

IDA Impedance Workflow

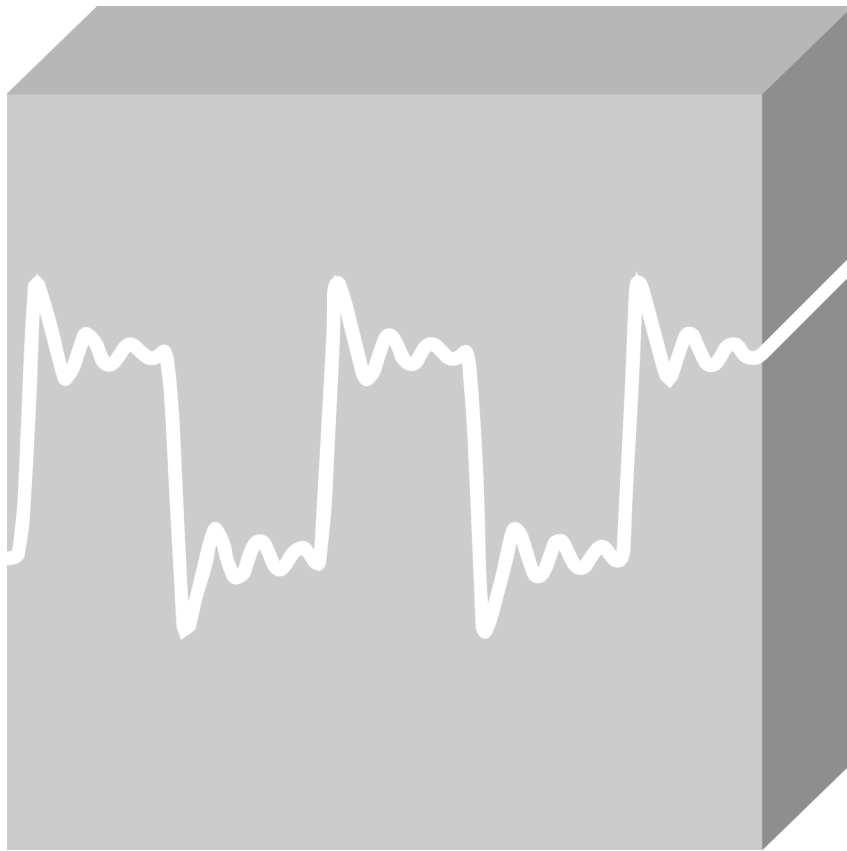


Table of Contents

1	Software Prerequisites	3
2	Starting Impedance Workflow	3
3	Performing Impedance Analysis.....	3
4	Specifying Directed Groups	9
5	Impedance Plots	11
6	References.....	12

1 Software Prerequisites

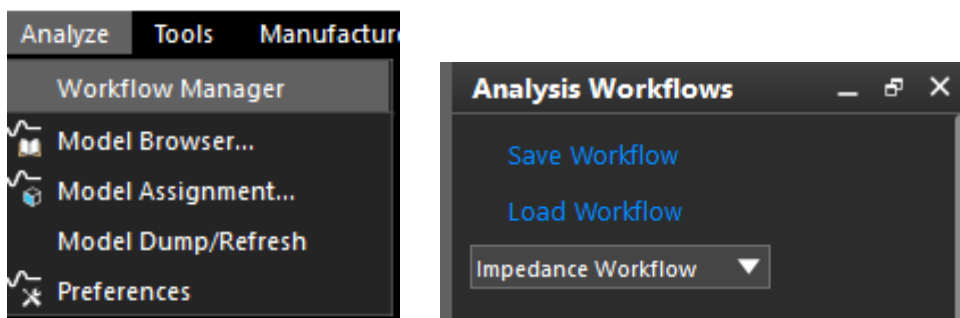
To use Sigrity Aurora, Allegro IDA, Topology Explorer, or SystemSI products from Allegro schematic or layout editors, you require both the following installations:

- Sigrity 2019 Hotfix 001 or higher
- OrCAD and Allegro Release 17.40-2019 QIR1 or higher
 - Install the latest versions of both products.

2 Starting Impedance Workflow

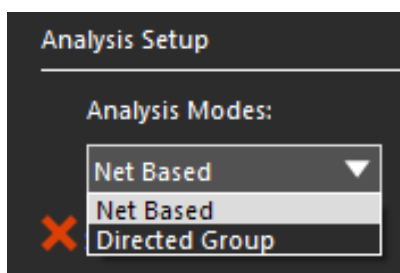
In OrCAD PCB Designer Professional, open the Workflow Manager by selecting **Analyze > Workflow Manager** from the menu. Make sure the **Impedance Workflow** is selected.

The Analysis Workflow Manager will walk you through the necessary steps and settings to perform the Impedance Analysis.



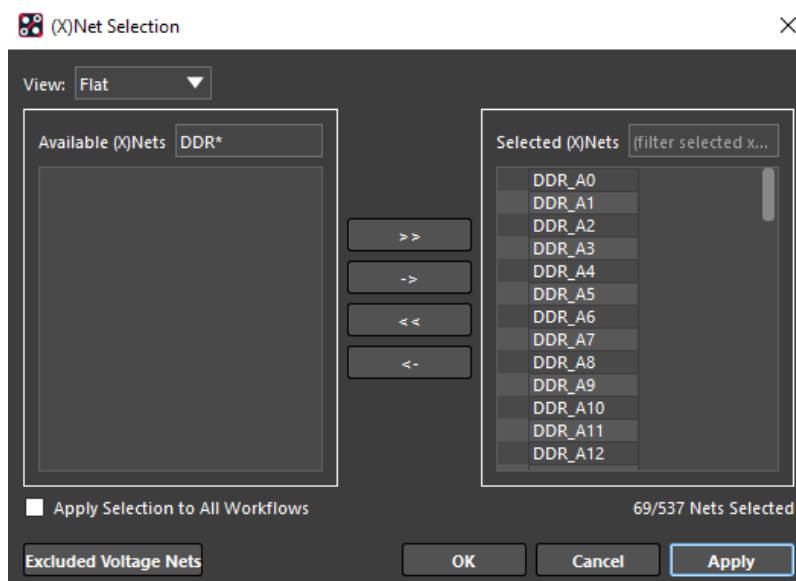
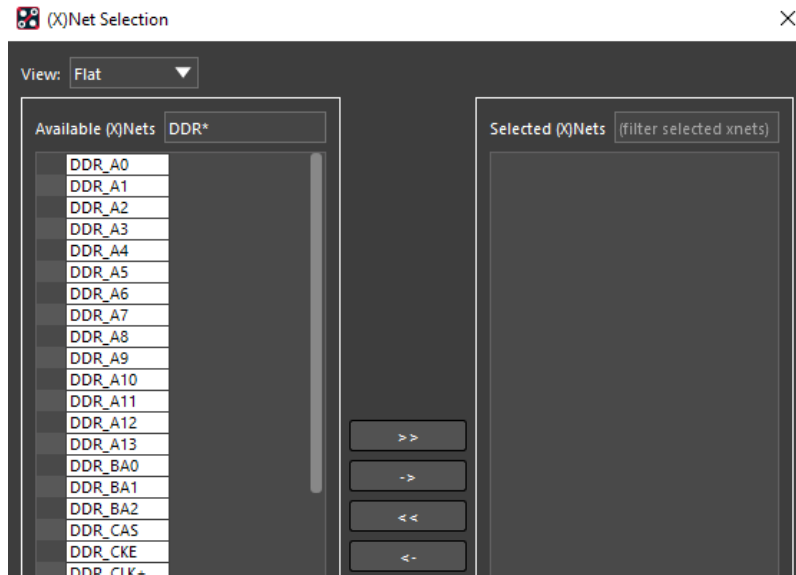
3 Performing Impedance Analysis

- Choose either Net Based or Directed Group from **Analysis Modes**. Net Based is chosen by default.

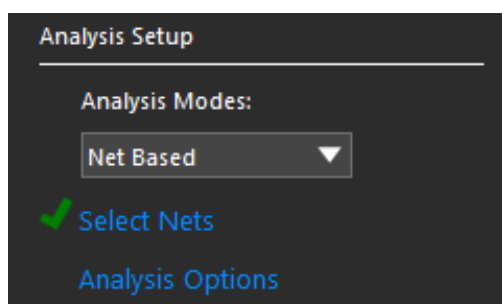


- Select the link for **Select Nets** nets or specify directed groups based on the analysis mode. In terms of setting up directed groups, you will find more information in section **Specifying Directed Groups**.
In the upper-left section of the dialog, notice the choice of **View**.
The **Flat** view lists all nets in the design in alphabetical order.
The **Hierarchical** view lists the nets under defined hierarchy objects, such as differential pairs, net groups, bus, XNets, or nets.

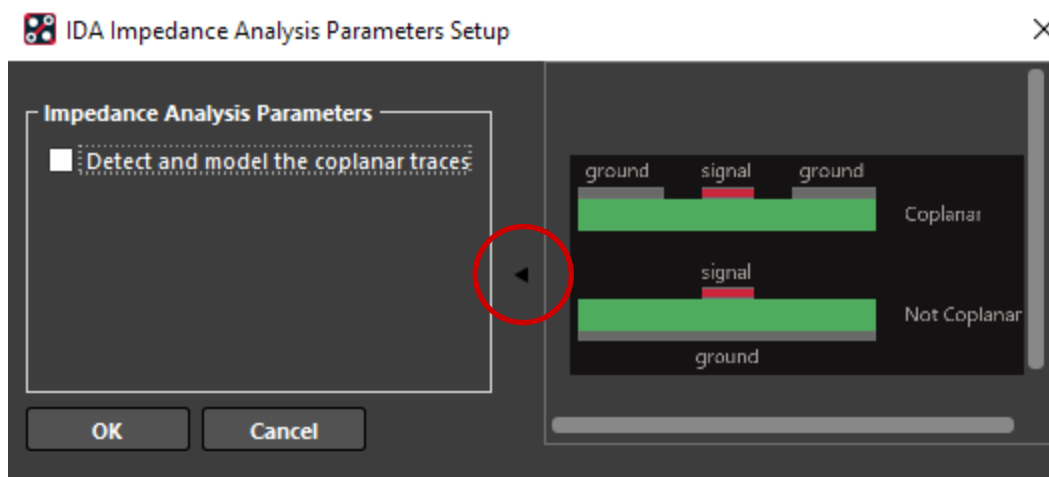
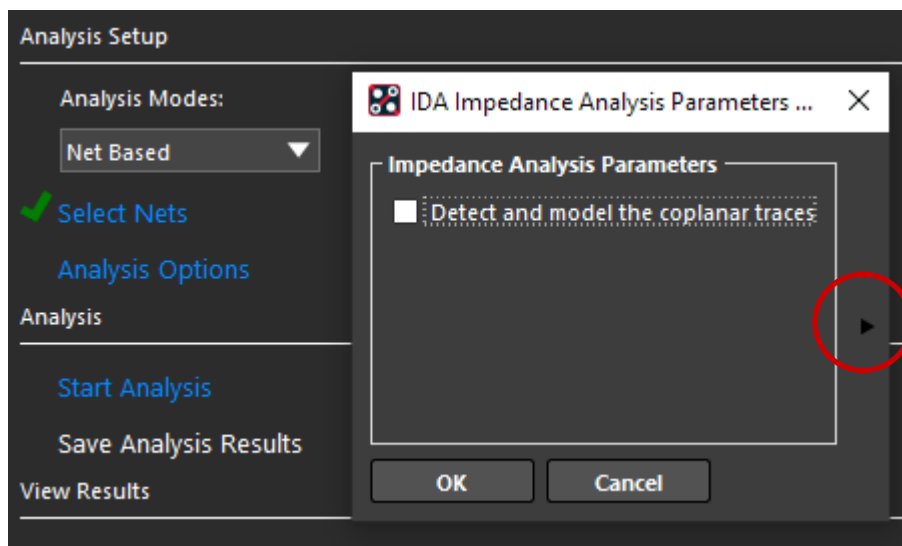
Use **Flat** when filtering by net name and use **Hierarchical** when selecting by Bus or Net Group.



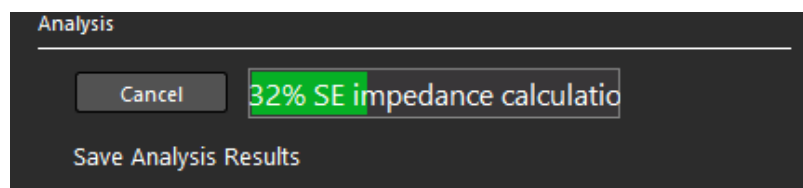
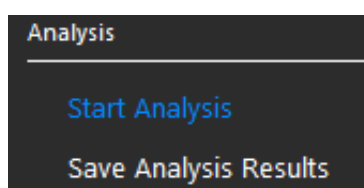
Notice that there is now a green check mark next to **Select Nets** and that the Start Analysis link is now active.



- To include coplanar traces in the simulation results, click **Analysis Options**, select **Detect** and model the coplanar traces.
- Click on the small arrow to open additional information.
- There is only one option here.
 - If you do not have coplanar structures in your design, you do not need to check this.

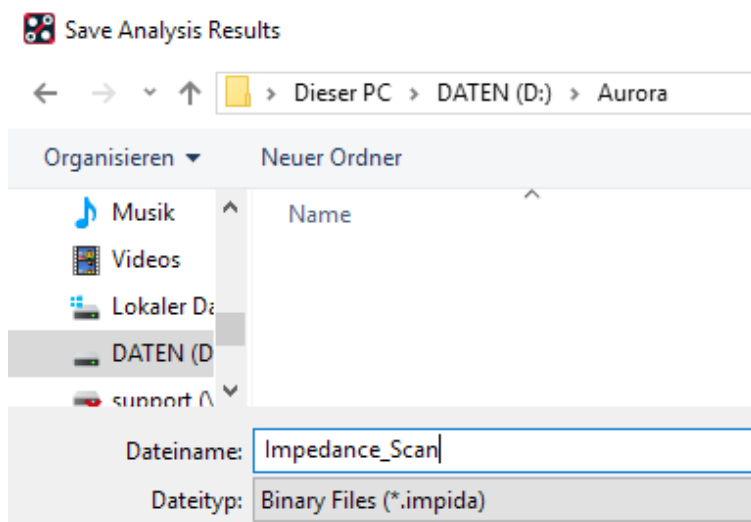
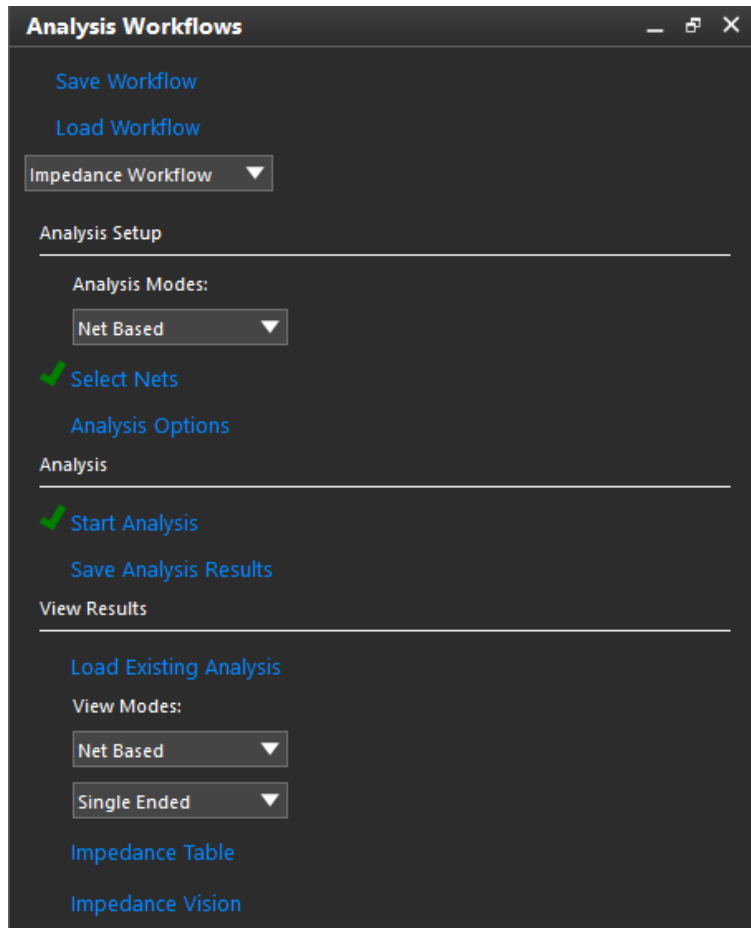


- Click **Start Analysis**. A progress meter will appear.



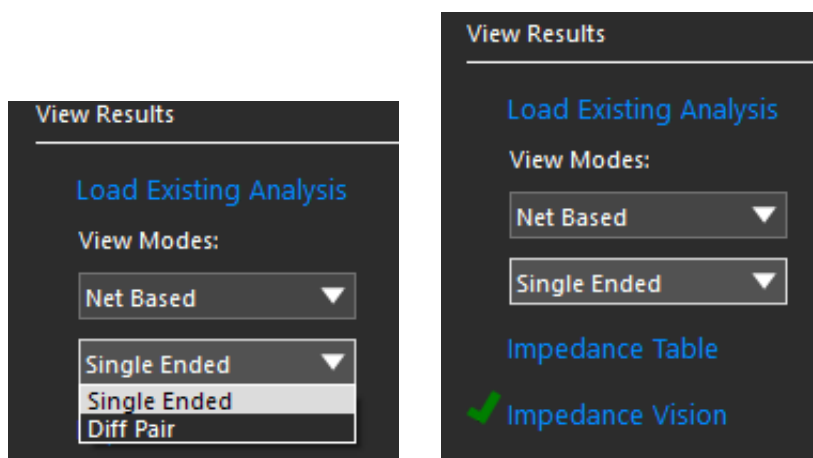
- When analysis is complete, a green check will appear for the Start Analysis task. If a previous analysis is stored, a prompt appears for overwriting the old analysis.

- The Save Analysis Results link also becomes active now.
- Click **Save Analysis Results** to save the results. Specify a name and location for the results file. The extension for the result file is .impida for impedance analysis.



Make sure the pulldowns in the **View Results** section of the workflow are set to **Net Based** and **Single Ended**, and select the Impedance Vision link.

- Specify **View Modes**. Choose:
 - Net Based** to view results for all segments of selected nets. Available for all analysis results.
 - Directed Group** to view results for segments defined in selected directed groups. Available only if analysis results contain directed group data.
 - Single Ended** to view single-ended analysis results for both single-ended and differential pair nets.
 - Diff Pair** to view differential pair analysis results for differential pair nets and single ended results for single-ended nets.



Note

Differential analysis results are reflected for coupled segments of differential pairs.

Click **Impedance Table** or **Impedance Vision** to view results.

- Impedance Table
 - Impedance Summary Table** lists summary impedance results for each net selected for analysis. Detailed Table lists the impedance results for each segment on the net selected in Summary Table.

Simulation Table

Single Ended

Summary Table

Net Name	Vias	No Ref	Impedance(Ohm)			Impedance Length(%)			Trace Total				
			Max	Min	Typ	Max	Min	Typ	Length	Delay(ns)	R(mOhm)	L(nH)	C(pF)
DDR_A0	4	0	63.40	52.10	57.30	3.24	47.53	48.11	2005.00	0.320	245.041	17.611	5.841
DDR_A1	3	0	63.40	57.30	57.60	7.21	33.20	59.60	2201.99	0.322	207.275	18.609	5.558
DDR_A2	3	0	63.40	57.30	57.60	6.27	33.54	60.20	2171.09	0.317	204.586	18.350	5.491
DDR_A3	3	0	63.40	57.30	57.60	8.88	33.47	57.65	2256.64	0.330	213.585	19.113	5.692
DDR_A4	3	0	63.40	57.30	57.60	9.73	34.04	56.23	2334.44	0.341	222.195	19.799	5.887
DDR_A5	4	0	63.40	52.10	52.10	7.69	56.39	56.39	1980.13	0.314	232.819	17.213	5.765
DDR_A6	4	0	63.40	52.10	52.10	13.28	42.89	42.89	2224.06	0.350	263.807	19.509	6.307
DDR_A7	4	0	63.40	52.10	57.30	9.66	42.85	46.52	2272.39	0.360	273.664	19.974	6.497
DDR_A8	3	0	63.40	57.30	57.60	6.16	31.19	62.65	2090.70	0.305	193.681	17.610	5.269
DDR_A9	4	0	63.40	52.10	52.10	9.82	46.69	46.69	2176.65	0.344	259.558	19.066	6.246
DDR_A10	3	0	63.40	57.30	57.60	3.50	32.84	63.67	2058.88	0.301	192.038	17.348	5.217
DDR_A11	3	0	63.40	57.30	57.60	5.35	34.06	60.59	2165.62	0.317	204.476	18.300	5.485
DDR_A12	3	0	63.40	57.30	57.60	11.81	26.15	61.94	2028.37	0.293	183.123	17.052	5.045

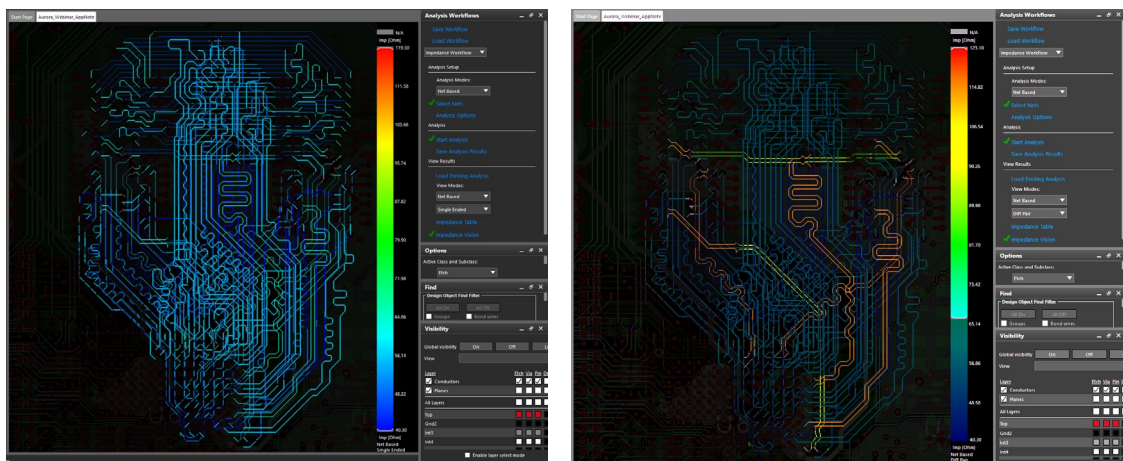
Detailed Table

Imp(Ohm)	Length	Trace Delay(ps)	Layer	R(mOhm)	L(nH)	C(pF)	Location (cyl)
63.40	4.40	1.20	TOP	2.818	0.278	0.069	(1481.42 3089.5...
63.40	9.08	2.30	TOP	0.802	0.079	0.020	(1475.00 3096.0...
63.40	16.38	2.30	TOP	1.447	0.143	0.036	(2148.92 3089.5...
63.40	7.67	1.10	TOP	0.677	0.067	0.017	(2143.50 3095.0...
57.60	0.01	0.00	TOP	0.001	0.000	0.000	(1690.13 2176.0...
57.60	22.20	3.10	TOP	1.569	0.178	0.054	(1690.13 2176.0...
57.30	157.50	25.30	INT4	21.679	1.447	0.441	(1795.50 3122.0...
57.30	59.60	8.10	INT4	6.964	0.464	0.141	(1993.00 3122.0...
57.30	32.53	5.20	INT4	4.477	0.298	0.091	(2003.60 3122.0...
57.30	112.90	18.10	INT4	15.540	1.035	0.315	(2026.60 3099.0...
57.30	5.66	0.90	INT4	0.779	0.052	0.016	(2139.50 3099.0...
57.30	8.82	1.40	INT4	1.215	0.081	0.025	(1475.00 3096.0...

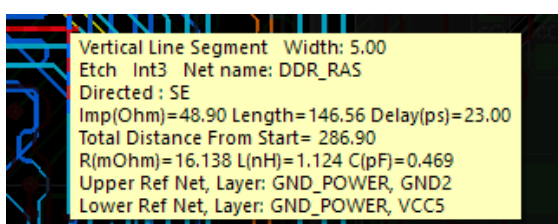
Note

To sort data in the table, click any column header.

- Select rows in the Detailed Table section to zoom / center the canvas on the impedance variance for investigation.
- If view mode is Single Ended, a Single Ended tab appears at the top of the summary table and all net results appear in the summary table.
- If view mode is Diffpair, both the Single Ended and Diffpair tabs appear at the top of the summary table. Select the Single Ended tab to view single ended nets or select the Diffpair tab to view differential pair nets.
- Impedance Vision
 - Clicking **Impedance Vision** displays color-coded impedance segments in the canvas.
 - The color coding is based on the segment or sub-segment impedance value as mapped into the impedance gradient, which is the continuous transition of colors from the minimum to maximum range of impedance values obtained by analysis.
 - Any nets part of the analysis results which have segments not associated with impedance analysis data will be displayed in the default color based on layer / net color assignments. Any nets which are not part of the analysis are dimmed in the default color.
 - The **Single Ended** view mode setting displays all Singled Ended impedance result for the selected nets; both single ended nets and differential pair nets of the results use single ended impedance data to color segments and for data tips.
 - The **Diffpair view** mode setting displays Single Ended impedance data for all single ended nets and all Diffpair impedance data for all differential pair nets.

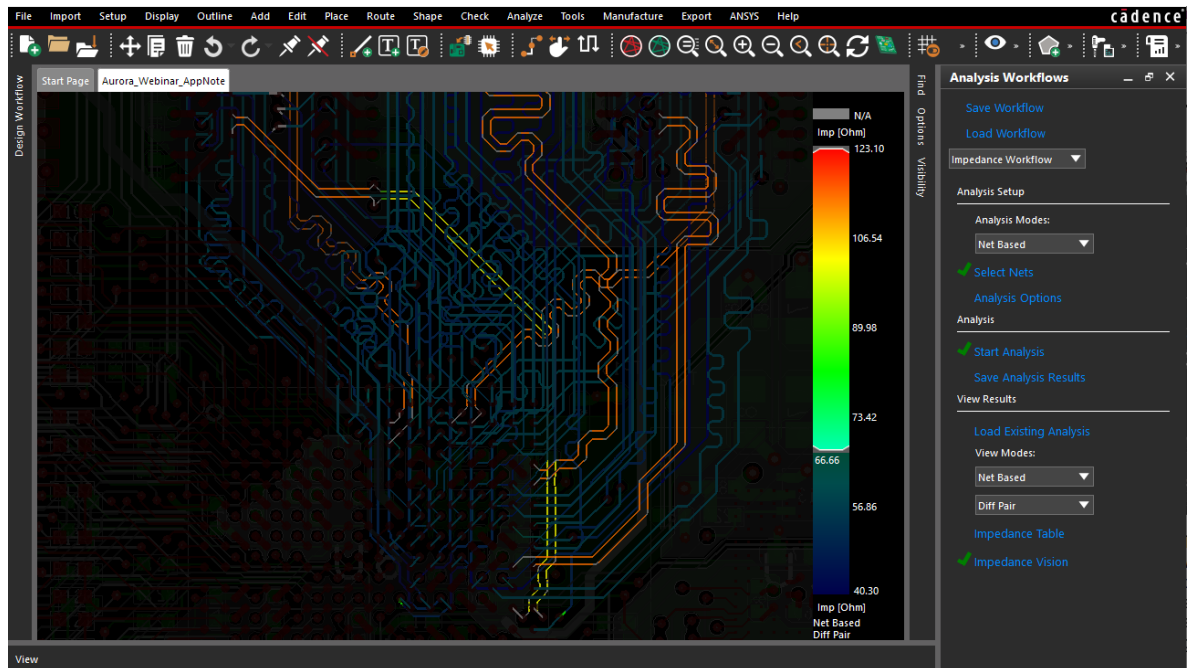


- Zoom in on a section close to the controller and use tooltips to investigate.



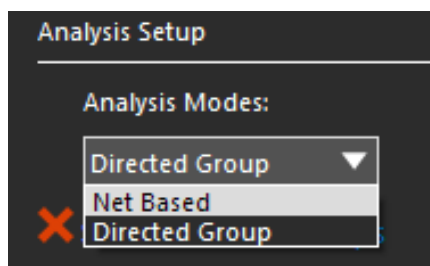
– Slider

The sliders on the color gradient allow for an easy way to filter what is seen in the vision. Also note the grey color at the top which indicates traces with no reference found in the stack-up.



4 Specifying Directed Groups

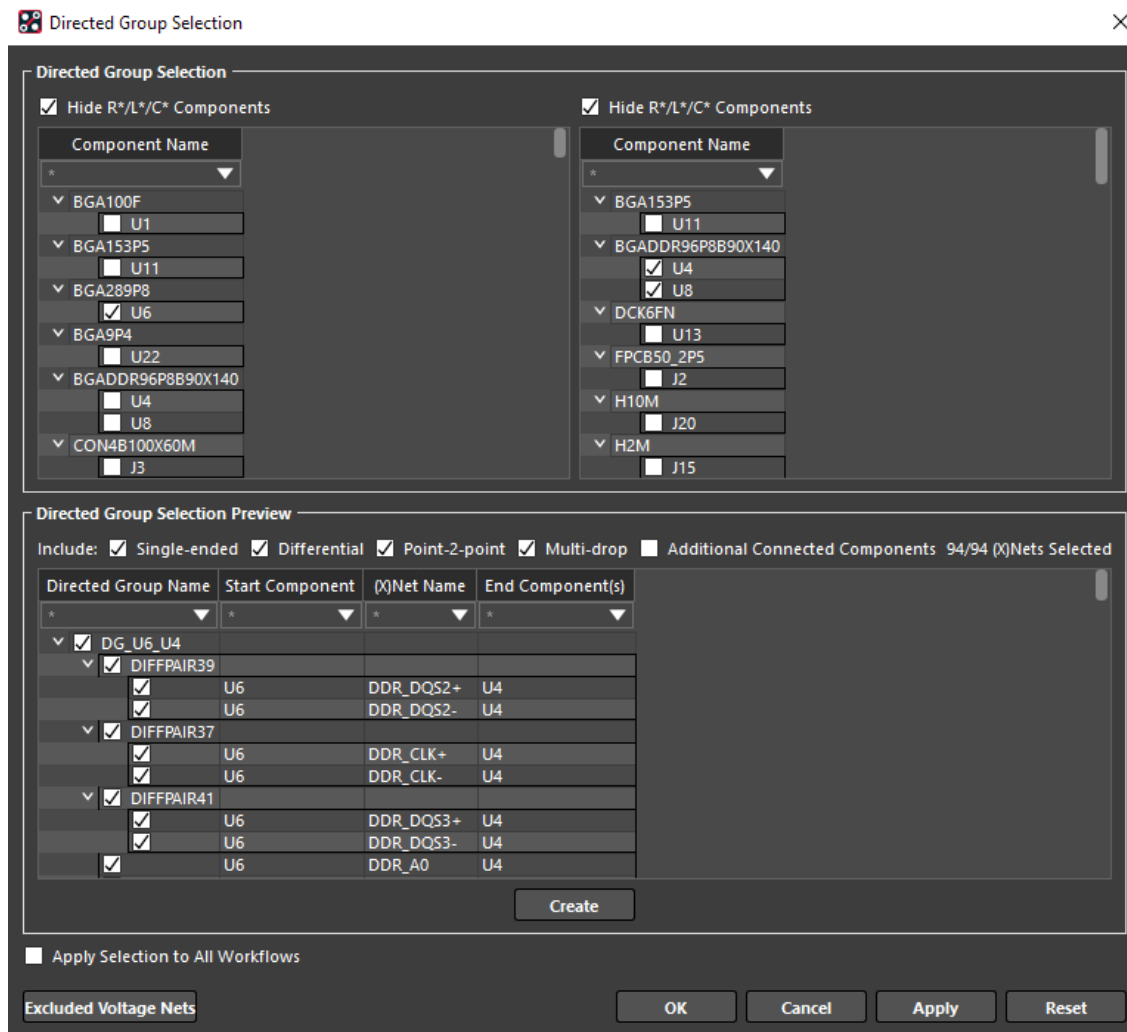
- In directed groups or component-based net selection, you can select an entire interface based on component connectivity rather than each net name.
- Directed groups help prioritize issues; for example, issues found near the start point as compared to issues found near the end point.
- In addition, directed groups let you associate specific analysis results for a collection of nets as encountered along the same time-line as progressing from the start to the end point.
- If analysis mode is set to Directed Group, the Select Directed Group flow step is presented and marked with a cross to indicate that the step must be completed.



- To select, create, or modify directed groups or modify a selection; click **Select Directed Group**. This opens the Directed Group Selection dialog box. You can select the directed groups from the canvas or using the dialog box.

A directed group is the portion of a net between a starting component and one or more ending components.

- The **Directed Group Selection** Preview of the dialog will show the directed groups selected from the canvas. The top section, Directed Group Selection Results, lists the selected directed groups. A directed group is the portion of a net between a starting component and one or more ending components.
- Use the expandable Create Directed Groups section to create directed groups.

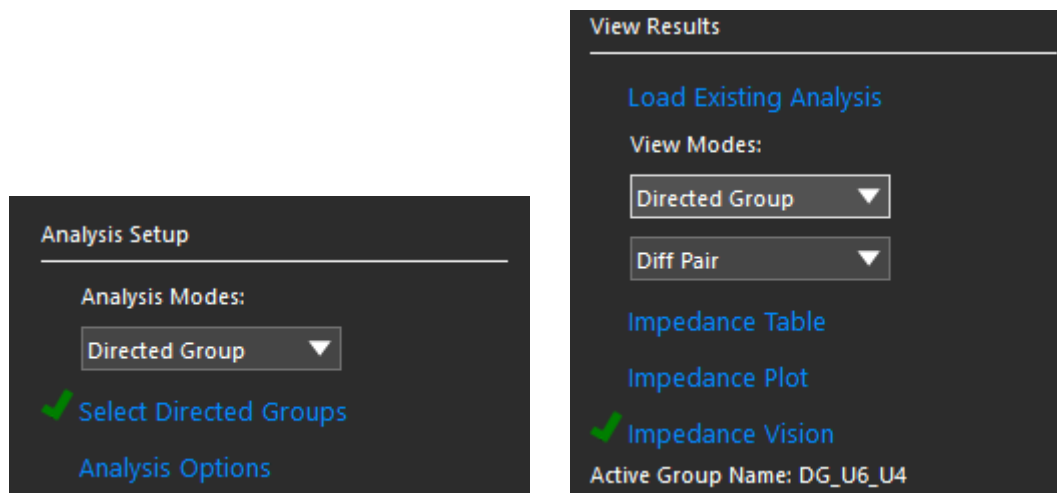


- To create a directed group, do the following steps:
 - Select a start component either on the canvas or from the list of components in the Start Component box or on the canvas. When you click a start component, all possible end components are visible while the other components are dimmed on canvas. The valid end components are listed under End Component.
 - Select one or more end components. A preview of the directed group is shown in the bottom section. All nets are selected by default. You can filter the list to display selected nets.
 - If needed, right-click in the preview area and choose any one of the options to select nets based on any filters.

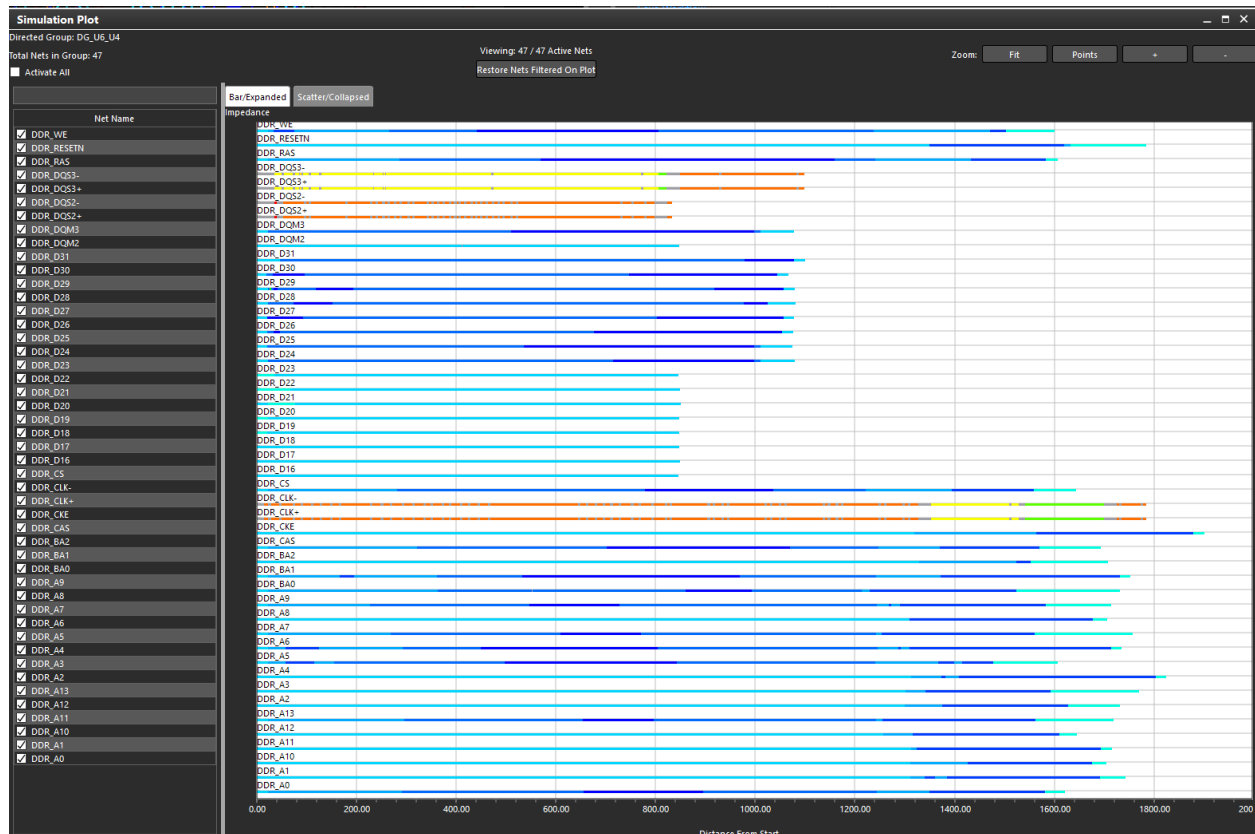
- The nets that are displayed based on the filter settings are the visible nets. In the example image, the filter is set to * to select all nets.
- Click **Create** to create the directed group. The group is created and selected by default.
- To select available directed groups or modify a section, do the following steps:
 - Select listed directed groups from the Directed Group Selection Results section. The selected groups are marked by a tick in the box on the left.
 - Click **Apply** to update the design or click **OK** to update the design and close the dialog box. **Select Apply Selection to All Workflows** to apply the directed groups selection to all the analysis flows that require or can use directed groups. The Select Directed Groups step has a green check and you can proceed with the rest of the impedance and coupling workflow steps.

5 Impedance Plots

Impedance Plot is available only for directed groups and shows results as bar (expanded) and scatter (collapsed) plots using the distance from start component on the x-axis as determined in the Directed Groups definition.



- Clicking **Impedance Plot** opens the dockable Simulation Plot pane, which has two tabs, Bar / Expanded and Scatter / Collapsed.
- The Bar / Expanded tab can be filtered using the options on the right while the Scatter / Collapsed tab offers a global view and can be used to quickly identify outliers.
- Both tabs support data tips.
- A net selection panel on the left lists all the nets in the active directed groups. All nets are selected by default. You can choose to select only specific nets.



6 References

- Cadence Allegro User Guide: Routing the Design, Product Version 17.4-2019, October 2019, Last Updated: June 2020
- Cadence Impedance and Coupling Analysis, Rapid Adoption Kit (RAK), Product Version SPB 17.4-2019 February, 2020
- Cadence Analysis Workflows – Impedance Analysis, Rapid Adoption Kit (RAK), Product Allegro Sigrity SI 17.4 or Allegro PCB Designer 17.4, July, 2020