DARWIN® 6.1 Release Notes

July 2008

Southwest Research Institute®

Summary of New Capabilities

DARWIN 6.1 includes the following new features:

- Stress Concentration Factors for General Inherent Analysis Mode
- New 2D Surface Damage Analysis Mode (Support for Turned Surfaces)
- New Crack Types and Revision of Existing Crack Types
- New Material Models (NASGRO® and Multilinear Equations)
- Mission Scaling Factor Enhancements
- Multiple Load Steps for 3D Surface Damage Analysis Mode
- New Finite Element Results Translator FE2NEU

Stress Concentration Factors for General Inherent Analysis Mode

A new DARWIN feature has been developed to address stress concentration factors (K_t) in 2D axisymmetric models for the General Inherent Analysis Mode. As shown in Figure 1, the feature is used to model the stress values at cross sections with and without holes (cross sections I and II, respectively). The analyst provides a 2D axisymmetric finite element model with hole elements removed. A GUI menu is provided that allows the analyst to describe the region over which K_t is applied. The analyst also provides values of K_t versus physical distance from the starting edge of a K_t region, which defines the length of the K_t region. Stress values within a K_t region are based on the product of K_t values and finite element stress results within the region.

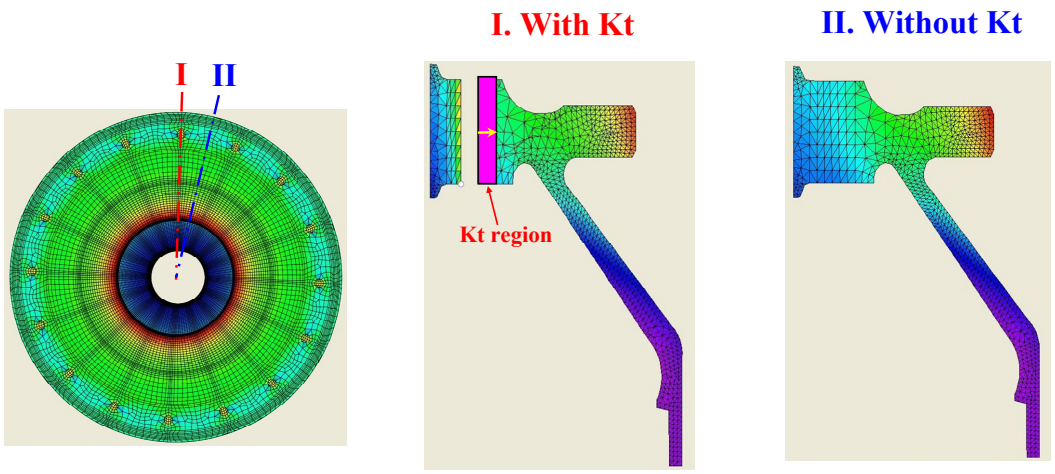

Figure 1: Disk cross sections influenced by stress concentration (K_t) effects

New 2D Surface Damage Analysis Mode (Support for Turned Surfaces)

The DARWIN Surface Damage Analysis mode has been extended for use with 2D axisymmetric finite element models. This new analysis mode includes many of the same features provided in the General Inherent Analysis mode, and was developed to provide treatment of turned surfaces associated with 2D axisymmetric finite element models.

New Crack Types and Revision of Existing Crack Types

Two new crack types are available in DARWIN 6.1: (1) bivariant surface crack SC19, and (2) univariant corner crack CC11. A number of crack types were revised in version 6.1. A summary of these revisions is provided in Table 1. Further details on these equations are provided in the DARWIN User's Guide.

New Material Models (NASGRO® and Multilinear Equations)

Two new fatigue crack growth equation forms are available in DARWIN 6.1: (1) the NASGRO® equation, and (2) a smoothed multilinear equation. Further details on these equation forms are provided in the DARWIN User's Guide.

Mission Scaling Factor Enhancements

A new mission scaling mode called "*Absolute*" is available in version 6.1 (Figure 2). In the *Absolute* mode, the user can specify stress and temperature values at the initial crack location for all zones within a specified zone group. The stress and temperature values in each zone are multiplied by the ratio of the specified absolute values to the FE stress and temperature values at the crack location in the zone. This is in contrast with the (previously available) "*Scale*" mode, in which the mission scaling factors are multipliers that are applied to stress and temperature values. In the "*Scale*" mode, the final stress and temperature values associated with a given load step are obtained by multiplying the stresses and temperatures from FE analysis by their respective scaling factors.

Table 1: Available crack types in DARWIN with associated damage parameters

P = polynomial solution, W = weight function solution

Crack Type	*Inherent (2D)*			*Surface Damage*		
	General		*Hard Alpha*	*1D*	*2D*	*3D*
	w/o Kt	w/ Kt				
EC02: univariant embedded crack (P,W)	✓	✓	✓	–	–	–
SC17: univariant surface crack (P,W)	✓	✓	✓	✓	✓	✓
SC15: univariant surface crack at hole (W)	–	–	–	✓	–	✓
SC18: univariant surface crack at off-center hole (W)	–	✓	–	✓	–	✓
SC19: bivariant surface crack (P,W)	✓	✓	✓	-	✓	✓
CC01: bilinear corner crack (P)	✓	✓	✓	-	✓	-
CC05: univariant corner crack at hole (W)	–	–	–	✓	–	✓
CC08: univariant corner crack at off-center hole (W)	–	✓	–	✓	–	✓
CC09: bivariant corner crack (P,W)						
Bivariant	✓	✓	✓	–	✓	✓
Univariant (primary direction)	✓	✓	✓	–	✓	✓
Univariant (secondary direction)	✓	✓	✓	–	✓	✓
CC10: bivariant corner crack at off-center hole (W)	–	✓	–	–	–	✓
CC11: univariant corner crack in plate (P,W)	✓	✓	-	✓	✓	✓
TC01: univariant embedded through crack (P)	✓	✓	✓	–	✓	–
TC02: univariant surface through crack (P)	✓	✓	✓	✓	✓	✓
TC11: univariant through crack at hole (W)	–	–	–	✓	–	✓
TC13: univariant through crack at off-center hole (**W**)	–	✓	–	✓	–	✓

General Notes:

1. SC02 has been renamed to SC17 to correspond to the same solution in NASGRO.
2. TC12 (which has always been a weight function solution) has been renamed to TC13 to correspond to the same solution in NASGRO.
3. SC19 and CC11 are new solutions appearing for the first time in DARWIN 6.1.
4. The former SC15, CC05, and TC11 are obsolete solutions that have been removed in DARWIN 6.1. These solutions were previously superseded by SC18, CC08, and TC13.
5. The EC02 solution also incorporates the EC03 solution for an embedded crack approaching a free surface.
6. **Bivariant solutions (SC19, CC09, CC10) are not implemented in 1D surface damage mode.**

Availability Notes:

1. Corner-crack-in-plate solutions CC01 and CC09 were not previously available in surface damage (1D or 3D) mode. The CC09 bivariant solution and the new CC11 univariant solution have been added to the appropriate surface damage analysis modes in version 6.1.
2. Through-crack solutions TC01, TC02, and TC13 are not currently available as initial crack types in the GUI in any mode (inherent or surface damage).

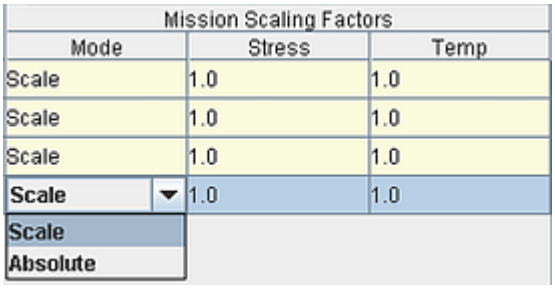

Figure 2: New "Absolute" mission scaling mode factor in DARWIN 6.1

Multiple Load Steps for 3D Surface Damage Analysis Mode

Support for multiple load steps was recently added to the 3D surface damage analysis mode (this analysis mode was previously limited to a single load step).

New Finite Element Results Translator FE2NEU

A new finite element results translator called FE2NEU has been developed to convert ANSYS finite element results to the neutral file format required by DARWIN. It is available for ANSYS version 10.0 (and previous versions). The FE2NEU translator supports all of the finite element types that were previously supported by ANS2NEU (Table 2) and provides filtering capabilities that are identical to those found in ANS2NEU (i.e., include/exclude specific load cases or elements based on element type, element number, material number, or load case number).

FE2NEU includes several important features not available in previous versions of ANS2NEU. It supports ANSYS version 10 files (and also previous ANSYS versions). FE2NEU provides support for large ANSYS results files (successfully tested for several large files including a file with 953226 PLANE82 elements). It allows the user to specify filter commands on a single (200 character) line (ANS2NEU requires multiple lines) and provides output with nodes and elements sorted in ascending order.

FE2NEU is also computationally efficient. For example, FE2NEU recently completed translation of a 1.5 GB test file (with nearly 1 million elements) in 37 seconds.

Table 2: Finite element types supported by FE2NEU

ANSYS Type	SIESTA Type	Description
PLANE42	EL2D	4-node quadrilateral
PLANE182	EL2D	4-node quadrilateral
PLANE82	PE2D	8-node quadrilateral
PLANE183	PE2D	8-node quadrilateral
SOLID45	BRI8	8-node brick
SOLID185	BRI8	8-node brick
SOLID95	VANS N20	20-node brick
SOLID186	VANS N20	20-node brick
SOLID92	TETS N10	10-node tetrahedron
SOLID187	TETS N10	10-node tetrahedron