

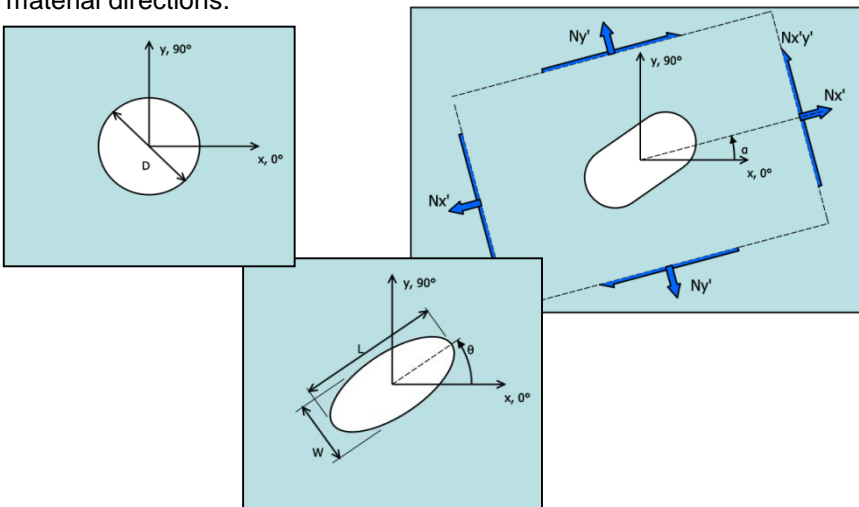


## Smart Application: Laminate Cutout Analysis Tool

LCAT computes the strain distribution around a circular, elliptical, or racetrack cutout in a composite plate subjected to biaxial and shear loading. Additionally, LCAT determines the **margin of safety for strain** and the **orthotropic strain concentration factor**. From the simple and intuitive interface, LCAT automatically builds, solves, and post-processes a planar StressCheck® finite element mesh and provides solution verification feedback.

### LCAT Highlights

LCAT simplifies analysis of composite cutouts by providing clearly defined input with step by step validated feedback. Material properties can be defined by building a laminate layup that is automatically converted to an equivalent planar orthotropic material using classical lamination plate theory (CLPT), or a user-defined plate material can be entered. In-plane axial and shear loads can be specified at any orientation relative to the material directions.



### LCAT Results

LCAT reports maximum and minimum principal strains, far-field strains, margins of safety for the unrepaired cutout, and the orthotropic strain concentration factors. These values are given both graphically and numerically, with convergence information to assess the reliability of the results. An interactive 3D viewer displays strain contour plots which can be dynamically manipulated by the user. All LCAT models and solutions can be exported as StressCheck files for further analysis.

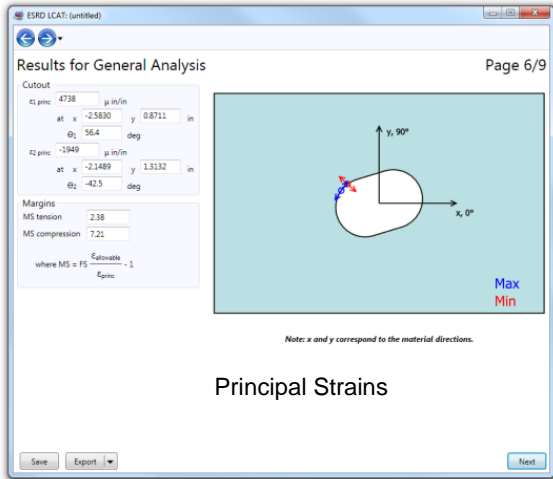
### Key Benefits

- Automatic computation of the laminate properties using CLPT given the lamina material and stacking sequence.
- The strain distribution around a cutout in the laminate composite being prepared for a repair.
- Margin of Safety calculation for the unrepaired cutout before sizing the repair patch.
- Both US (lbf, in) and SI (N, mm) unit systems are supported.
- A simple and intuitive user interface allows input engineering data in pre-defined templates with **step by step validation feedback**.
- Solution obtained with StressCheck, verified by p-extension.
- Built-in Results Viewer provides dynamic viewing of the FE mesh, the strain fringe contours and the strain directions.
- Modeling approach validated with Experimental results.

Laminate Cutout Analysis Tool

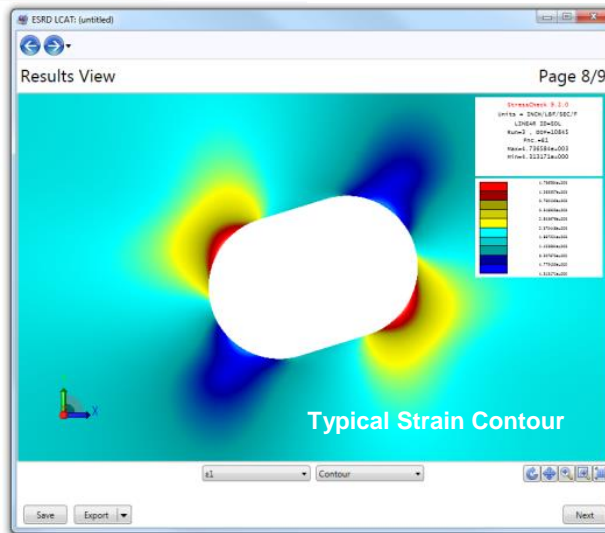
## Solution Verification

The StressCheck solver obtains a sequence of three solutions by increasing the polynomial order of elements on a fixed mesh. The principal strains, along with their direction and location in the mesh are then reported as a function of the number of degrees of freedom (DOF) to provide feedback information for solution verification.



**Convergence**

	Run 1	Run 2	Run 3	
Degrees of Freedom	6229	8365	10845	
$\epsilon_1$ princ	4736	4740	4738	$\mu$ in/in
$\epsilon_1$ princ x-location	-2.5830	2.6184	-2.5830	in
$\epsilon_1$ princ y-location	0.8711	-0.8200	0.8711	in
$\epsilon_1$ princ direction	56.3	57.4	56.4	deg
$\epsilon_2$ princ	-1958	-1960	-1949	$\mu$ in/in
$\epsilon_2$ princ x-location	-2.1982	-2.1489	-2.1489	in
$\epsilon_2$ princ y-location	1.2753	1.3132	1.3132	in
$\epsilon_2$ princ direction	-41.6	-42.5	-42.5	deg



## Deployment Options

LCAT is available as a StressCheck module or as a stand-alone 64 bit application accessible through the StressCheck Tool Box (SCTB) framework. A complete developer API is included with the LCAT installation for automation and custom .NET application development.

**SCTB** deploys FEA-based Smart Applications for the analysis of structural details such as single and multiple fastener joints composed of metal or laminated composite plates. Each Smart App provides convergence information to ascertain the quality of the numerical solution obtained with StressCheck.