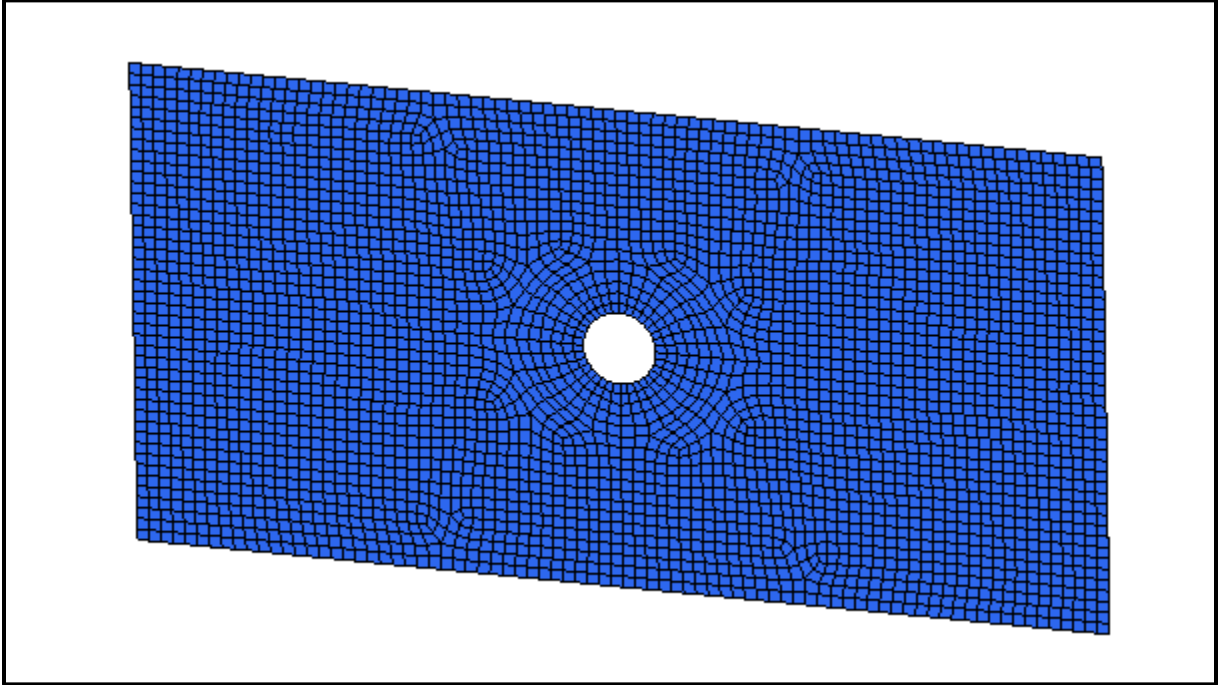


Exercise 3E: Simulating a Plate with Hole Test Coupon

This exercise requires the user to set up, load, and analyze a standard plate coupon with a hole in the center. The analysis will require users to create everything necessary for the analysis in the HyperMesh Desktop environment except for the mesh and material properties.

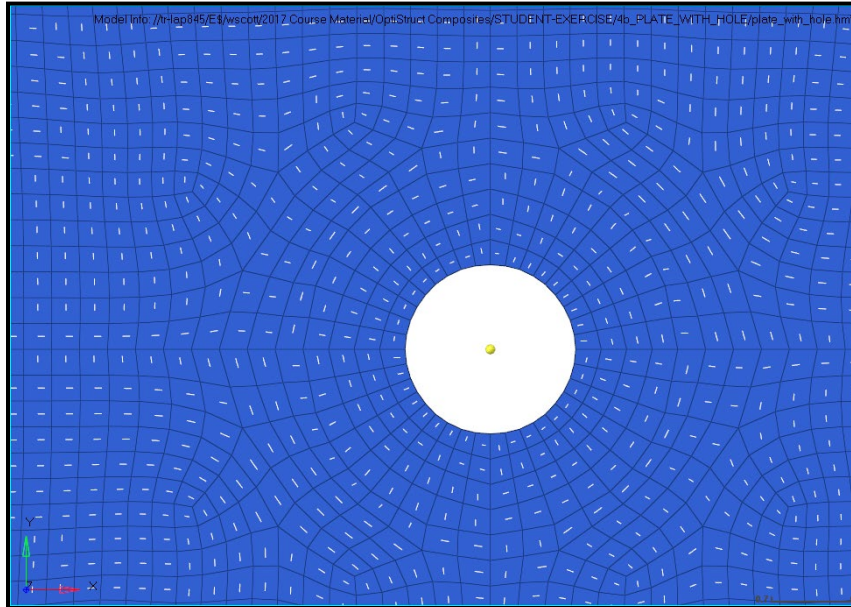


Problem Setup

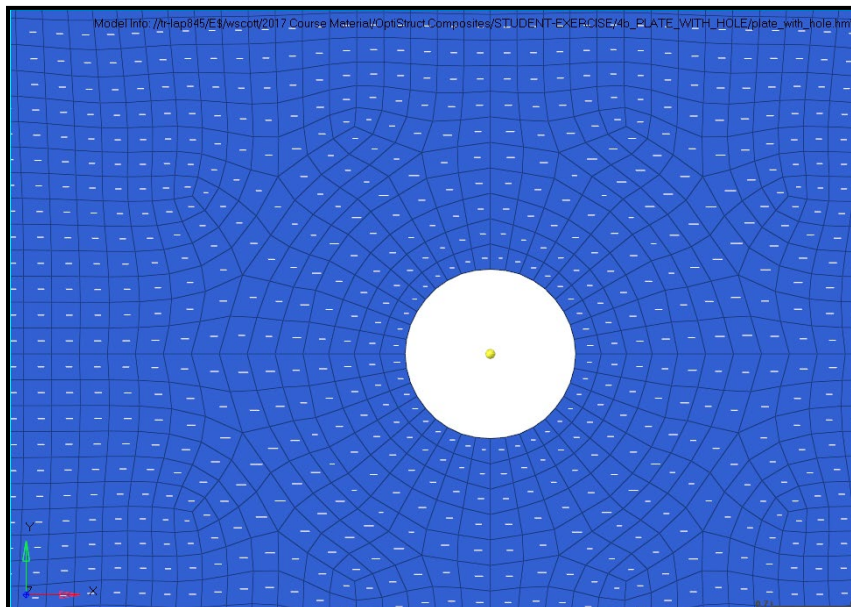
You should copy the file: `plate_with_hole.hm`

Step 1: Open the model in HyperMesh Desktop with the OptiStruct user profile

Step 2: Review the element orientations









Step 3: Update the `plate_with_hole` element orientations to align with the global X-axis



Tip: If using the material orientation subpanel, set the orientation method to by system axis using the local 1-axis and select the global system in the lower left hand corner of the graphics area to align all elements orientation with the global X.

Step 3: Create a new PCOMPP property card with the following properties and assign the property to the `plate_with_hole` component

Name	Value
Solver Keyword	PCOMPP
Name	PCOMPP
ID	1
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	PCOMPP
User Comments	Hide In Menu/Export
<input checked="" type="checkbox"/> Z0 OPTIONS	REAL
Z0	
NSM	
SB	
FT	
TREF	
<input checked="" type="checkbox"/> GE_USEMAT	
GE	
PCOMPX	

Name	Value
Name	plate_with_hole
ID	1
Color	
FE style	
Geometry style	
Include	[Master Model]
Property	(1) PCOMPP
Material	<Unspecified>

Step 4: Create six plies which use all elements in the model as the ply shape

Tip: Ensure that `Output results` is checked.


Name	Material	Thickness	Orientation	Color
Zero_Bottom	mat8	0.05	0	Blue
Forty-five	mat8	0.05	45	Yellow
Ninety_Bottom	mat8	0.05	90	Red
Negative Forty-five	mat8	0.05	-45	Green
Zero_Top	mat8	0.05	0	Purple
Ninety_Top	mat8	0.05	90	Orange

Tip: Since the ply shapes are identical for all plies, a single ply can be created, duplicated 5 times, and each ply edited to the proper name, color, & orientation.

Step 5: Create a new laminate, stacking the plies in the following order

[illegible]

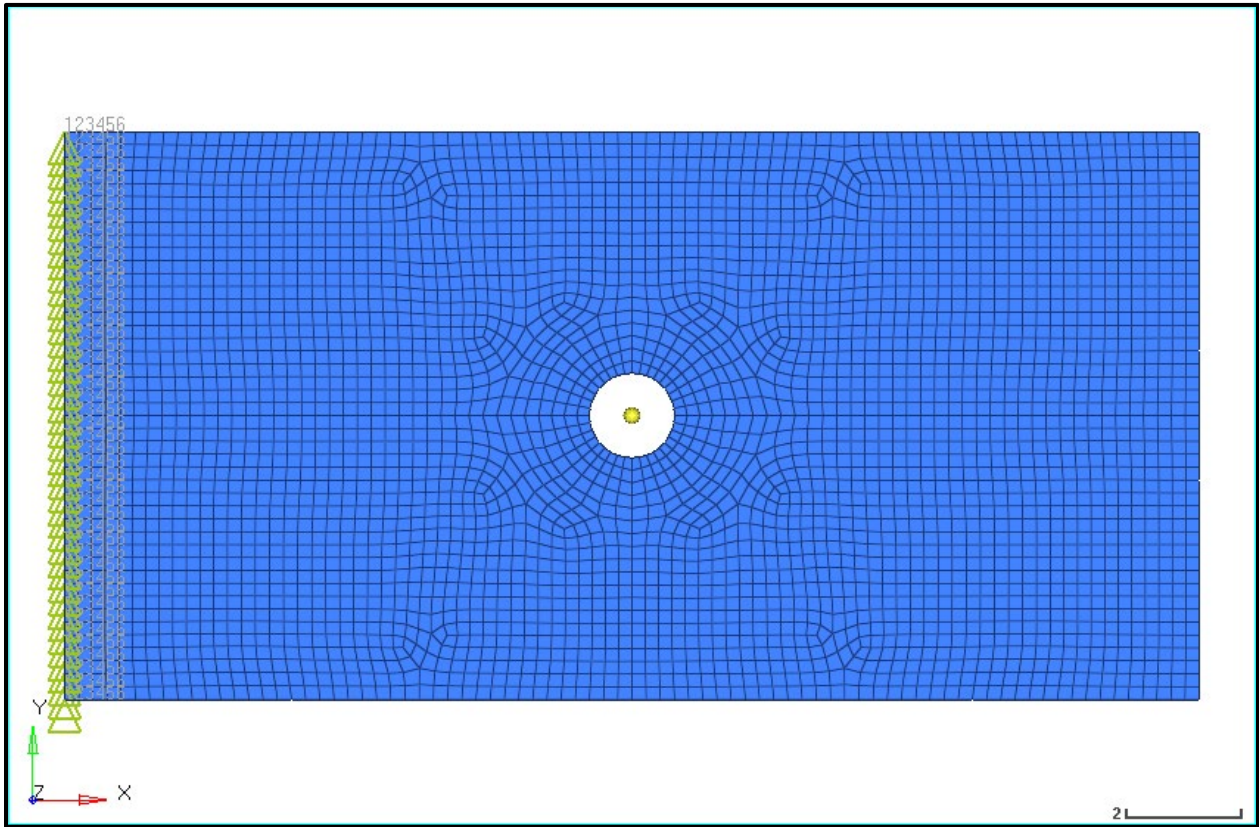
Step 6: Create a new load collector named SPC

Name	Value
Name	SPC
ID	1
Color	
Include File	[Master Model]
Card Image	<None>

Step 7: Create SPC constraints in DOFs 1-6 along the X- edge of the plate

1. In the menu, select **BCs > Create > Constraints** to enter the **constraints** panel.

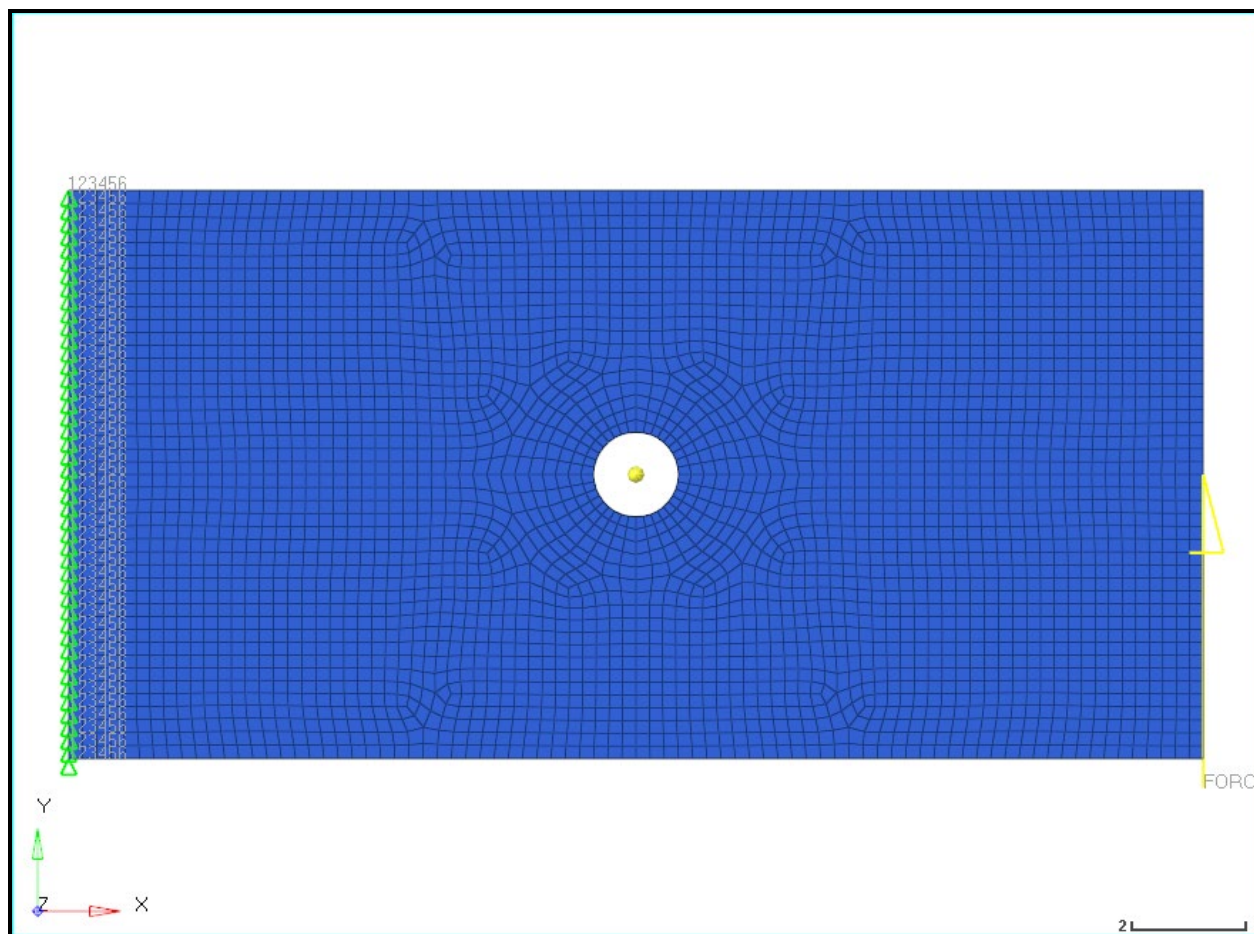
2. With the **nodes** entity selector active, select the column of nodes along the X⁻ edge of the mesh.
3. Ensure that all of the DOF check boxes are marked and click **create**. Click **return** to close the constraints panel.



Step 8: Create a new load collector named **Force**

Step 9: Create a force of 10 units in the Y-axis direction on node 2111

1. From the menu, select **BCs > Create > Forces** to enter the **forces** panel.
2. With the **nodes** entity selector active, select node 2111 using the **by id** option.
3. Set the **system** toggle to **global system**, the **magnitude** to 10, and the **direction** dropdown to **y-axis**.
4. Click **create** to create the force. Click **return** to close the **forces** panel.



Step 10: Create a loadstep of Analysis type Linear Static named Vertical with the SPC set to the SPC load collector and LOAD set to Force load collector

Name	Value
Solver Keyword	SUBCASE
Name	Vertical
ID	1
Include	[Master Model]
User Comments	Hide In Menu/Export
Subcase Definition	
Analysis type	Linear Static
SPC	(1) SPC
LOAD	(2) Force
SUPPORT1	<Unspecified>
PRETENSION	<Unspecified>
MPC	<Unspecified>
DEFORM	<Unspecified>
STATSUB (PRELOAD)	<Unspecified>
STATSUB (PRETENS)	<Unspecified>
SUBCASE OPTIONS	
LABEL	<input type="checkbox"/>
SUBTITLE	<input type="checkbox"/>
ANALYSIS	<input checked="" type="checkbox"/>
TYPE	STATICS
ASSIGN	<input type="checkbox"/>
EIGVRETRIEVE	<input type="checkbox"/>
EIGVSAVE	<input type="checkbox"/>
ENDLOAD	<input type="checkbox"/>
POST	<input type="checkbox"/>
RADSND	<input type="checkbox"/>
RESVEC	<input type="checkbox"/>
SUPER	<input type="checkbox"/>
SUBMODEL	<input type="checkbox"/>
GLOBSUB	<input type="checkbox"/>
SOLVTYP	<input type="checkbox"/>
RGYRO	<input type="checkbox"/>
TEMP	<input type="checkbox"/>

Step 11: Enable output of composite strain and composite stress results

1. From the menu, select **Tools > Create Cards > C > CSTRAIN (OR)** to enable the Control Cards section of the Entity Editor.
2. In the section for **CSTRAIN**, set **FORMAT** to H3D, **TYPE** to ALL, and **OPTION** to ALL as shown below.

Name	Value
CMKE	<input type="checkbox"/>
CMSE	<input type="checkbox"/>
CONTF	<input type="checkbox"/>
<input checked="" type="checkbox"/> CSTRAIN	<input checked="" type="checkbox"/>
<input type="checkbox"/> CSTRAIN_NUM =	1
<input checked="" type="checkbox"/> GLOBAL_OUTPUT_REQUEST 1	
FORMAT	H3D
TYPE	ALL
EXTRA	
NDIV	1
OPTION	ALL
CSTRESS	<input type="checkbox"/>
DAMAGE	<input type="checkbox"/>
DISPLACEMENT	<input type="checkbox"/>
DRESPONSE	<input type="checkbox"/>
DSA	<input type="checkbox"/>

3. Check the box for **CSTRESS** and set **FORMAT** to H3D, **TYPE** to ALL, and **OPTION** to ALL.

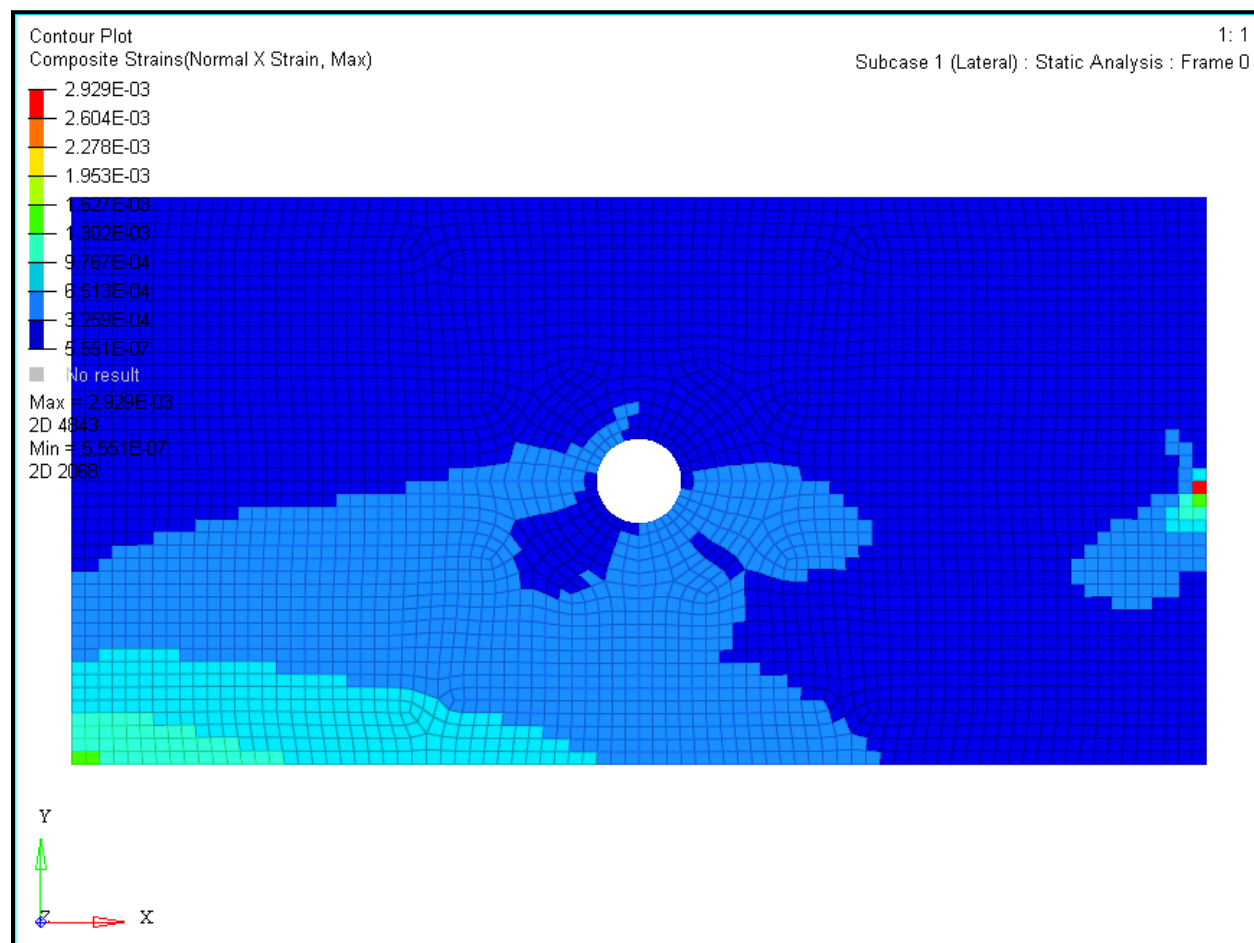
Name	Value
CONTF	<input type="checkbox"/>
<input checked="" type="checkbox"/> CSTRAIN	<input checked="" type="checkbox"/>
<input type="checkbox"/> CSTRAIN_NUM =	1
<input checked="" type="checkbox"/> GLOBAL_OUTPUT_REQUEST 1	
FORMAT	H3D
TYPE	ALL
EXTRA	
NDIV	1
OPTION	ALL
<input checked="" type="checkbox"/> CSTRESS	<input checked="" type="checkbox"/>
<input type="checkbox"/> CSTRESS_NUM =	1
<input checked="" type="checkbox"/> GLOBAL_OUTPUT_REQUEST 1	
FORMAT	H3D
TYPE	ALL
NDIV	1
OPTION	ALL
DAMAGE	<input type="checkbox"/>

Step 12: Save the model as `plate_with_hole_analysis.hm` and run in OptiStruct

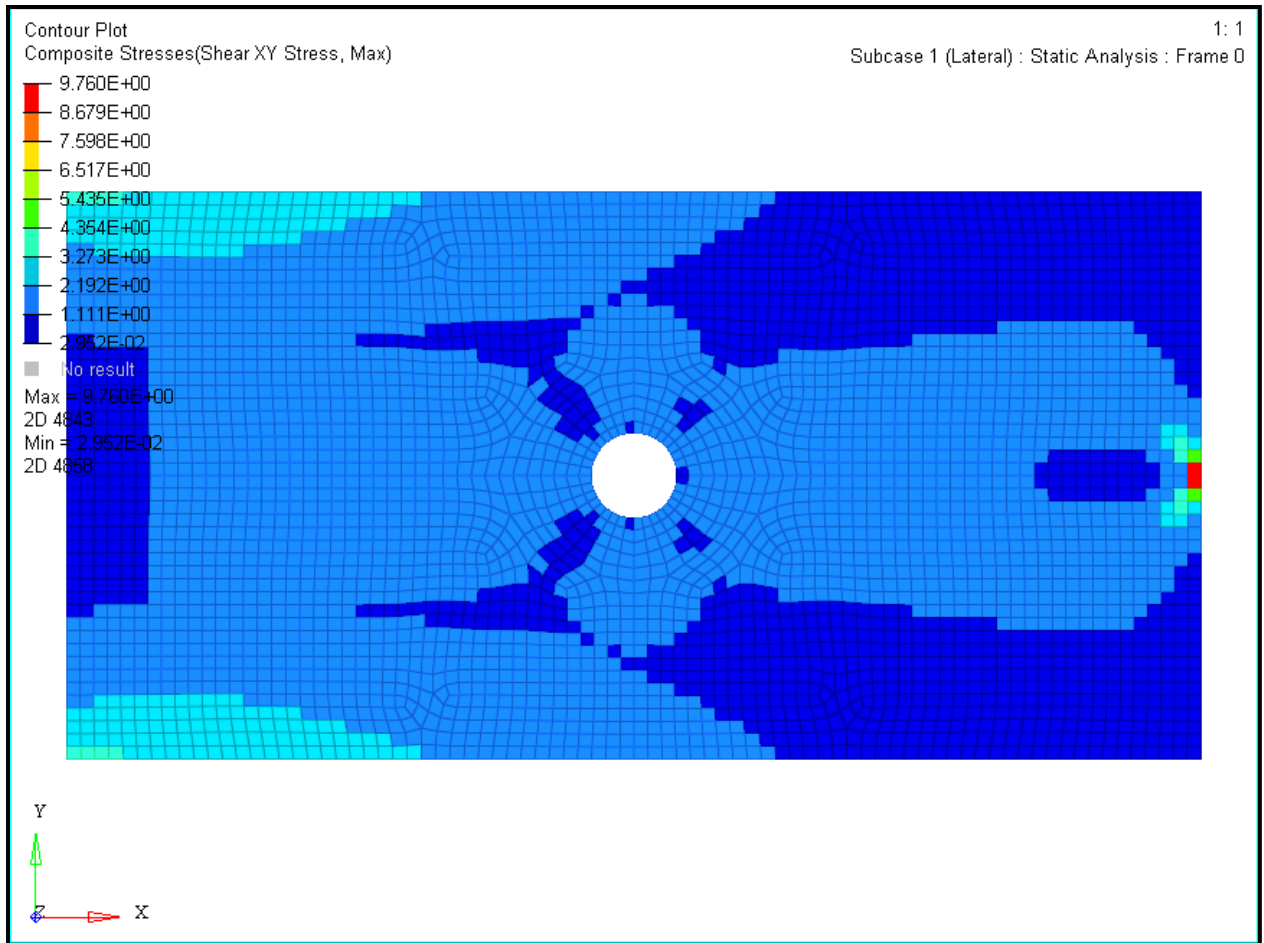
Tip: Menu option *Optimization > Optistruct* enters the **Optistruct** panel. Set *run options:* to `analysis`. Clicking **OptiStruct** will export the saved model and launch the solver in a single step.

Step 13: Post-process the stress and strain contours from the plate analysis in HyperView

EXERCISE RESULTS: plate_with_hole_analysis.h3d



Normal X Strains of plate_with_hole_analysis.h3d



Shear XY stress (Max Layers) of plate_with_hole_analysis.h3d