make it active and note the addition of a **Show** column to control visibility of individual CAM nets.
9. On the **Nets** pane choose **View Nets** > **Hide All Nets**. This hides all design elements that belong to a CAM net. Note the presence of some partial net data. These pins are not associated with any CAM net.
10. On the **Nets** pane choose **View Nets** > **Hide No Net Elements**. The remaining elements are hidden.
11. On the **External Nets** pane, use **View Nets** to toggle the display of all external nets to Hide and then back to Show. Leave all external nets visible.
12. On the **Nets** pane choose **View Nets** > **Show All Nets** and uncheck **Hide No Net Elements**. All net and no net elements are made visible again.

Now let's experiment with the usage of these new options.

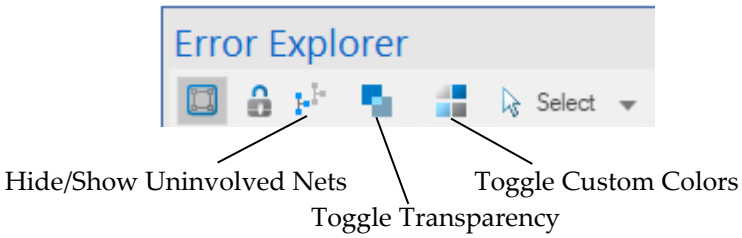
1. On the **Analysis** ribbon, choose **External Nets**.

There are changes to the CAM Netlist Extract options as highlighted below. The Assign Negative plane as single point net was removed. It was no longer necessary. A new Post Extraction option was added to overwrite CAM net names with external net names. This should be used whenever Net Bridges are present in your design. It will preserve external net name associations with the net bridges.

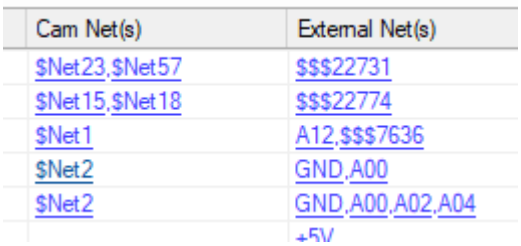


CAM350 Version 14.6 CAM350 Version 15.0

2. Disable **Compare Against External IPC-D-356 Netlist File** option.
3. Leave the remaining options unchanged and choose **Run Now** > **Inside Board Outline Only**. After a few moments, you are prompted that errors were found. Choose **OK** to make the Error Explorer pane active.
4. Close the External Nets pane.
5. In the **Error Explorer** pane, click error ID 0 to choose the **first error** in the list. Note how the display changes to use the custom color option.
6. On the **Error Explorer** pane, choose **Use Custom Colors** again to restore the previous state.
7. On the **Error Explorer** pane, choose **Hide Uninvolved Nets** and note how all nets disappear except those involved in the net.



8. On the **Error Explorer** pane, choose **Hide Uninvolved Nets** again to restore the previous state.
9. On the **Error Explorer** pane, choose **Use Transparency** and note how all nets are transparent except those involved in the net.
10. On the **Error Explorer** pane, choose **Use Transparency** again to restore the previous state.
11. Another improvement is the ability to zoom to a net involved in a netlist compare error. For the first error, click each individual hyperlinked CAM net name and External Net name. Note how the display is updated to show the selected net.

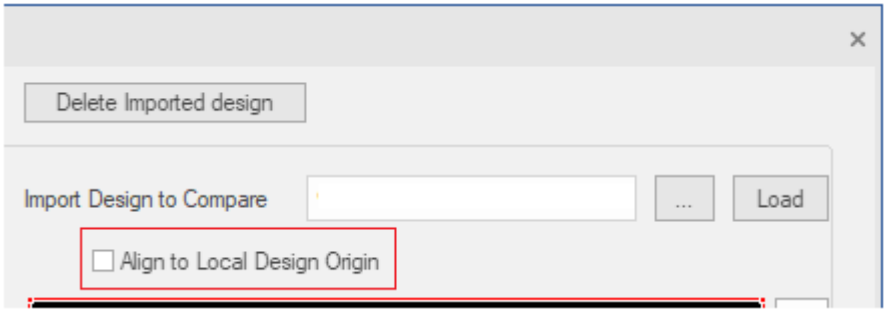


DESIGN COMPARE ENHANCEMENTS

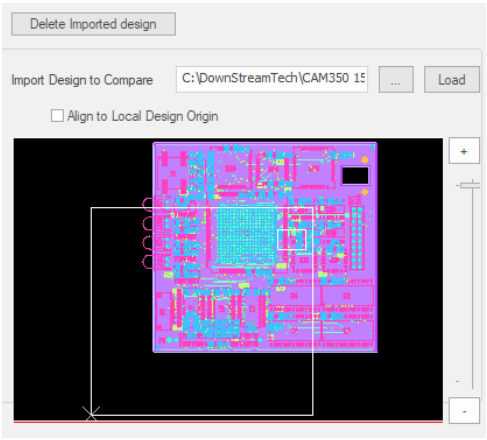
New features were added to Design Compare to expedite the compare process. Enhancements include automated alignment of the designs to compare and simplified layer mapping. In addition, Design Compare is now available as a licensed option that can be added to lower level CAM350 configurations.

EXAMPLE – In this example, you will open a design and compare it against another design and use new automatic design alignment and improved layer mapping features.

1. Start CAM350 and open the DST Demo 2018.cam file from the Demos folder.
2. On the **Analyze** ribbon, choose **Design** (Compare). The Design Compare pane appears.
3. Note the new **Align to Local Design Origin** option. For the purpose of this training, disable the option.



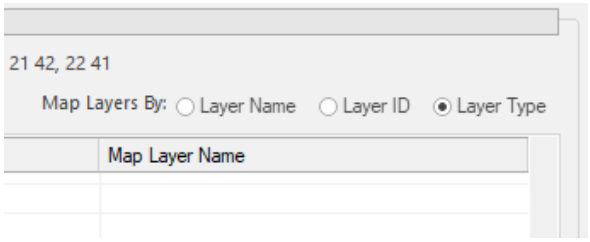
4. In the Import Design to Compare, choose Browse and browse to and select the DST Demo 2018 gerber.cam file from the same folder as the previously opened file.
5. After a few moments the design is opened and you will clearly see the design origins are different as evidenced by the board outline of the design in memory overlayed over the newly opened compare design.



6. Choose **Delete Imported Design** to purge the imported design from memory.
7. Enable the **Align to Local Design Origin** option and click **Load**. The design is loaded, but now the design is aligned to the local design origin.

Improvements were also made to layer mapping to make it easier to map layers for comparison between designs to compare.

1. Scroll down the Design Compare pane until the layer mapping area is fully visible.
   Hint: You may have to expand the size of the pane.
2. Note the new layer mapping preference choices of Layer Name, Layer ID or Layer Type.



These options are design to make layer mapping more expedient. In this design compare example, Layer type mapping does not work for all layers because of the variance in layer type assignment per design.

3. Scroll vertically the list of layers to map until you reach the top of the list. The default option is to map by layer type. Note that some, but not all layers are mapped. This is because the designs to compare did not have identical layer type.
4. Choose Map Layer by **Layer ID**. Note how the layer mapping instances improve somewhat, but the mapping is not correct because of differences in layer order.
5. Choose Map Layer by **Layer Name**. Note how the layer mapping is now empty because there are no matches between layer names in the designs to compare.
6. Choose Map Layer by **Layer Type** and we will use other features to map mismatched layers.

CAM350 15.0 BUILD 2075 DEFECT FIXES

73149	CAM350 DXF import – issue handing elliptical arcs corrected
73113	CAM350 DFM error count incorrect on certain designs due to a change in how CAM350 generates vias for padstacks. This was corrected.
73091	CAM350 Import DXF – This file imports with incorrect Dcodes and the re-export of DXF or Gerber results in unnecessarily large files
73077	CAM350 fails when importing this DXF file
73071	CAM350 fails when importing this ODB++ design

CAM350 15.0 BUILD 2067 DEFECT FIXES

72999	CAM350 15.0 Netlist Compare navigation features not working properly
72981	The older CAM350 macro language does not report Stream Errors
72979	When user deletes a layer, errors referring to layer should be removed as well
72971	Auto Detect Text Areas fails on this design
72943	DFM Analysis – Backdrills that are internal to the stackup are not being found during drill to copper checks
72924	Stencil Designer – Footprint is not available until the file is saved and reopened
72863	Mill Tab import failure
72860	CAM350 API – various defects and enhancements:
72859	DXF Import – Full circles imported as half circles for this DXF
72856	CAM350 API – Enh to Pause and Resume script playback
72855	CAM350 API – Enh to force redraw during VB script playback
72854	CAM350 API – CAM350 12.2 DXF Import dialog incorrectly appearing in 15.0
72851	Wheel Mouse Panning is different in CAM and World views
72846	CAM350 15.0 Japanese UI text corrections
72836	ODB++ Import – precision issue
72815	Draw to Flash – Interactive any-angle misses some rotated pads
72811	Draw to Flash – Conversion results in oddly rotated flashes for this design
72805	Draw to Flash – Fails to convert pattern when selected after a previous pattern conversion
72558	Analysis – Copper spacing check fails to find unplated drills shorted to copper on this design
72510	DXF Export – Failure on mirrored CAM350 data
72715	Panel Editor – panel exported to gerber and reimported has data inverted
72706	Circle Mill Paths – Incorrect display using OpenGL
72098	Custom Apertures: Pad Mask to Pad Ration failure

CAM350 15.0 BUILD 2059 DEFECT FIXES

72797	ODB++ Import – Copper voids missing after import for this Altium design
72787	Regression - File SaveAs sets the wrong CAM version in the CAM file.
72784	IPC2581 import missing internal pads for this design
72782	IPC2581 Import does not correctly mirror padstack element instances
72781	IPC2581 import failure on this Zuken CR8000 design with slot cavity
72774	API macro playback improvement for drill import
72749	Drill to Flash using “any angle” is missing some angles
72743	Install - DisableEnv not Getting Set Properly in CAM350 ini file
72742	ODB++ import failure for this design
72728	Stencil Designer – layer is sometimes not visible in this design
72718	DXF import failure for this design
72698	Stencil - Patterns being applied to previously completed pattern
72670	New Cadence interface improvement if there is data loaded already in the associated CAM350 window
72663	IPC-2581 import problem
72596	Oblong pads dropped on netlist extract
72542	Automatic mode in Stencil designer should sort with the highest "pin" (or pad) count to the top
71948	This design displays incorrectly using OpenGL
70895	ODB++ import - Pad lost

CAM350 15.0 BUILD 2053 DEFECT FIXES

Defect	Description
72658	Mill Circle – Add Mill Circle only adds ½ circle
72656	Allegro – CAM350 interface ODB++ does not work for filenames with embedded spaces
72654	Extra pad causing shorts in this IPC2581 rev C file from Allegro.
72630	CAM350 15.0 Wizard Cancel does not work
72595	Licensing updates for new 15.0 features
72591	Auto Save - Auto Options not working as expected
72581	File Open - Change default Database not recognized
72577	Allegro DXF incorrectly imported
72569	Stencil - Remove Mirror Settings for all Stencil Shapes
72566	Stencil - Internal Library list populated with External Library list
72564	Stencil - Remove Commands from Create Stencil From Library Item Dialog
72563	Stencil - Change Save to Library Buttons to Save, Close and Cancel
72562	Stencil - Add Prompt for Overwrite on Save To Library
72560	Stencil - DogBone Aperture (MELF/CPAD from Stencil) not visually correct in OpenGL
72556	OpenGL display is incorrect for pads display on this design
72529	OpenGL graphics are incorrect after Mirror Vertical operation
72539	Stencil - Change Shape to Thermal Leads to Crash
72538	Annular Ring - Minor Text UI Issue
72535	Stencil - Field Editing in non-units fields has unintentional cursor movement
72550	Align Netlist - ScreenTip on External Netlist is incorrect
72515	DXF – Failure on export for this design
72512	DXF – Import in append mode
72510	DXF Export fails on mirrored CAM350 data
72500	Failure in Draw to board outline
72494	ODB++ import misses PADS offset on this design
72493	Stencil - Interactive Select Library not Working
72488	Design Compare – File no longer allows mapping changes after Save
72478	Object snap does not work in Add Part command for CAM350 15.0
72473	ODB++ import - failure for rare data situation
72468	Issues when trying to create a d-pack from this data
72462	Cross Probing - PADS Layout and Xpedition - reconnect fails after Streams execution
72461	Enhancement to Via and Backdrill Templates – symmetric column creation should not align vias in adjacent layers

72460	Backdrill Stackup template drawing elements shift on execution
72448	Stencil - When you have one layer turned on before going into Stencil creation mode, it should default to that layer in the pulldown.
72469	Gerber Export default decimals should be 5
72462	CrossProbing for PADS and Expedition reconnect failure after Streams execution
72426	Enhancement to make ODB++ import (panel dialog) resizeable
72418	Stencil - DogBone aperture created with wrong shape for some parameters
72399	Mill Tab - Addition of Perforation to Mill Tab does not update Mill Tabs in Design
72394	PADS Import - incorrect component outline due to extremely small text height
72244	Stencil kit – Heel and Toe Settings not applied correctly to some pin patterns
72231	Design with rout compensation is incorrectly merged when imported with the merge cmd
72218	ODB++ Import - Drill layers missing on import for this old design file
72205	Auto Align - Auto Align Command Missing ScreenTip
72089	Enhancement to Pass parent PID for BluePrint and CAM350 to Winwrap BasicIDE.exe on command line for custom menu commands
72082	DXF import – some lines dropped on this DXF file
72069	Compare Results multiple selection is not working properly
72049	Print does not work on this design with duplicate layer names and layer types
72045	CAP library Delete all
71949	VB Recording - script is incorrect for adding layer sets
71854	IPC-2581 Rev C import does not support updated SlotCavity element
71799	Mill Data in merged databases displays incorrectly when using OpenGL
71641	ODB++ Import and then Save does not work for this design
71571	PADS Layout .REP files should not be selected for RS274X Gerber files
71545	OGL display issue with Mill Data on this design
71482	Component outline color is not displayed in OGL for this design
71353	SUV header column cleanup for usability
71352	Layer labels cleanup in the Stackup Preview window so they are not truncated
71345	SUV - Context menu commands in the Stackup Table disable the table
71335	SUV - Refresh issue on SUV grid in Autoimport mode when scrolling horizontally
71158	DXF Import - TTF fonts does not match AutoCAD size and position
70911	Enhancement to Query External Net - report layer and location for external net point
70623	IPC-2581 import - layer mapping for ALL LAYERS stackup not initialized
70514	Compare layers not working on this design data, but worked in CAM350 12.2
70421	Japanese version only - Negative Plane check does not work in DFM for this design
70404	VB Automation samples added to installation
70341	New Paste Mask spacing check to soldermask opening without paste
70310	File Auto Import command does not work for this design
70309	Macro Recording - Recording and execution of Streams Analysis does not work
70218	Auto hide stops working in error explorer after answering dialogue

70179	OpenGL display issue - Test points and Probes not visible for this design
70157	Clarification in user interface for "Paste Mask to Copper" check
70148	Bad mapped drive on PC causes CAM350 File open delay
70141	Streams paste checks not working on bottom paste only checks.
70099	IPC-2581 import - Invalid REV C warnings for IMPEDANCE and LogicalNet SPECREF definitions
70082	This old CAM database gets no results in streams
70043	Problem import Mill Data for this design
70010	DXF import - Failure, but works in earlier releases
69967	Cannot import stackup XML for this design
69934	Rename Layer - Usability improvement to to highlight old name
69924	Pad shift on netlist extract
69893	Draw to Flash behaves differently on different data types
69870	GerberX3 import improvement
69869	Route file will not load during autoimport, but loads via import nc mill data
69868	Gerber Import - arcs incorrect after import for this gerber
69855	IPC2581 import - failure, but imports in CAM350 12.2 with warning messages
69821	Draw to Flash failure on this data
69814	DFM Stream - Mask to Via check ignores layer stackup
69794	CAMSettings Import failure
69713	Draw to Flash - failure on this design with rounded shapes with small cuts in them
69568	DXF IMPORT support for version AC1032 - AutoCAD 2018,2019,2020
69567	ODB++ import - PAD dropped on layer signal_1 for this design
69549	CAM350 language versions – “Streams Errors Listed” must be in English text
69510	NC data with negative tool size
69324	CAP Editor - Edit features do not work if invoked from Panel Editor
69303	VB Script - Record and Playback for PCB Images table in Panel Editor incorrect
69232	Custom tools do not allow path specification for arguments and starting directory
69268	Export Gerber gives Web Browser ActiveX warning message
69214	Start Page links should bring up default browser, but always try to invoke IE right now
69154	Import Altium ODB++ - "the parameter is incorrect" on netlist compare
69000	Gerber to Mill (Panel) failure on this design
68941	Add Polygon - Hatched pattern preview does not match results
68921	Layer Compare fails on this design
68751	Enhancement to support Embedded Test Points support
68642	Mill Tab – BluePrint Conversion of Mill Tab to CAM350 is not Correct
68596	GerberX3 import improvement
68510	AutoImport failure
68464	Layers Pane - Disable of Auto-renumber does not work

68316	Streams VB execution results in different error count on playback
68091	Nelist Compare enhancement to cover unplated drills, drill slots and mill paths
67603	Draw 2 Flash not working correctly
66025	DXF import - Failure on this DXF with hatches
65514	Measure enhancement to add more info such as drill or pad size
65440	Composite to layer failure
64714	Auto-import - Multiple layer type assignment problems.
62733	Draw to Flash - Interactive does not work on circle
56294	Improve layer mapping in design compare

HOW TO CONTACT

Please send any defects, feedback or questions to support@downstreamtech.com.

PATENTS, COPYRIGHTS, AND TRADEMARKS

PATENTS

"AUTOMATED PCB MANUFACTURING DOCUMENTATION RELEASE PACKAGE SYSTEM AND METHOD", United States Patent No. 7,409,666 B2
"ADAPTIVE TEMPLATE SYSTEM FOR AN AUTOMATED PCB MANUFACTURING RELEASE PACKAGE SYSTEM", United States Patent No. 8,875,072 B2

COPYRIGHTS

DownStream Technologies, LLC, 290 Donald Lynch Blvd, Suite 300, Marlborough, Massachusetts 01752

TRADEMARKS

We have done our best to ensure that the material found in this publication is both useful and accurate. However, please be aware that errors may exist in this publication, and that neither the authors nor DownStream Technologies, LLC make any guarantees concerning the accuracy of the information found here or in the use to which it may be put.