Southwest Research Institute

D

DARWIN[TM]

R&D 2000 Winner

Design **A**ssessment of **R**eliability **W**ith **IN**spection

A Tutorial

Sposored by the **F**ederal **A**viation **A**dministration

About this tutorial:

The purpose of this tutorial is to show the user how to use the DARWIN™ Graphical User Interface (GUI).

It is recommended that first-time users open the DARWIN™ GUI alongside this tutorial to practice execution of the instructions included in each lesson.

ENJOY!

This tutorial was created by Southwest Research Institute as part of the ongoing Turbine Rotor Material Design effort funded by the Federal Aviation Administration.
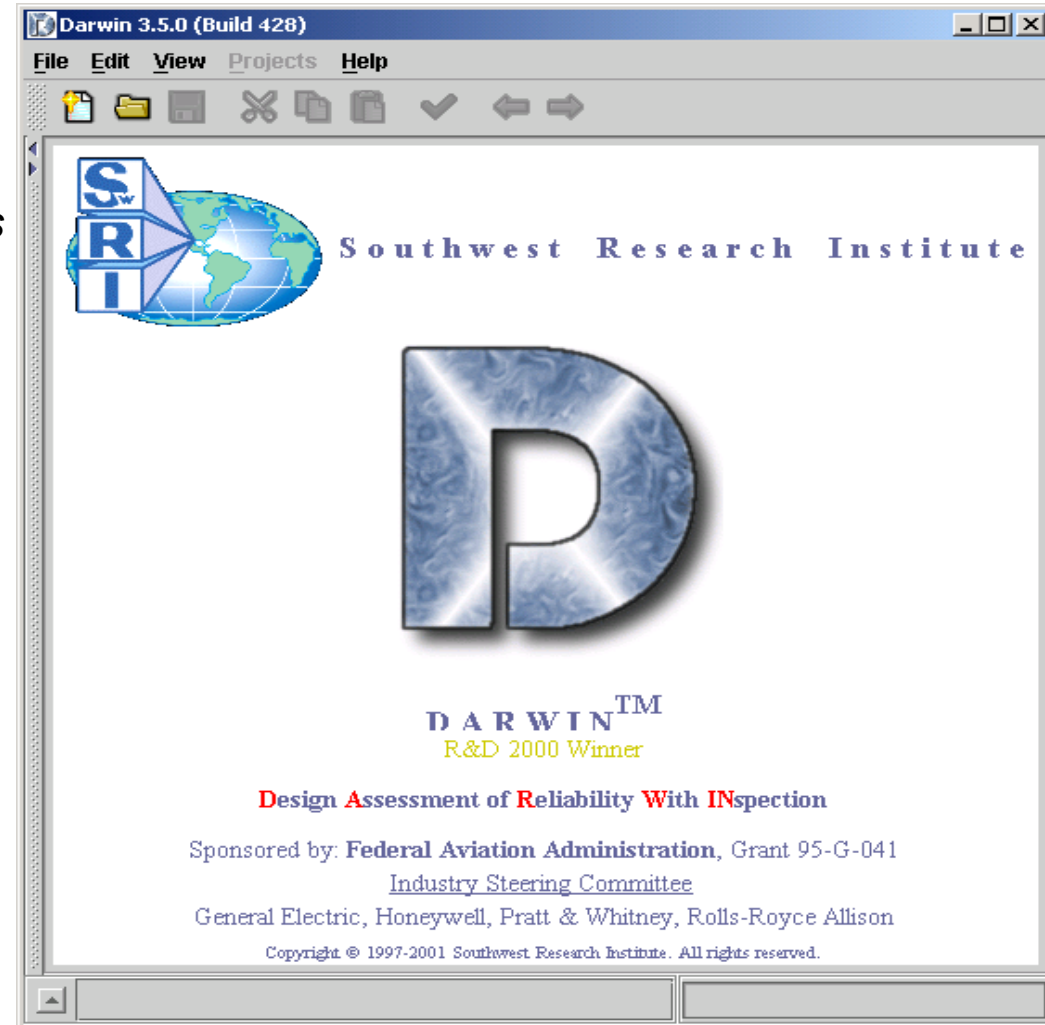
To begin using the DARWIN™ GUI:

It is recommended that you open the DARWIN™ GUI and practice the procedures covered in this tutorial.

For Unix Users:

Type DarwinGUI/ and press *enter*.

For Windows 95, 98, 2000, NT:

Double click on the desktop icon , or execute the following sequence of steps:

Darwin 3.5.0 (Build 428)

File   Edit   View   Projects   Help

Southwest Research Institute

DARWIN™
R&D 2000 Winner

Design Assessment of Reliability With INspection

Sponsored by: **Federal Aviation Administration**, Grant 95-G-041

Industry Steering Committee

General Electric, Honeywell, Pratt & Whitney, Rolls-Royce Allison

Copyright © 1997-2001 Southwest Research Institute. All rights reserved.

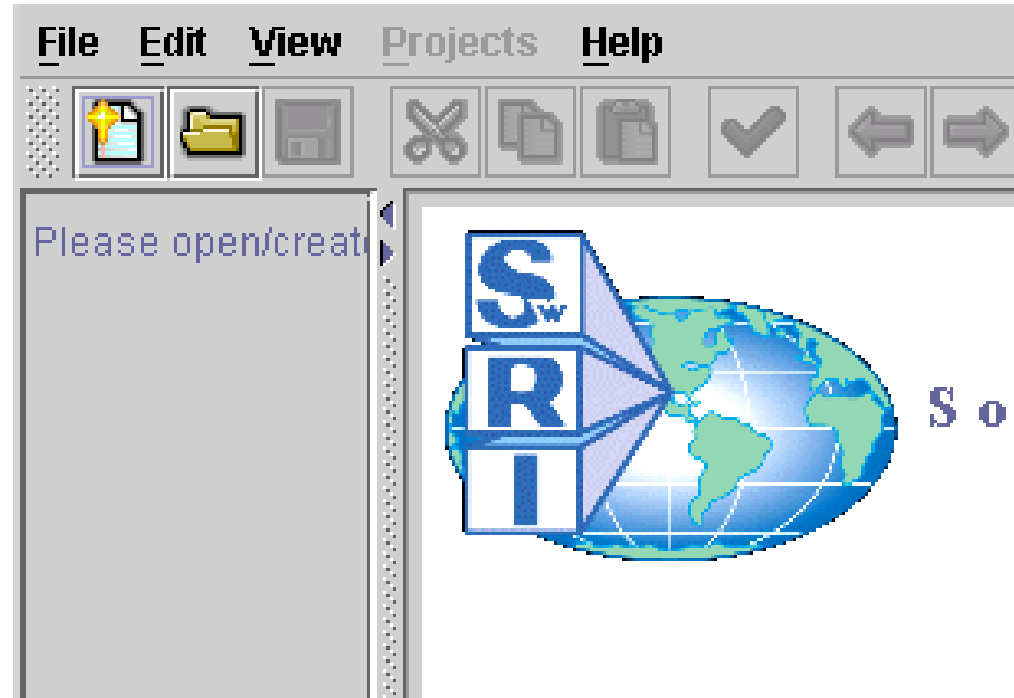Start ➡ Programs ➡ Darwin ➡ DarwinGUI

Creating a Project

Darwin organizes analysis using projects. Projects are organized in 2 parts, **Assessment Definition** *and* **Evaluate Results***, which are explained in detail in this tutorial. The first thing you must do to begin your analysis is create a project.*

1 ➡ **File** **Edit** **View** Projects **Help**

2 ➡

Please open/creat

To Create a Project (you may do either 1 or 2):

1. Click on **File**, and then click on **New**.

-or-

2. Click on [ ] .

The Outline

impeller-fl-normal.ddb
- Assessment Definition
  - Setup
  - Fracture Model
  - Defect Distribution
  - POD Curves
  - Material Properties
  - Inspections
  - Missions
  - **Stress and Zones**
  - Probability Method
  - Analysis Options
- Evaluate Results
  - Job Files
  - *pf* Risk Assessment
    - Disk Assessment
    - Disk Assessment/Cycle
    - Zone Assessments
    - Contribution Factors
  - Fracture Mechanics
    - kMax vs. Cycles
    - Area vs. Cycles
    - Crack State
  - Stress Processing

The outline is located on the left-hand side of the Graphical User Interface.

It allows for quick browsing through the sections of your project.
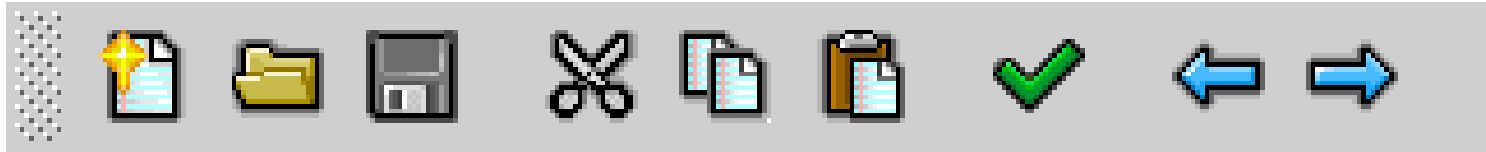
To Use the Outline:

Click on the name of a section or on the icon next to the name. Details will appear on the right-hand side of the screen.

The Main Toolbar

The main toolbar is located near the top of the DARWIN^{TM} GUI window:

- Create a new session (Ctrl+N)

- Open a saved session (Ctrl+O)

- Save current session (Ctrl+S)

- Cut text (Ctrl+X)

- Copy text (Ctrl+C)

- Paste Text (Ctrl+V)

- Validate Current Data (Ctrl+R)

- Go to previous Step (Ctrl+B)

- Go to next step (Ctrl+F)

The Setup Screen

*The Setup screen is the first part of the **Assessment Definition** section of your project.*

General Information can be entered in the spaces provided.

NOTE: A problem title is <u>required</u>, comments are optional.

Service Life - Enter service life information or use the default values.

---

impeller-fl-normal.ddb: Setup

General Information

Problem Title:	IMPELLER TEST CASE WITH GRADIENT WITH INSPECTION
Comment Line 1:	RANDOM INSPECTION TIME SPECIFIED BY A NORMAL DISTRIBUTION
Comment Line 2:	
Comment Line 3:	
Comment Line 4:	

Analysis mode
- ⦿ Inherent
- ◯ Surface Damage

Units
- ⦿ US Customary
- ◯ Metric

View Units...

Service Life

Beginning Cycle: 2000
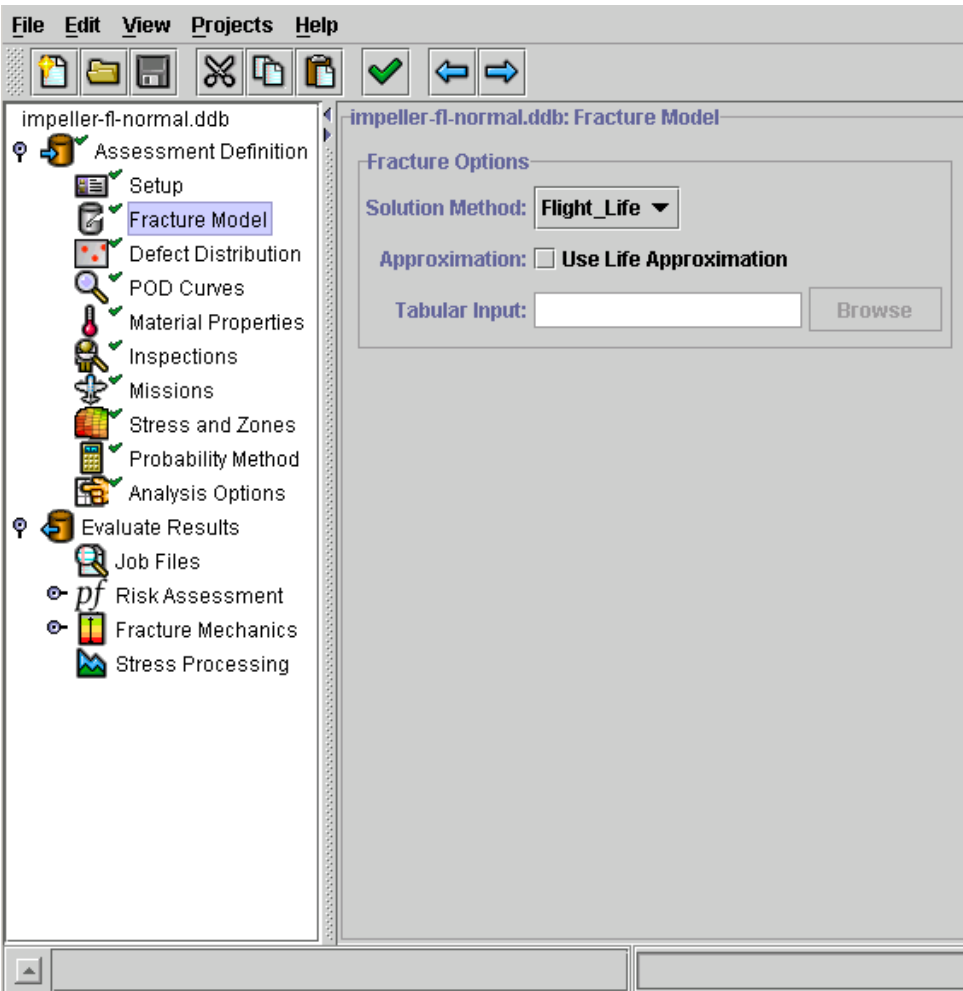Ending Cycle: 20000
Cycles per Increment: 2000

Analysis Mode - Click on one of these circles to change the Darwin™ Analysis Mode between Inherent (Hard Alpha) material defect analysis and Surface Damage analysis.

Units - Click on one of these circles to select either Metric or US Customary units.

Fracture Model

The Fracture Model part of your project assessment definition allows you the freedom to choose your own fracture mechanics **Solution Method,** including the DARWIN™ "Flight_life" module, a user-supplied software module, or the tabular data method.

DARWIN™ also provides a fast solution technique called "Life Approximation" that can be used with the Flight_life or user defined modules (see the DARWIN™ Users Guide for further information).

To change Fracture Model Options:
Solution method may be changed by clicking on the down arrow ▼ on the title bar. Life Approximation can be toggled on and off by clicking on the checkmark ☑ .

In the Defect Distribution part of your project assessment definition, you will need to add a distribution. You can add a distribution from either a previously made file (see DARWIN™ Users Guide for file creation instructions), or from the DARWIN™ defect distribution library.
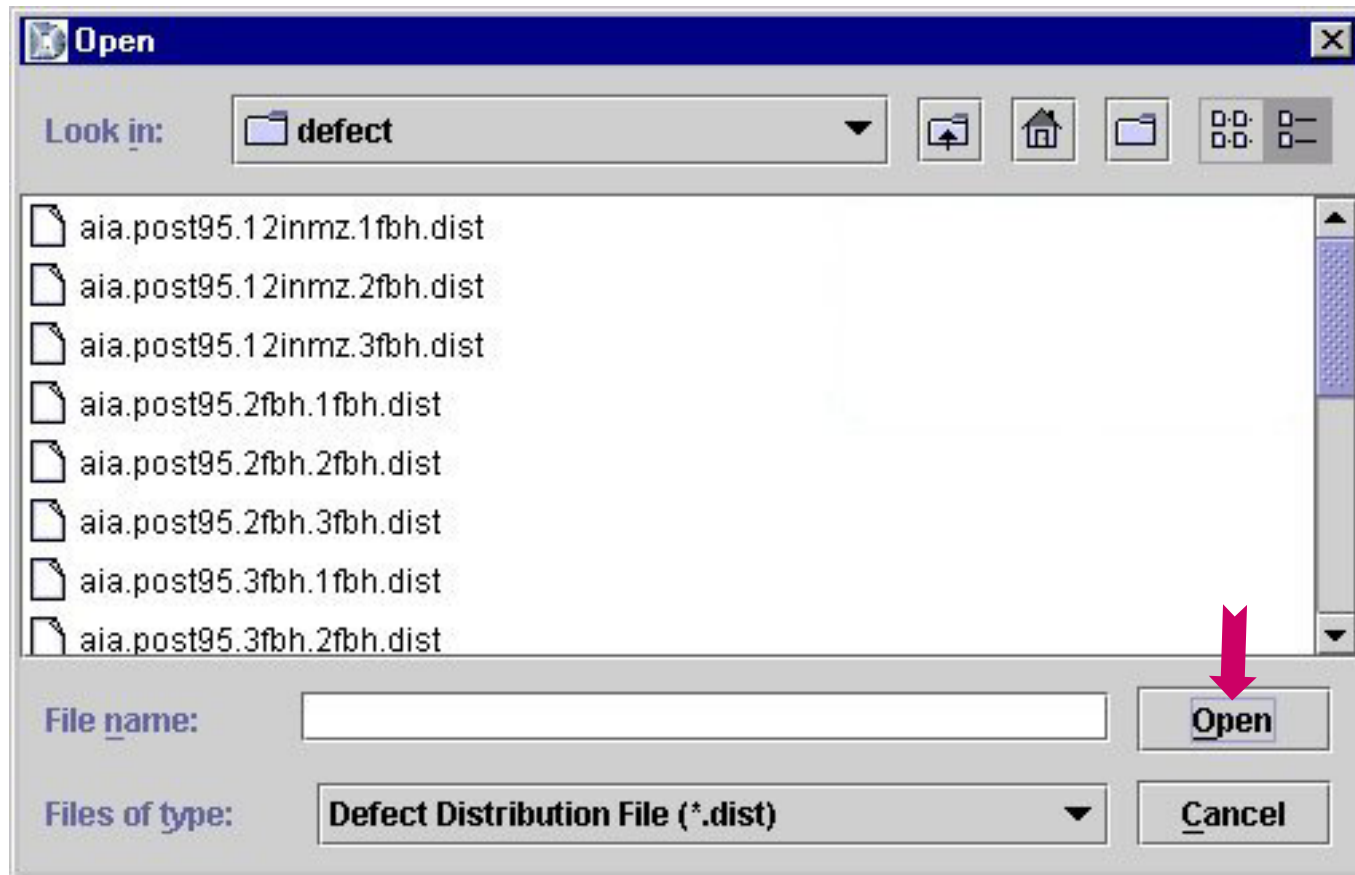
To Add a Distribution:

1. Click on [+] .
2. Choose "File" to add a distribution from a user-defined file, or "Library" to add a distribution from the DARWIN™ library.
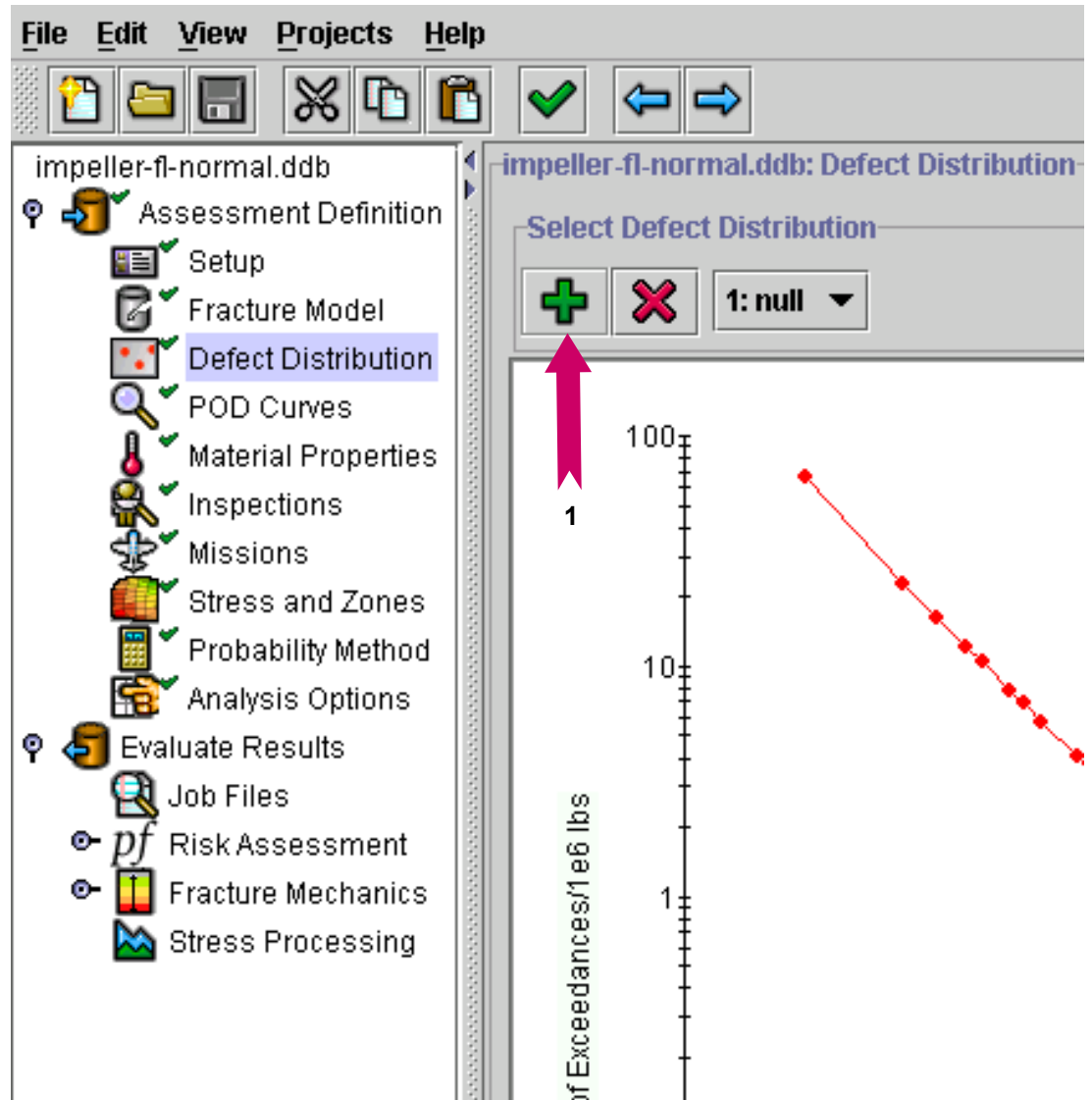
If you chose to select a file from the DARWIN™ Library, the above window will appear.

To Open a File: Click on the file you wish to open, and then click on "**Open**".
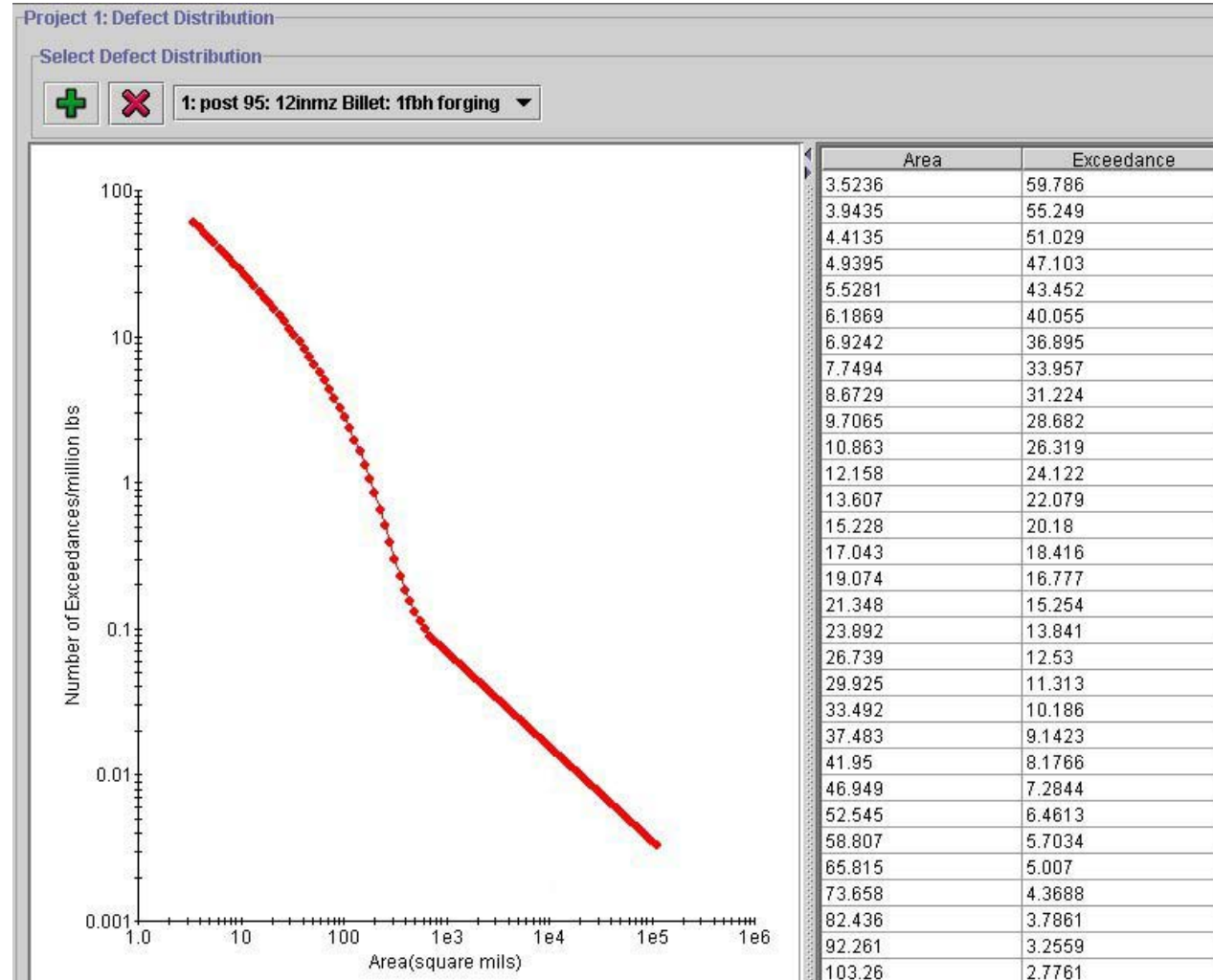
DARWIN TIPS:

Now that a distribution has been added, you can view the area and exceedance values on the right hand side of the screen.

You may add more distributions in the same way that you just learned.

If you wish to remove the distribution that you have selected, click on the ✖ and the distribution you selected will be removed from the list.
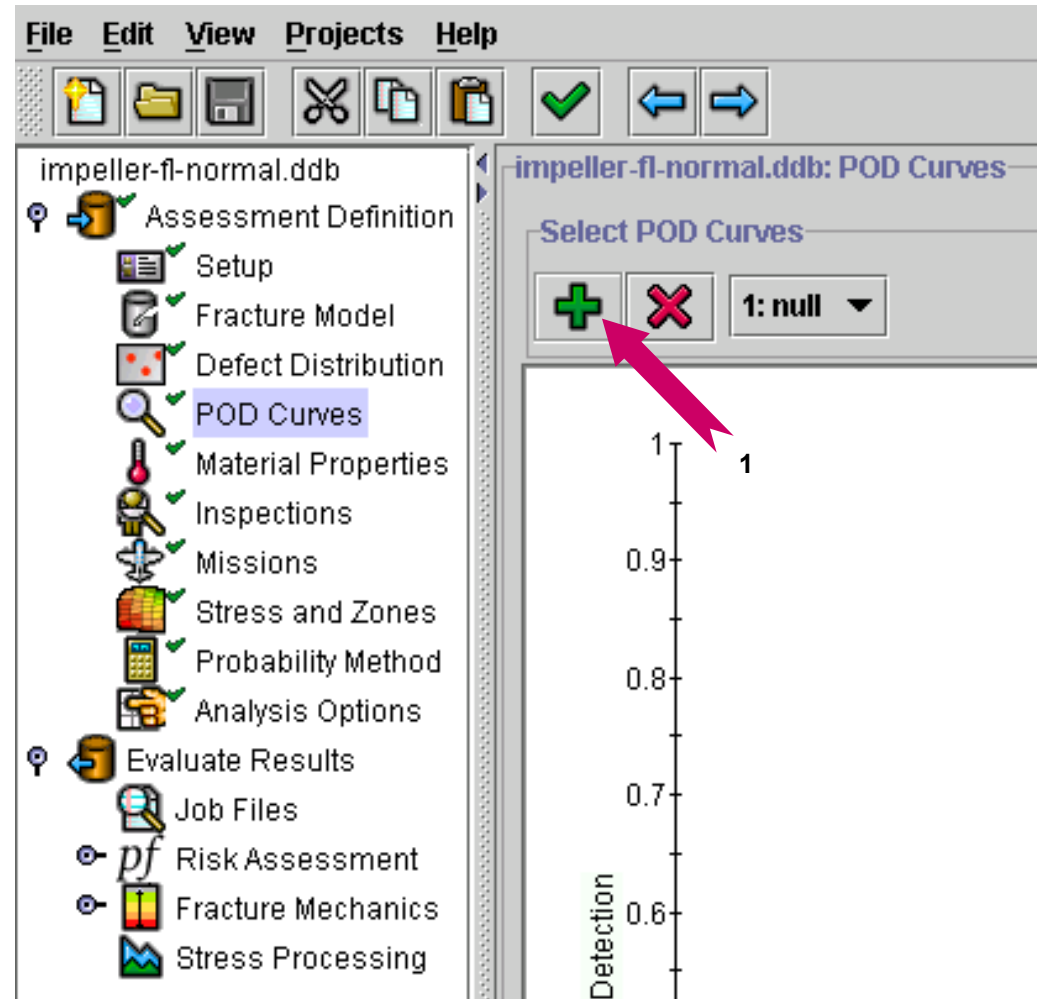
Project 1: Defect Distribution

Select Defect Distribution

1: post 95: 12inmz Billet: 1fbh forging

Area	Exceedance
3.5236	59.786
3.9435	55.249
4.4135	51.029
4.9395	47.103
5.5281	43.452
6.1869	40.055
6.9242	36.895
7.7494	33.957
8.6729	31.224
9.7065	28.682
10.863	26.319
12.158	24.122
13.607	22.079
15.228	20.18
17.043	18.416
19.074	16.777
21.348	15.254
23.892	13.841
26.739	12.53
29.925	11.313
33.492	10.186
37.483	9.1423
41.95	8.1766
46.949	7.2844
52.545	6.4613
58.807	5.7034
65.815	5.007
73.658	4.3688
82.436	3.7861
92.261	3.2559
103.26	2.7761

Number of Exceedances/million lbs

Area(square mils)

In the Probability of Detection Curves part of your project assessment definition, you may add a POD curve. You can add POD data either from a previously made file (see DARWIN^{TM} Users Guide for file creation instructions), or from the DARWIN^{TM} POD curve library.
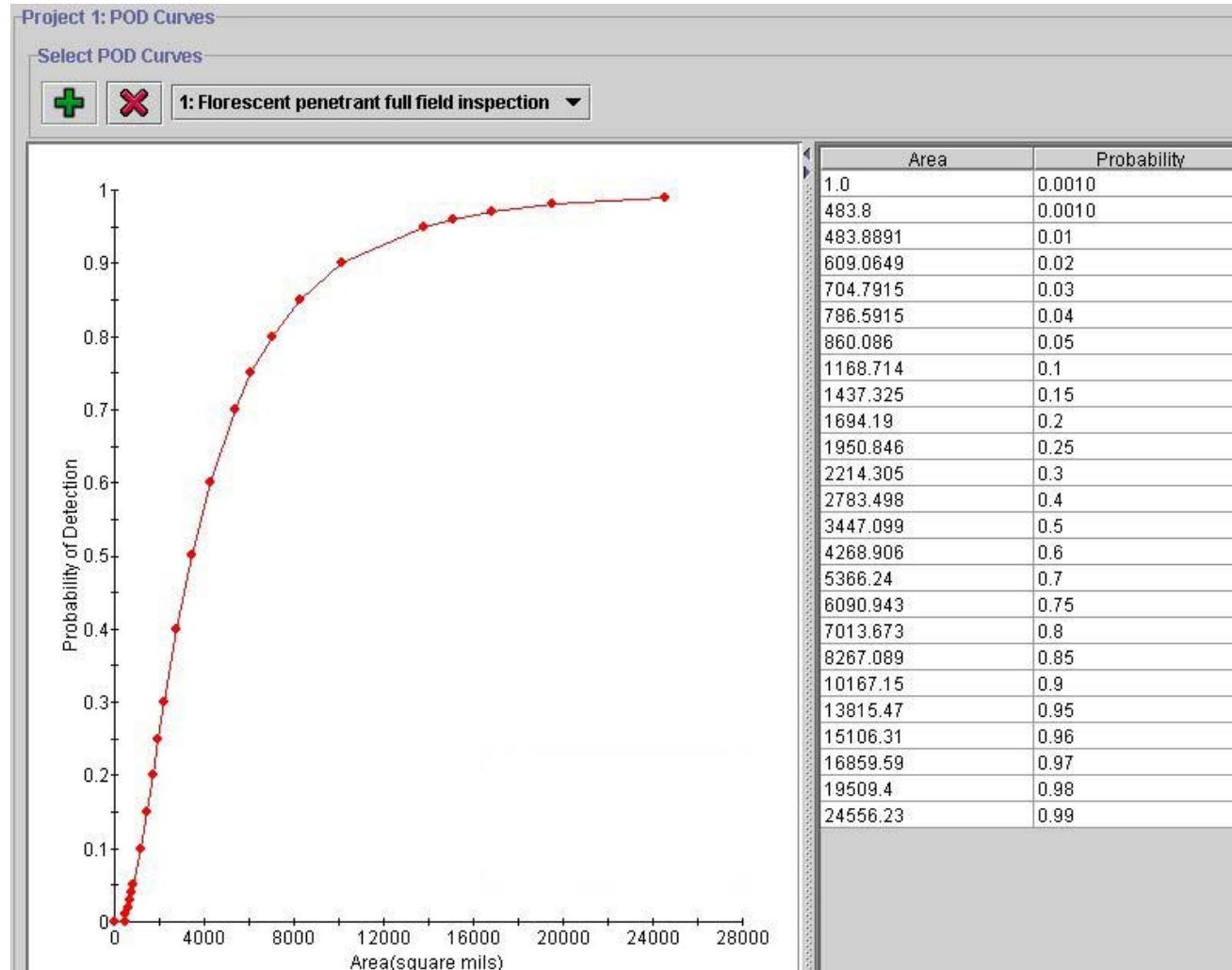
To Add a POD curve*:

1. Click on [+].

2. Choose "File" to add a POD curve from a file, or "Library" to add a POD curve from the DARWIN^{TM} library.

File Edit View Projects Help

impeller-fl-normal.ddb

- Assessment Definition
  - Setup
  - Fracture Model
  - Defect Distribution
  - POD Curves
  - Material Properties
  - Inspections
  - Missions
  - Stress and Zones
  - Probability Method
  - Analysis Options
- Evaluate Results
  - Job Files
  - *pf* Risk Assessment
  - Fracture Mechanics
  - Stress Processing

impeller-fl-normal.ddb: POD Curves

Select POD Curves

[+] [X] 1: null ▼

Detection

1
0.9
0.8
0.7
0.6

*NOTE: This process is very similar to the one used in the previous section (Defect Distribution).
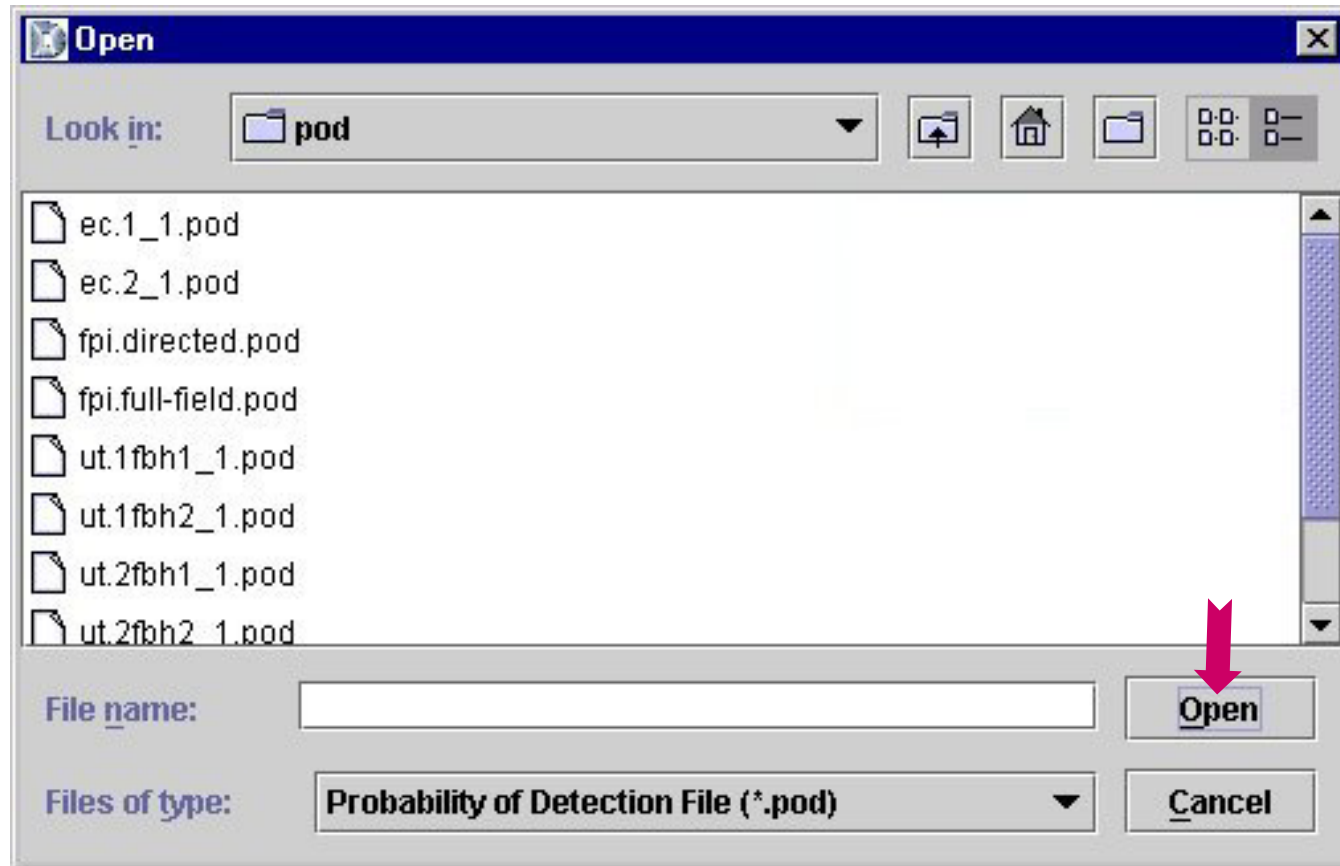
If you chose to select a file from the DARWIN™ Library, the above window will appear.

To Open a File: Click on the file you wish to open, and then click on "**Open**".

DARWIN TIPS:

Now that a POD curve has been added, you can view the area and the probability of detection values on the right hand side of the screen.

You may add more functions in the same way that you just learned.

If you wish to remove the function that you have selected, click on ✖ *, and the function you selected will be removed from the list.*

Project 1: POD Curves

Select POD Curves

➕ ✖ 1: Florescent penetrant full field inspection ▼

Area	Probability
1.0	0.0010
483.8	0.0010
483.8891	0.01
609.0649	0.02
704.7915	0.03
786.5915	0.04
860.086	0.05
1168.714	0.1
1437.325	0.15
1694.19	0.2
1950.846	0.25
2214.305	0.3
2783.498	0.4
3447.099	0.5
4268.906	0.6
5366.24	0.7
6090.943	0.75
7013.673	0.8
8267.089	0.85
10167.15	0.9
13815.47	0.95
15106.31	0.96
16859.59	0.97
19509.4	0.98
24556.23	0.99

In the Inspection part of your project assessment definition you will define the inspection schedule information.

First, you will need to add a new Inspection Schedule.
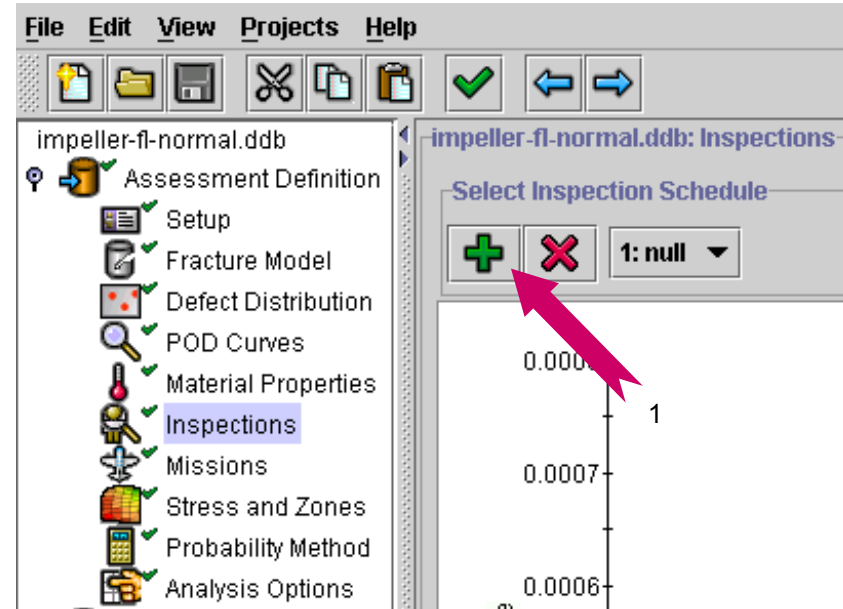
To add an Inspection Schedule:

1. Click on [+].
2. When the "Input" window appears on the screen, enter a name for the Inspection Schedule by typing it in the white text box.
3. Click "**OK**" when completed.
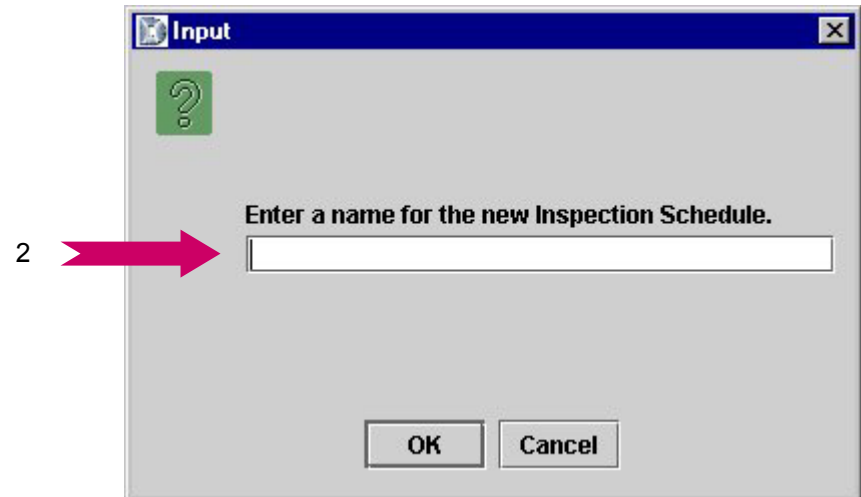
Now that you have assigned a name for your Inspection Schedule, you can input information about the schedule using either "Normal" or "Tabular" format. In addition, you can specify the percentage of rotor-disks that are not inspected. Results can be viewed in either PDF (Probability Density Function) or CDF (Cumulative Distribution Function) format.

1. Not Inspected (%)
Enter a number (0-100) indicating the percentage of rotor-disks that are not inspected.

2. Format
Left-clicking on this box will allow you to toggle the format mode between "Normal" and "Tabular" (further details on the next two pages).

3. Associated POD Curves
If your project contains POD curves that you defined in previous parts of the assessment definition, they will appear here.

4. Inspections
The inspection times associated with each inspection are entered in this section. To add additional inspections, click "**NEW**".
To remove them, select the item and click "**REMOVE**".

PDF or CDF

When specifying an inspection distribution using "Normal" format, enter the mean and standard deviation of the inspection time associated with each inspection in the "Inspections" table. You may then view your distribution in PDF or CDF form.

To Use "Normal" Format:

1. First, change **Format** to show the word "Normal".

2. Next, enter the mean and standard deviation in the "Inspections" table.

3. You may add and remove inspections using the **New** and **Remove** buttons, respectively.

Not inspected (%): 0.0

Format: Table ▼

Associated POD Curves

Inspections

#	# Table Values
1	0 (click to edit)

To Use Table Format:
1. Select "Table" instead of "Normal" in the **Format** section. Also, make sure the graph is set to CDF mode.
2. Next, click on the white space underneath the **#Table Values** title in the **Inspections** section. The window below will appear.

Inspection Probabilities

Inspection 1

Cycles	Prob(%)
100	0
200	25
300	50
400	75
500	100

OK   Cancel

3. Insert the number of cycles and their corresponding cumulative probabilities in the table.

NOTE: Inspection times (cycles) must be entered in ascending order and their corresponding probabilities must also be entered starting with 0% and ending with 100%, as shown in this figure

4. Click **OK** when complete.

DARWIN TIPS:

Specification of Associated POD Curves:

POD curves previously defined in the POD section of the assessment definition
will appear in the **Associated POD Curves** window.

To select a single POD curve associated with your inspection
schedule, click on the name of the curve.

To select multiple POD curves, hold down the "Ctrl" key and
click on the names of the curves.

NOTE: All of the inspections within an inspection schedule must be associated with the same
POD curve(s). If you need to define inspection times that have different POD curves,
create a separate inspection schedule for each inspection time.

Associated POD Curves
1: Eddy current inspection wi...
2: Ultrasonic inspection, #1 ...
3: Ultrasonic inspection: #3 ...

Adding data for the Material Properties section of your project assessment definition is very similar to the way we added it in the other sections of this tutorial. You can add data from the DARWIN™ library or from a user defined file (see the DARWIN™ Users Guide for file creation instructions).

To Add Material Data:

1. Click on ⊞ .

2. The Open selection screen will appear. Click on the file that you wish to open, and click **Open**.

File   Edit   View   Projects   Help

impeller-fl-normal.ddb
- Assessment Definition
  - Setup
  - Fracture Model
  - Defect Distribution
  - POD Curves
  - **Material Properties**
  - Inspections
  - Missions

impeller-fl-normal.ddb: Material Properties

Select Material Properties

1: <no name>

da/dN Data (Paris in Vacuum)

Temp	100.0
C	5.24799995E-11
n Exp	3.86999989
delta Kth	0.0
Kc	58.7000008

Open

Look in: material

- ti64_Paris_R03.mat
- ti64_Paris_R25.mat
- ti64_Paris_R54.mat
- ti64_Paris_closure.mat
- ti64_Paris_walker.mat

File name:

Files of type: Material File (*.mat)

Open   2

Cancel

The materials data contains two different types of property data: (1) fatigue crack growth rate (da/dN) and (2) constitutive relationship (stress-strain). You can choose which data you wish to view by clicking on the top part of the tab (tabs are located on the right side of the screen). You can view the da/dN data in either vacuum or air environments by clicking in the circles in the "FCG Environment" section (Note: due to compatibility issues with the current defect distributions, actual vacuum data should not be used, and so vacuum properties are currently set equal to air properties). You can also view the data in graphical format (da/dN vs. ΔK, or stress vs. strain).

To view material data and da/dN graph:

1. Select the da/dN tab by clicking on it (select Vacuum to Air environment).

2. Click on the **Plot** button near the bottom center of the screen.

3. The graph of da/dN vs. ΔK will appear.

4. Click on **Close** when you are finished viewing the graph.

1

| da/dN | Stress-Strain |

FCG Equation
Paris

Stress Ratio Model
Walker Equation

Temp Interpolation
Interpolate

FCG Environment
⦿ Vacuum   ◯ Air

2

Plot

3

Ti64 props. in air: Paris FCG eqn., Walker_Eqn...

Ti64 props. in air: Paris FCG eqn., Walker_Eqn stress ratio eqn.
FCG Equation: Paris
Stress Ratio Model: Walker Equation
Temp Interpolation: Interpolate
Environment: Vacuum

Temperature Sets
● 75.0
■ 300.0
▲ 600.0

Close
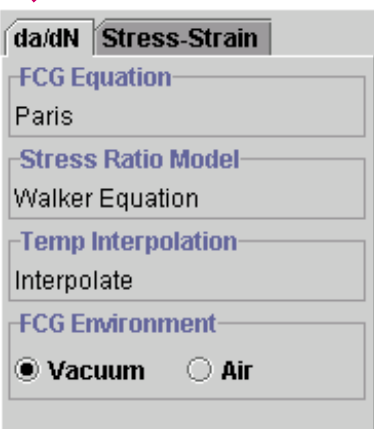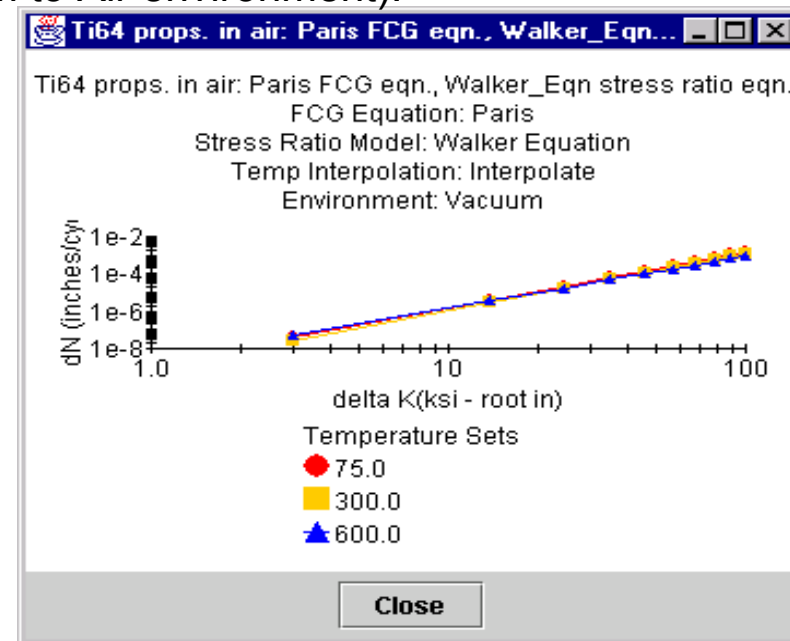
To view a Stress vs. Strain graph:

1. Select the Stress-Strain tab by clicking on it (select Monotonic or Cyclic).

2. Click on the **Plot** button near the bottom center of the screen.

3. The graph of Stress vs. Strain will appear.

4. Click on **Close** when you are finished viewing the graph.
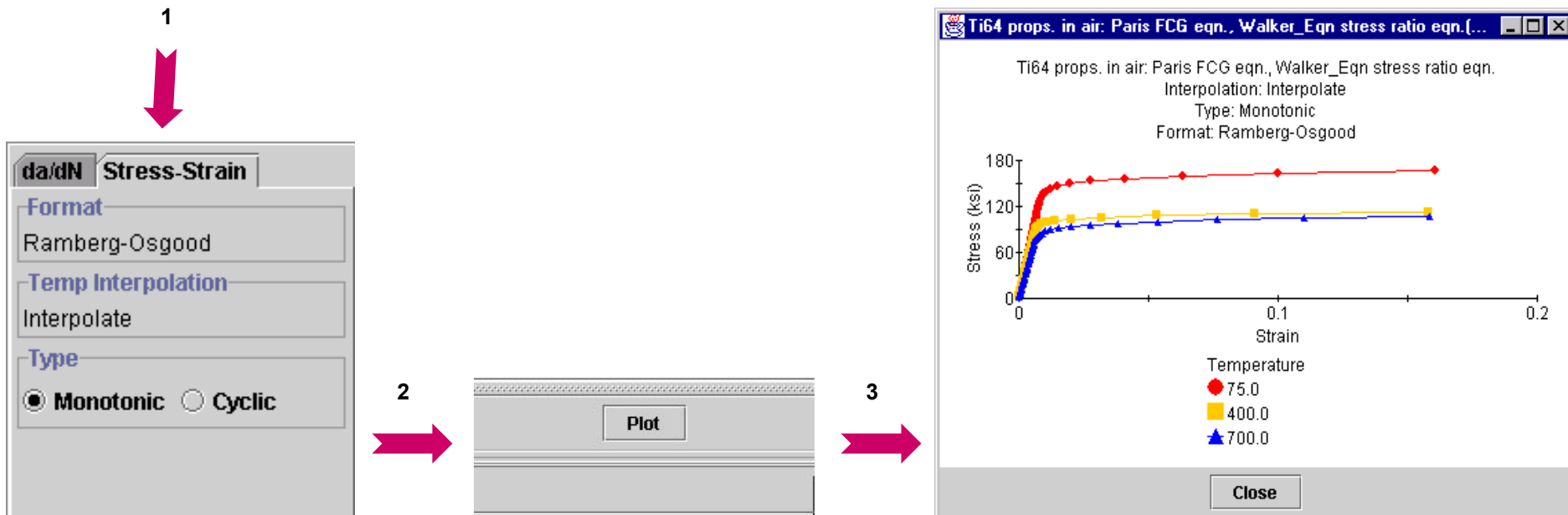
*The Stress and Zones section allows you to define zones and specify associated zone data. You will learn how to load finite element results into the GUI to define disk geometry and associated stresses & temperatures. You will learn how to create zones and specify the crack location, plate size boundaries (for fracture mechanics analysis), and associated material properties and inspection data. **This section only appears if the Analysis Mode is set to "Inherent" (see Setup section for details).***

To Load Geometry:

1. Make sure you're in the Stress and Zones section of the Assessment Definition (i.e. click on "Stress and Zones" in the outline on the left side of the screen).

2. Click on ⬚ .

3. After you have done this, a window will appear. Select the *.uif file you wish to load (e.g. impeller.uif), and click **Open**.

4. The model geometry will appear in the project window as shown on the next page (this may take several seconds depending on the size of the finite element data file).
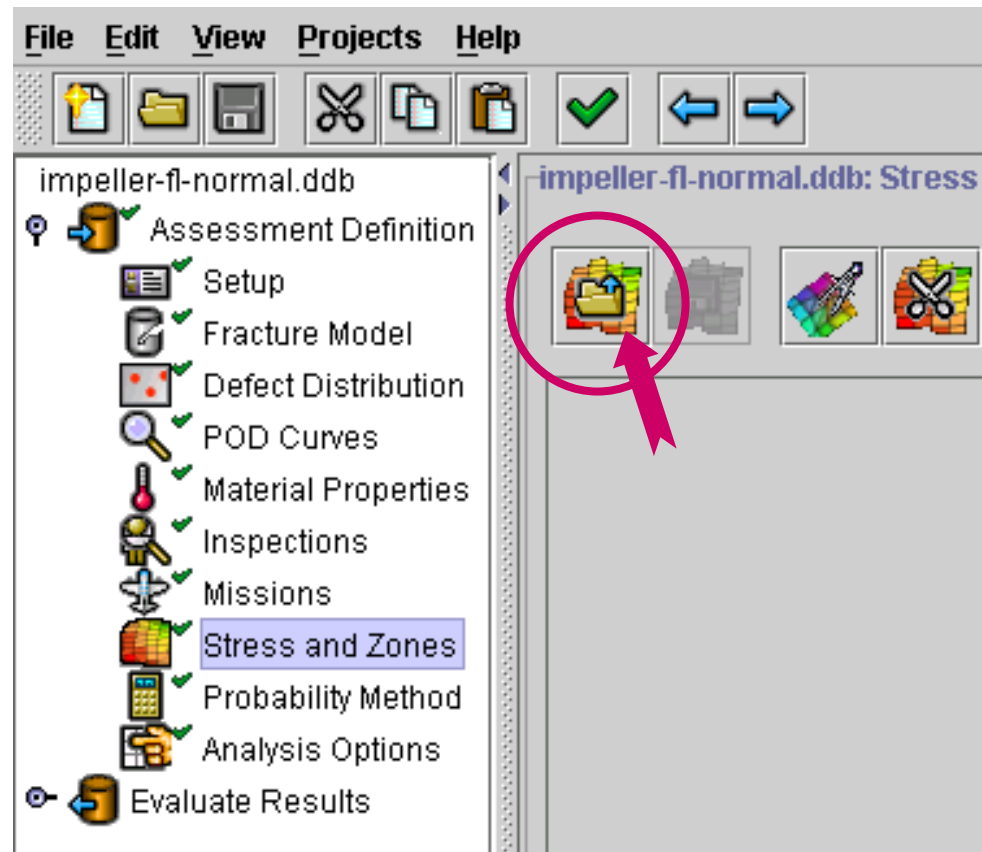
Let's examine the project window and some of the GUI functions you will use.

1. Load Geometry
Click here to load finite element geometry (see previous page).

2. Edit Zones
This function is discussed on page 6 of this section.

3. Subdivide Elements
See Advanced Topics.

4. Set Contour Values
This function is discussed on page 6 of this section.

5. Set Zone Risk Contribution
See Advanced Topics

6. Load Cases
To add load cases, click ➕ .
To delete load cases click ✖ .
This function is discussed on page 5 of this section.

impeller-fl-normal.ddb: Stress and Zones

Load Cases

2 (impneu.uof)

1   2   3   4   5

6

25

Min Stress:
Max Stress:
Mouse Coord (R, Z):
Current Stress:
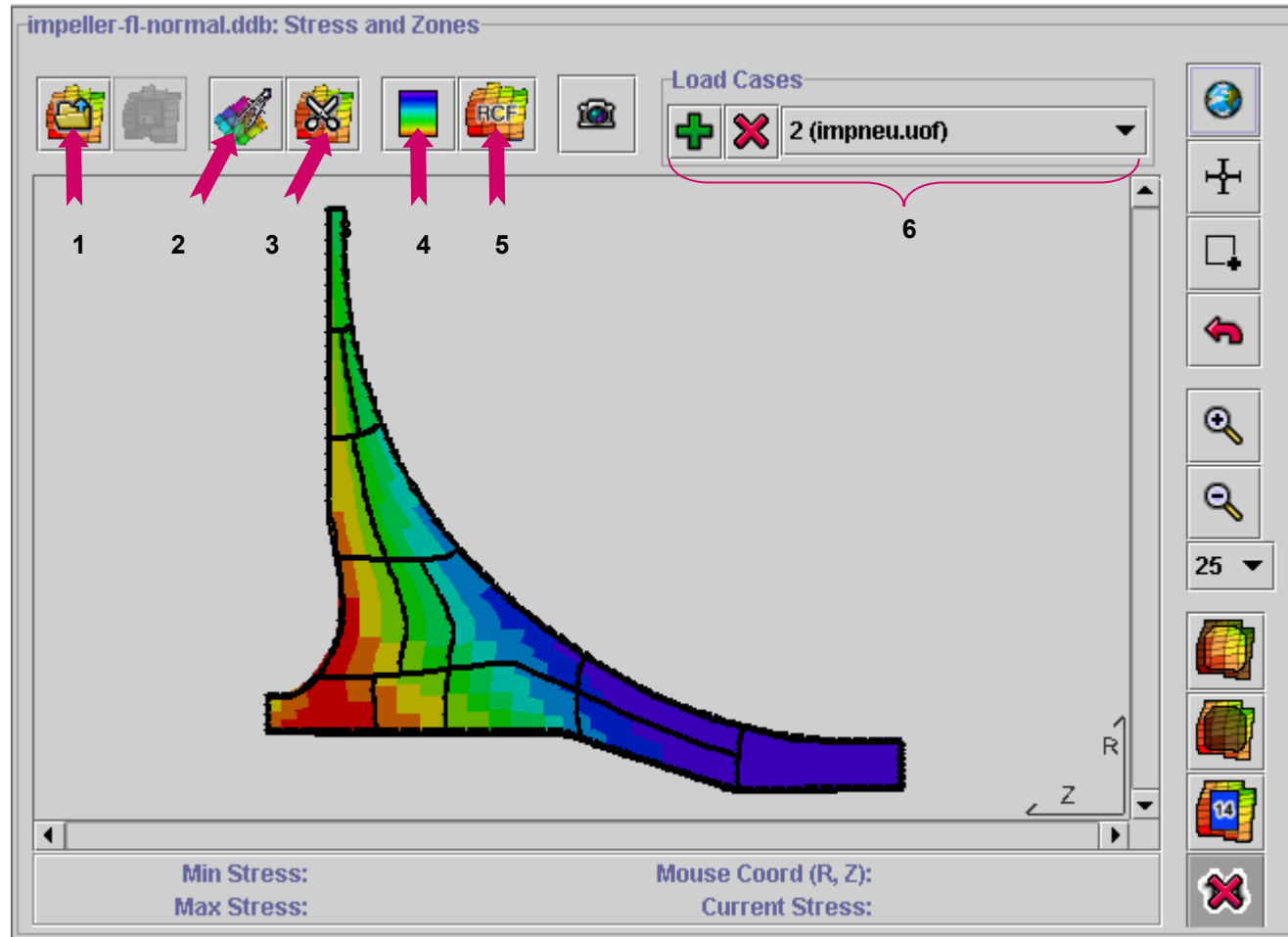
Viewing Functions:

1. Show Default Rotor Model - Click on this button to view the entire rotor model (restores the zoom settings to the original state).

2. Center on Location - To center the view window on a particular point, click this button, and then click on the point at which the view is to be centered.

3. Zoom Box - If you want to zoom in on a specific area, click this button, then click and drag to define a rectangular area.

4. Go Back - Click this button to return to the previous view.

5. Zoom IN - Click this button to zoom in on the rotor model. See (7) to change the zoom percentage.

6. Zoom OUT - Click this button to zoom out on the rotor model. See (7) to change the zoom percentage.

7. Zoom Percentage - Click this button to change the zoom percentages. This button affects the Zoom Box, Zoom In, and Zoom Out commands.
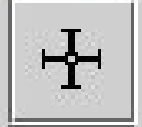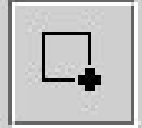
Viewing Functions (continued):

8. Hide/Show Non-surface Elements - Click this button to hide or show the embedded rotor model elements.

8 ➡

9. Hide/Show Surface Elements - Click this button to hide or show the surface elements of the current rotor model.

9 ➡

10. Hide/Show Zoned Elements - Click this button to hide or show all rotor elements that are part of created zones.

10 ➡

11. Hide/Show Zone Numbers - Click this button to hide or show the zone numbers on the rotor model

11 ➡

Now that a model geometry has been added, you need to add load cases (stress and temperature results). You can use DARWIN^{TM} GUI functions to modify the appearance of stress contours and to define multiplier factors to be applied to stress results. The latest DARWIN^{TM} GUI will add default load cases automatically.

To Add Load Cases (manually):

1. First, click on ➕.

2. When the window appears, select the file you wish to add (e.g., impeller.uof), and click **Open**.

3. Stress contours will appear super-imposed on the rotor model geometry. You can view stress contours for various load cases (click on the titles in the title bar list).

4. You can click on the **Set Contour Values** button to modify the appearance of stress contour values (see next page).
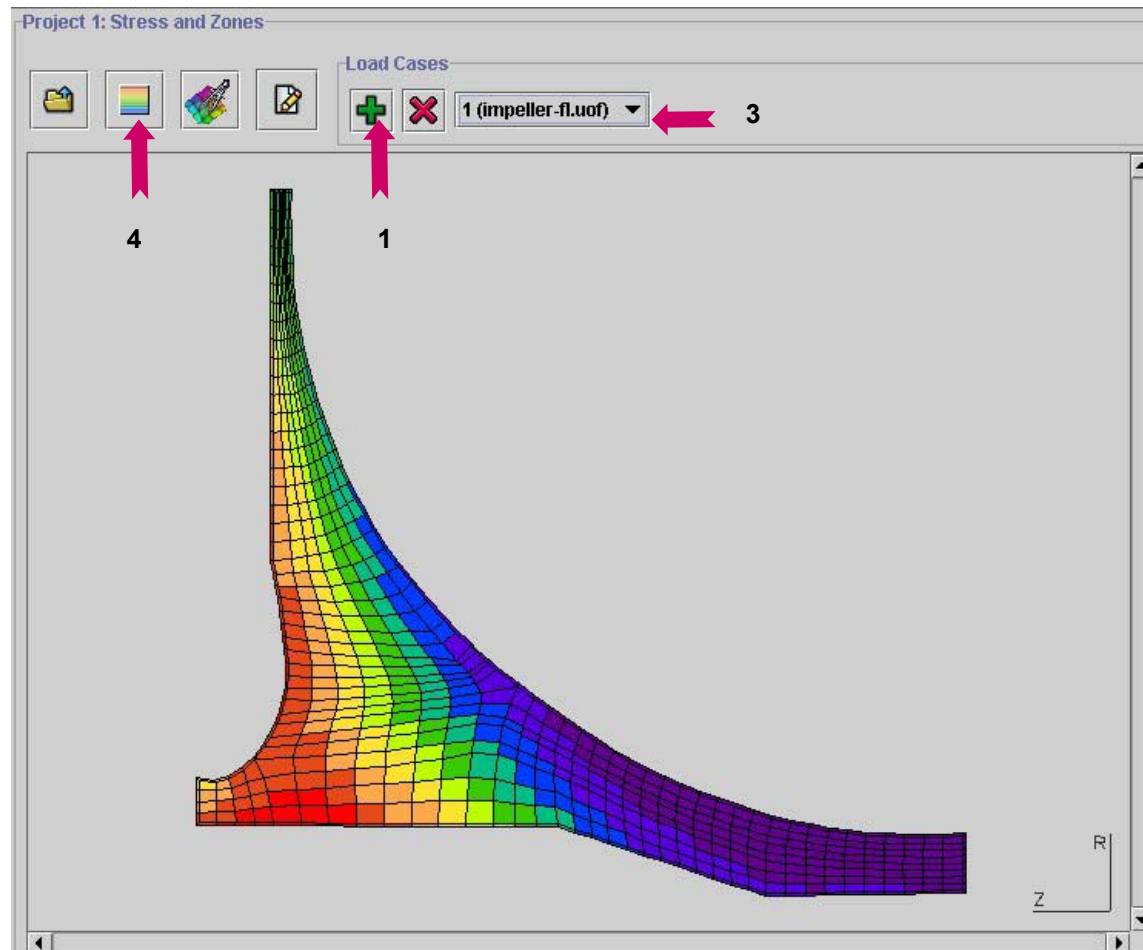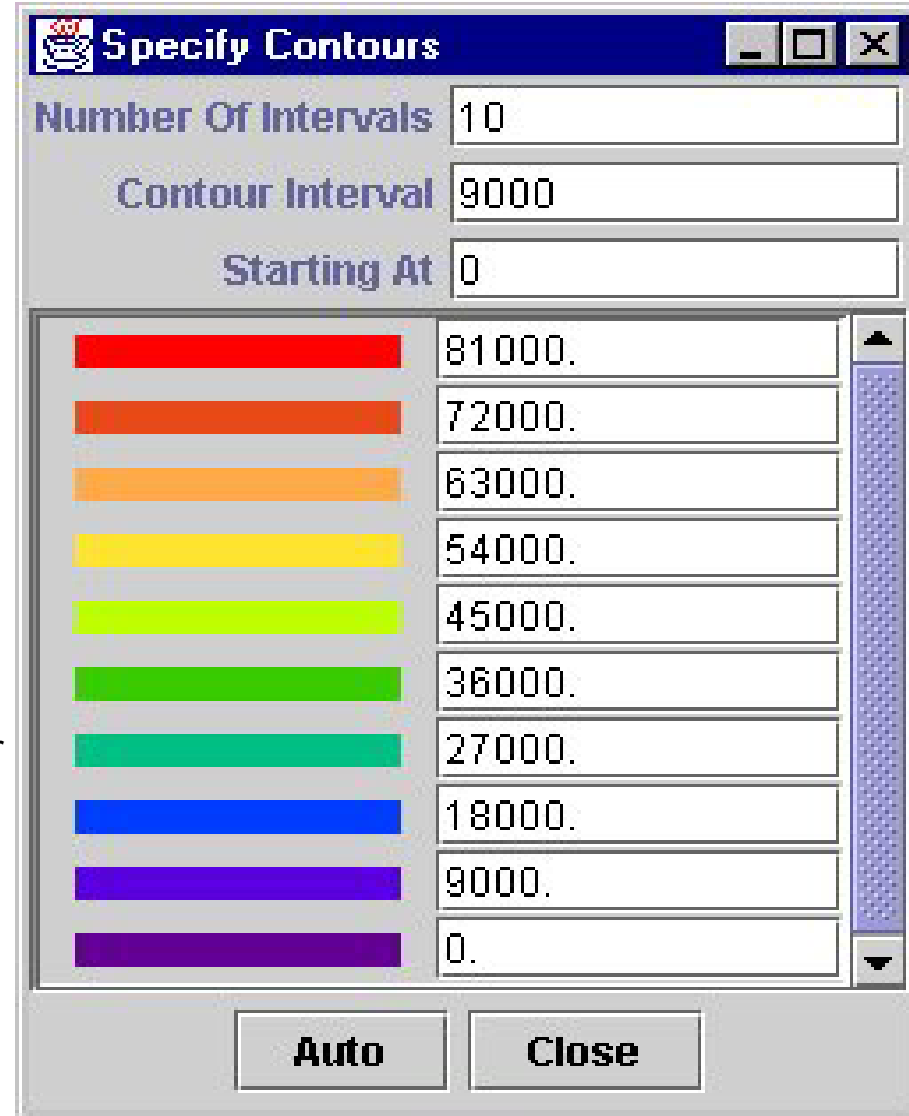
To Set Contour Values:

1. Click on [icon] near the top part of the project window and this will appear.

2. Click in the **Number of Intervals** box to change the number of contour divisions (the default is 10). Note: you must hit the "return" or "enter" button on your keyboard for changes to take effect.

3. The **Starting At** box lets you set the lowest stress value at which contours will be shown (hit "return" or "enter" to see changes).

4. The **Contour Interval** lets you set the size of the stress interval between any two contour values. Notice that it is currently set to 9000, so the difference between each color is exactly 9000 units.

5. The **Auto** button can be used to automatically set contour values for a given number of intervals and starting value. Click on **Close** upon completion.

Specify Contours

Number Of Intervals	10
Contour Interval	9000
Starting At	0

Color	Value
(red)	81000.
(red-orange)	72000.
(orange)	63000.
(yellow)	54000.
(yellow-green)	45000.
(green)	36000.
(teal)	27000.
(blue)	18000.
(violet)	9000.
(purple)	0.

Auto   Close

Zone creation and editing is initiated by clicking the *button (see page 2 of this section). Zones are defined and modified using the Zone Editor which consists of five distinct input sections:* ***Zone****,* ***Elements****,* ***Crack****,* ***Plate****, and* ***Properties****. Instructions for using these GUI functions are provided in the following pages.*

The Zone Section

To Create or Edit Zones:

1. Click on [icon], located at the upper left corner of the project window (see page 4 of this section).

To Add a Zone:

1. Click on [icon] in the **Zone** section of the Zone Editor Window.

In the following pages, we will examine each section of the Zone Editor.
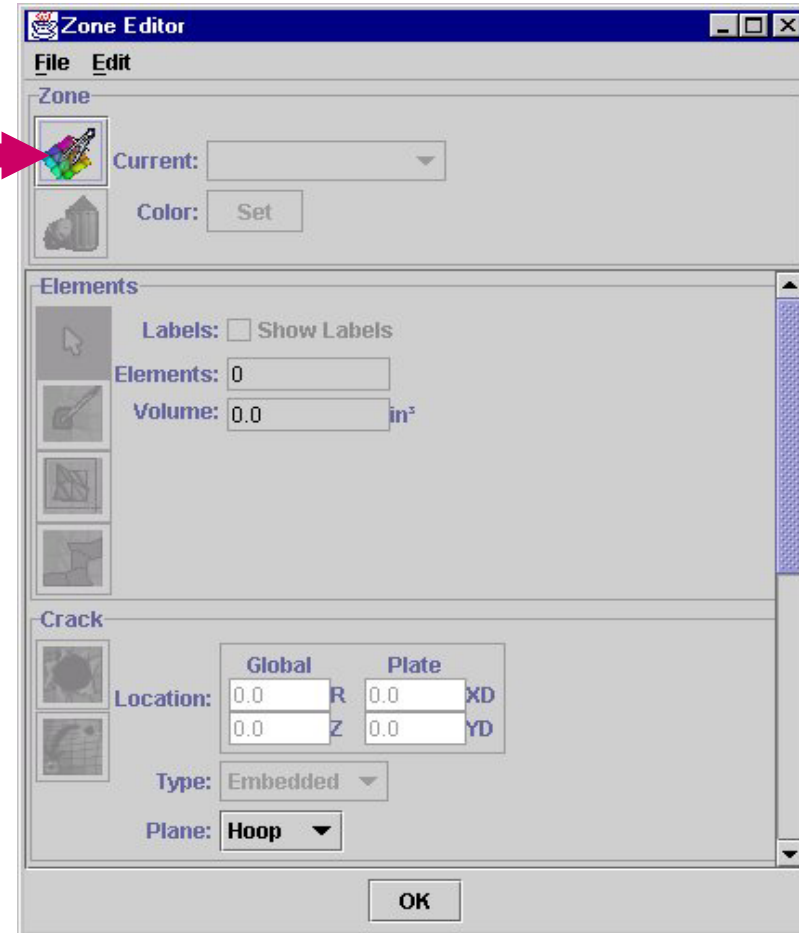
The Zone Section (cont.)

Add/Delete a Zone
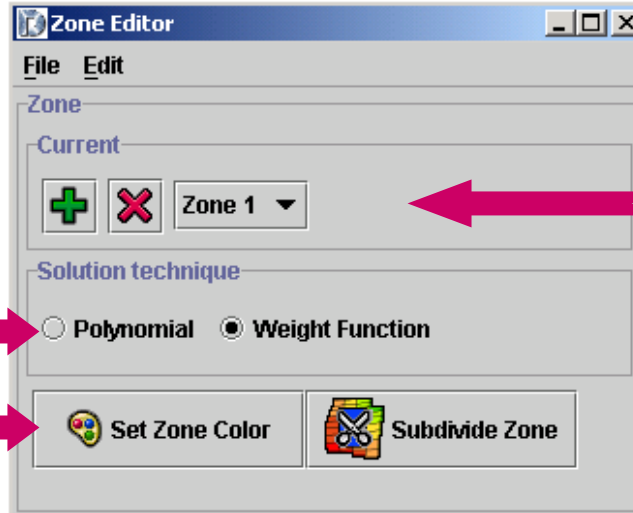
Select Solution Technique

Sets Zone color on the model

Current keeps track of the number of zones you have added. Clicking on the Zone number will show you a list of these zones, and clicking on a Zone number on the list will allow you to select that zone.

About Setting Colors:

1. After clicking on the **Set current** button in the *Color* option, this window will appear.
2. You may select the color for a zone either by name (click in the *By Name* section), or by color (click on the color in the *By Color* section). You may also choose the *Transparent* option.
3. A preview of the zone and the text is shown in the *Preview* section.
4. If you wish to have a custom color, you may choose the "RGB" (Red-Green-Blue) tab instead of the "Select Colors" tab to create a custom color.
5. Click **OK** when finished or **Cancel** to quit.
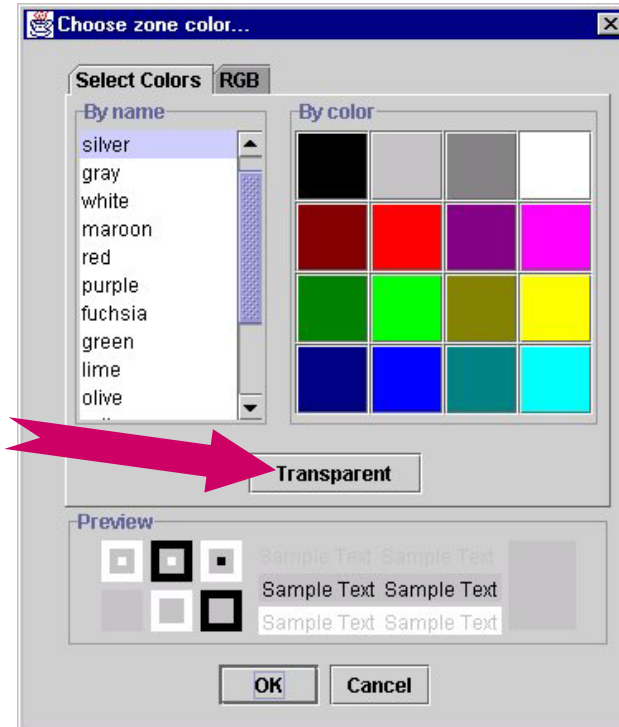
The Elements Section

During zone creation, the user must specify the elements associated with the zone. Several element selection methods are available as described below.

1. Default Cursor: Click on this icon to deselect any zone-editing tools.

2. Single Elements: Click on this option to select individual elements one at a time. Left-click on an element to select it, right-click on an element to deselect it.

3. Elements by Box: Choose this option to select a group of elements. Left-click on the graph, drag the mouse over the elements you wish to select, then left-click again to select all the elements touching the box. If you wish to deselect elements, right-click on the graph, drag the mouse over the elements you wish to deselect, then right-click. All of the elements touching the box will be deselected.

4. Elements by Contour: To select all of the elements that fall within a specific stress contour, click on this option, then left-click on an element with the stress contour value that you wish to select. To deselect all elements of a specific contour, click on this option, then right-click on one of the elements.

5. Labels: Click on the **Show Labels** box to display element numbers in a currently selected zone.

6. Elements and Volume: DARWIN™ automatically counts the number of elements you select and also calculates the total volume of all all the elements. These fields are provided for reference.
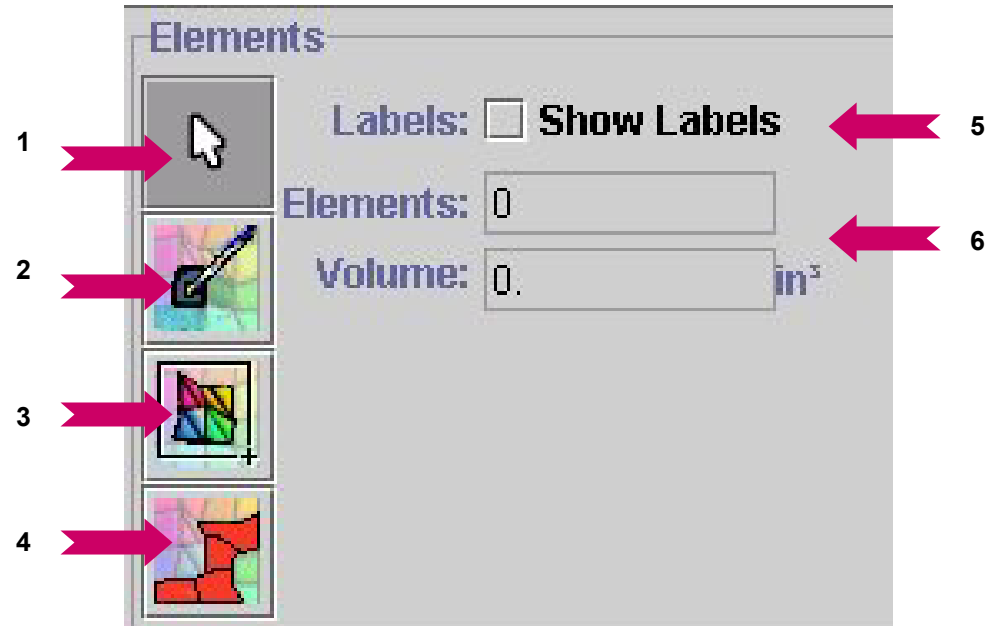
The Crack Section

During zone creation, the user must specify a crack location for fracture mechanics computations.
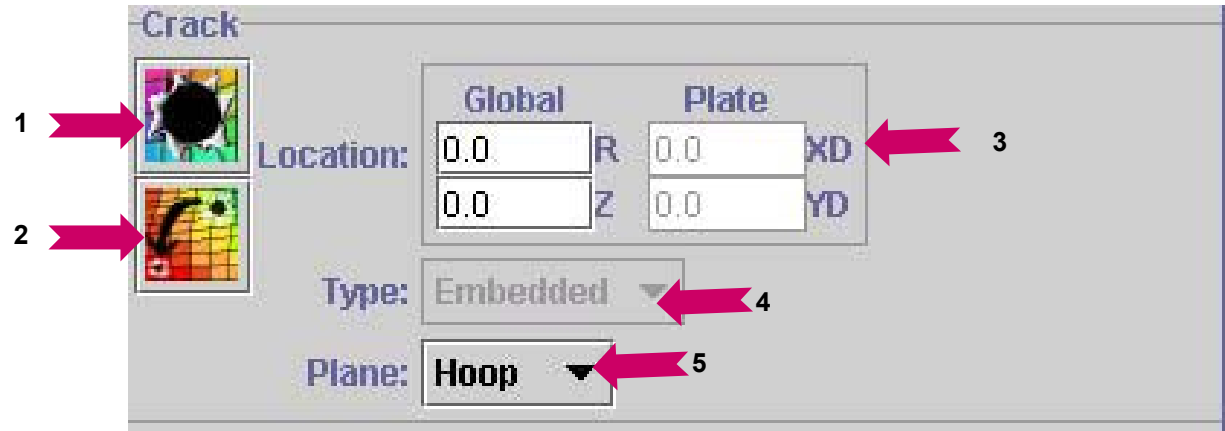
1. Set Crack Location: Click on the *Set Crack Location* button, and left-click on a point within the zone you have selected (Note: You must define the zone and associated elements before using this option).

2. Set Crack Location to Highest Stress: This button tells DARWIN™ to find the point of greatest stress in a selected zone and places the crack location at that point.

3. Location: Crack coordinates are displayed in *Global* (with respect to the entire model) and Plate coordinates under *Plate* (definition is described on the next page). You can also enter global crack coordinates manual (type values in the boxes located below the word *Global*).

4. Type: Available crack types are surface, corner, and embedded. Embedded crack is the default option. The crack type cannot be changed until the plate is defined (see next page for plate definition).

5. Plane: The crack plane is Hoop by default. It may be changed to Radial or Axial by clicking on the pull-down menu. This changes the stress component displayed in the model.

The Plate Section

The rotor model geometry in the vicinity of the zone is idealized as a rectangular plate with user defined dimensions. Plate definition is described below.

1. Draw a Plate: Click on this icon to begin the creation of a plate. Left-click on the model in the location where you wish to place one end of one side of the rectangular plate. Then (without holding down the mouse button), move to the end location of the first side and left-click. Next, move the mouse perpendicular to the side you have just created and left click to form the rectangular plate. Plate dimensions should be adjusted so that plate edges are near the edges of the model geometry.

2. Rotate Gradient 90°: Clicking on this icon will allow you to rotate the stress gradient 90 degrees. *NOTE: DARWIN will only perform this process for Embedded cracks.

       *Tip: (for any crack type) Holding <ctrl> and a mouse button down will allow free rotation of the plate.

3. Visibility: If you wish to have the plate visible at all times (including during editing of other zones), left-click in the small box next to this option.

4. Plate Viewer: DARWIN will allow you to see plate details if you click this option
                            (see following page for details).

The Plate Section (cont.)

The Plate Viewer

1. Clicking **Plate Viewer:** ☑ **Show Viewer** will cause the "Plate Viewer" window to appear.

2. Clicking on **>>** expands the viewer window in (1) to look like the viewer window in (3).

3. This version of the "Plate Viewer" window shows the exact coordinates of the plate origin, plate dimensions, and the current plate angle, as well as the coordinates of the current crack location (given globally and with respect to the plate).

4. Clicking on **<<** retracts the viewer window in (3) to look like the viewer window in (1).
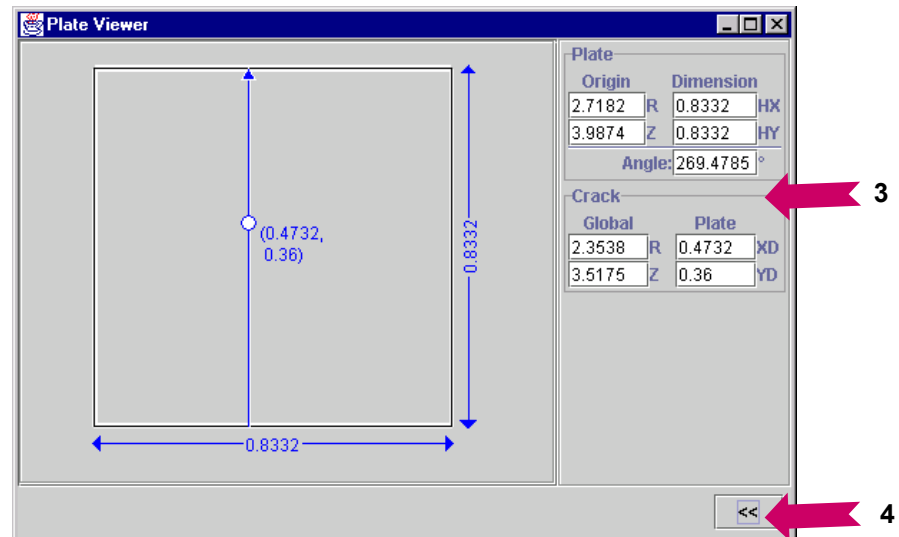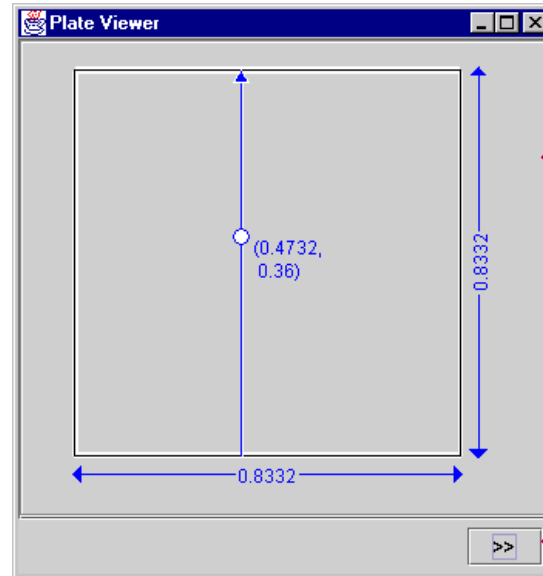
Inspection Settings - Embedded Cracks

You can assign an inspection schedule to both the Pre- and Post-Transition stages of an embedded crack.

To assign an Inspection Schedule:

1. Pre-Transition - Click on and then click on an available inspection schedule given in the **Available Inspections** box.

2. Post-Transition - Click on and then click on an available inspection schedule given in the **Available Inspections** box.
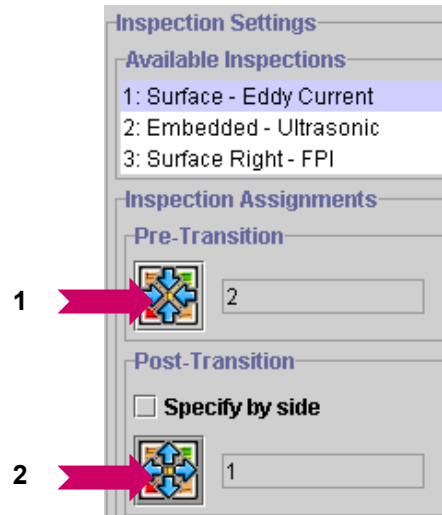
To specify Post-Transition inspection schedules by side:

1. Click on the ☑ Specify by side box.
2. Next, click on 🔙, 🔜, ⬆️, or ⬇️ and then click on an available inspection schedule given in the **Available Inspections** box.
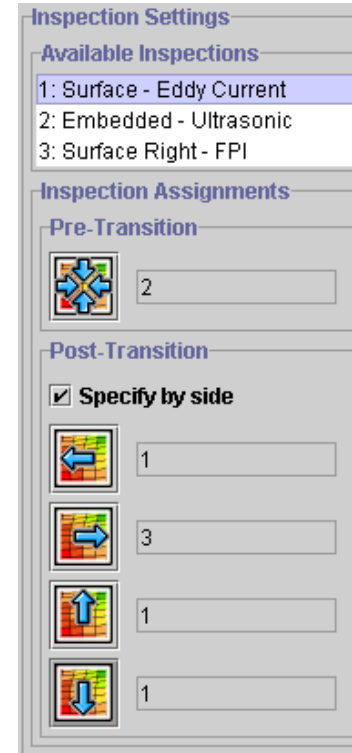
Inspection Settings - Surface & Corner Cracks

By definition, surface and corner cracks have already transitioned. A single Post-Transition inspection schedule is used to define the inspection.

To assign an Inspection Schedule:

1. Click on , and then click on an available inspection schedule given in the **Available Inspections** box.
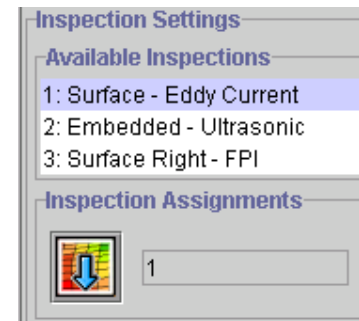
The Properties Section

*This is the last section of the Zone Editor. After setting up these options, click **OK** at the bottom of the Zone Editor Window.*

Properties

1 → Material: **1: Ti64 props. in air: Paris FCG eqn., R=0.25** ▼

2 → Defect Distribution: **1: post 95: 12inmz Billet: 3fbh forging** ▼

3 → Inspection Schedule:
1: sample

4 → Volume Multiplier **1.0** Value

5 → Life Scatter Factor: **1.0** Median
0.0 COV

1. Material: Here you will see Material Properties files that you selected in the previous part of your assessment definition. Select one of these to define the material properties for each zone (clicking on the drop-down menu allows you to switch between files).

2. Defect Distribution: Similar to the Material option, you must select a defect distribution for each zone.

3. Inspection Schedule: Select an inspection schedule for each zone (you can select multiple schedules by holding down <ctrl> and clicking on the schedules).

4. Volume Multiplier: Input a multiplier as desired (see DARWIN™ Users Manual for instructions). A default is provided.

5. Life Scatter Factor: Input Median and Coefficient of Variation values if desired.

## Zone Creation Examples: The Embedded Zone

Here is the sequence of steps for creating an embedded zone:

1. Create a zone - Click on in the **Zone** section of the "Zone Editor Window".

2. Define elements - Define elements in the zone using the **Elements** section tools (see page 9 of this section). For an embedded zone, select elements that do not border the model geometry.

3. Define crack location - Choose a crack location using the **Crack** section tools (see page 10 of this section).

4. Define a plate - Draw a plate about the crack and elements using the **Plate** section tools (see page 11 of this section).
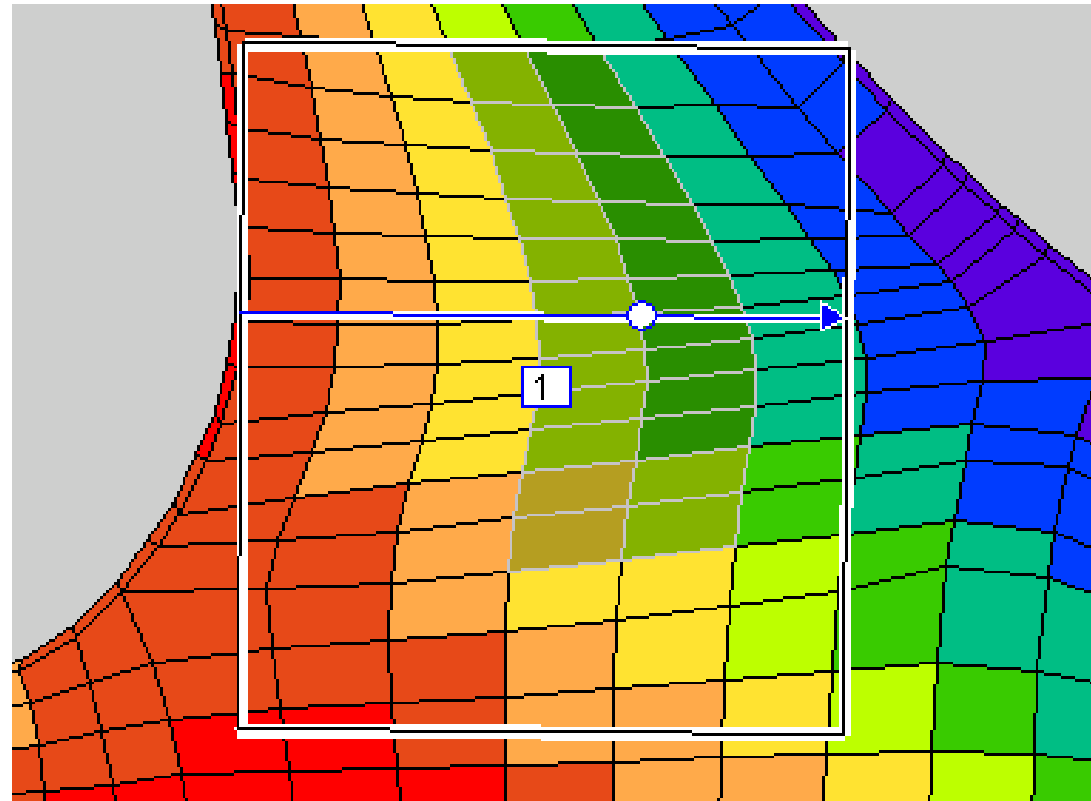
<u>Zone Creation Examples:</u> The Surface Zone

Here is the sequence of steps for creating a surface zone:

<u>1. Create a zone</u> - Click on [icon] in the **Zone** section of the "Zone Editor Window".

<u>2. Define elements</u> - Define elements in the zone using the **Elements** section tools (see page 8 of this section). For a surface zone, select at least a few elements that border the model geometry.

<u>3. Define crack location</u> - Choose a crack location on one of the border elements using the **Crack** section tools (see page 9 of this section).

<u>4. Define a plate</u> - Draw a plate about the crack and elements using the **Plate** section tools (see page 10 of this section).
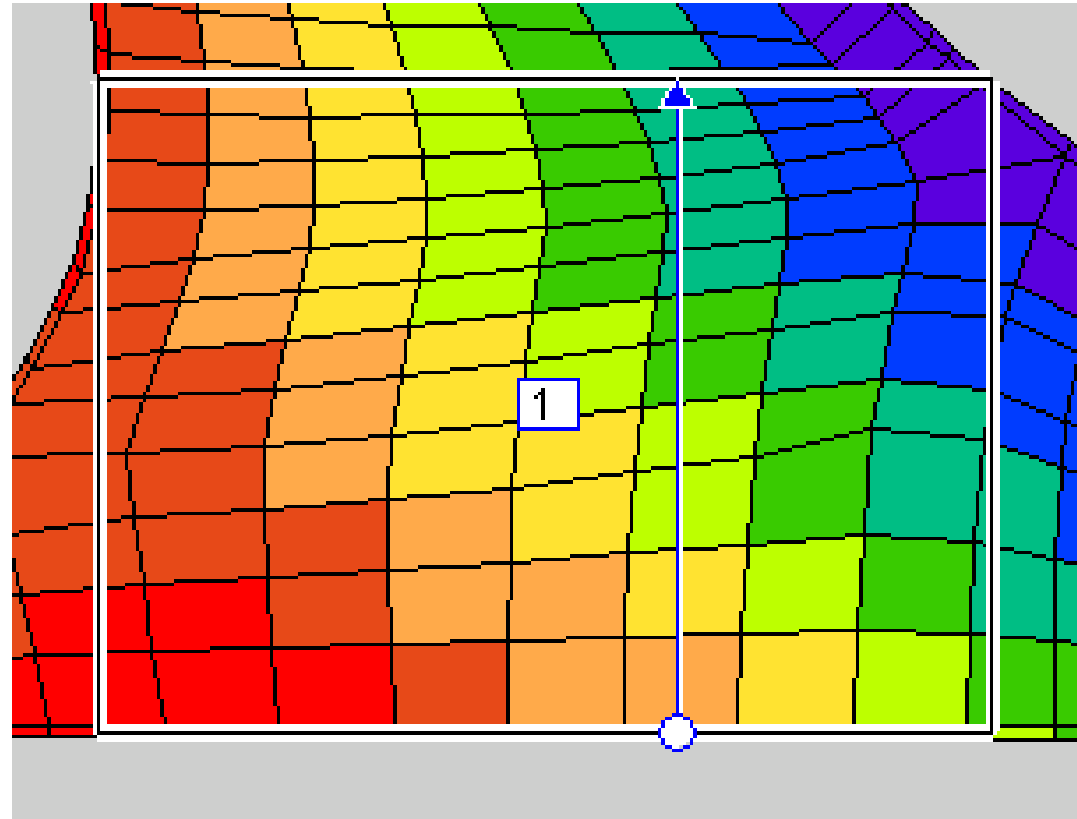
Zone Creation Examples: The Corner Zone

Here is the sequence of steps for creating a corner zone:

1. Create a zone - Click on [icon] in the **Zone** section of the "Zone Editor Window".

2. Define elements - Define elements in the zone using the **Elements** section tools (see page 8 of this section). For a corner zone, select elements that are near a corner of the model geometry.

3. Define crack location - Choose a crack location near the corner of the model geometry(see page 9 of this section).

4. Define a plate - Draw a plate about the crack and elements using the **Plate** section tools (see page 10 of this section). Once a plate is drawn, select "corner" as the crack type in the **Crack** section. See the next page for additional details.
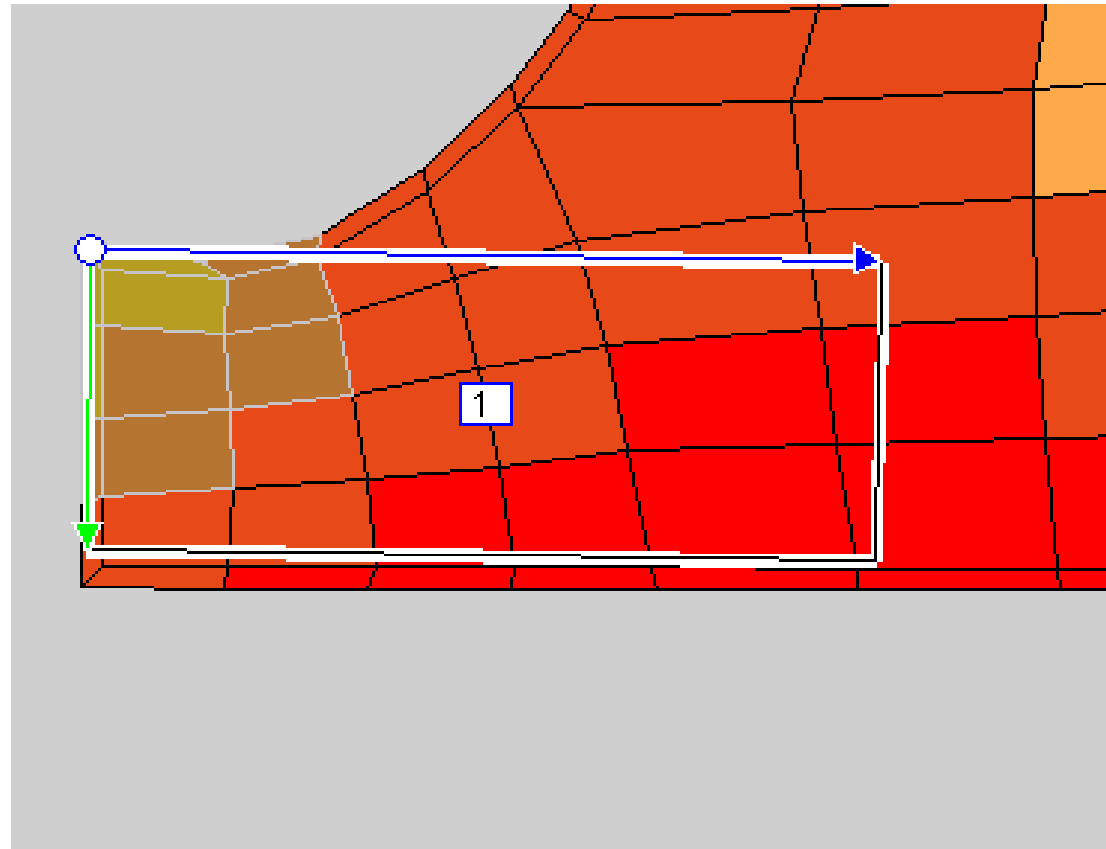
DARWIN TIPS:

Creating Corner Cracks

The following is a good rule of thumb for creating a plate for corner cracks:

1. For upper-left side corner cracks, click on the crack, move the mouse pointer down to form the first side, click, and then move the mouse pointer to the right and click.
2. For upper-right side corner cracks, click on the crack, move the mouse pointer left to form the first side, click, and then move the mouse pointer down and click.
3. For lower-left side corner cracks, click on the crack, move the mouse pointer right to form the first side, click, and then move the mouse pointer up and click.
4. For lower-right side corner cracks, click on the crack, move the mouse pointer up to form the first side, click, and then move the mouse pointer left and click.
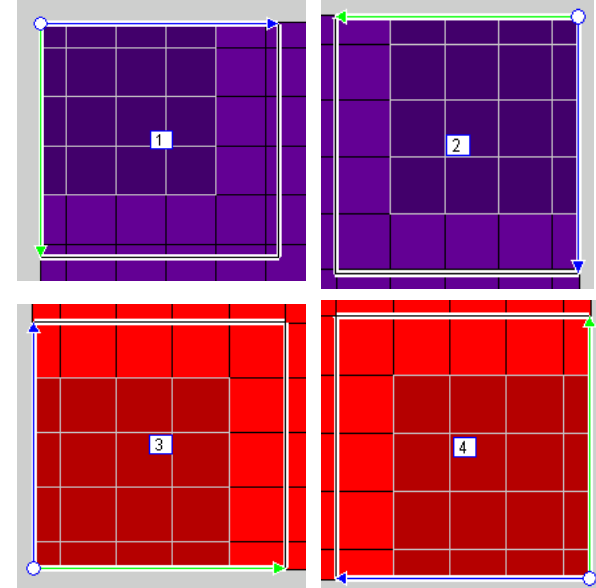
Adjusting the Gradient Angle

Another feature of plates defined about a corner crack is the ability to adjust the gradient angle.

1. Once a plate is defined, left-click and hold on the green arrow head and move the mouse pointer to adjust it.

Before Adjustment **After Adjustment**
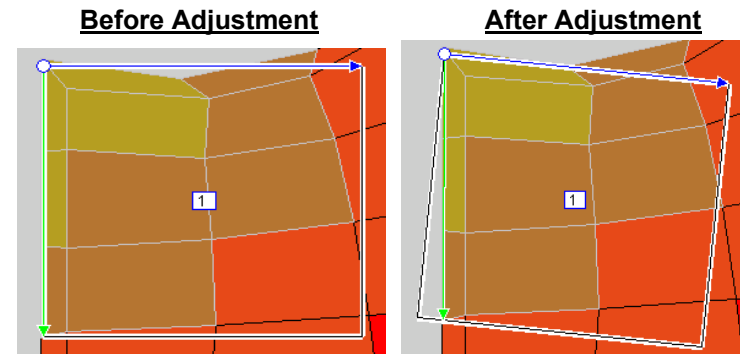
Darwin™ does not limit the user to creating zones with the default elements on the model geometry. For a more precise method of zone creation, you can refine the elements of the rotor model by either subdividing the elements or "onion-skinning" the rotor model.

To start element refinement, click on [icon] **to open the Refinement Editor window.**

To Refine Elements:

1. Select the elements you wish to refine by using the element selection tools in the Refinement Editor window.

2. Select "Subdivide" or "Onion Skin". For "Onion Skin", select the amount in mils (thousandths of an inch) of the desired thickness. You can view a sample of the element refinement at the center of the Refinement Editor Window.

3. Click [Apply] to make the changes. For convenience, click [↩] to "undo" these changes or click [↪] to "redo" the changes in element refinement.

4. Click [Close] when finished.
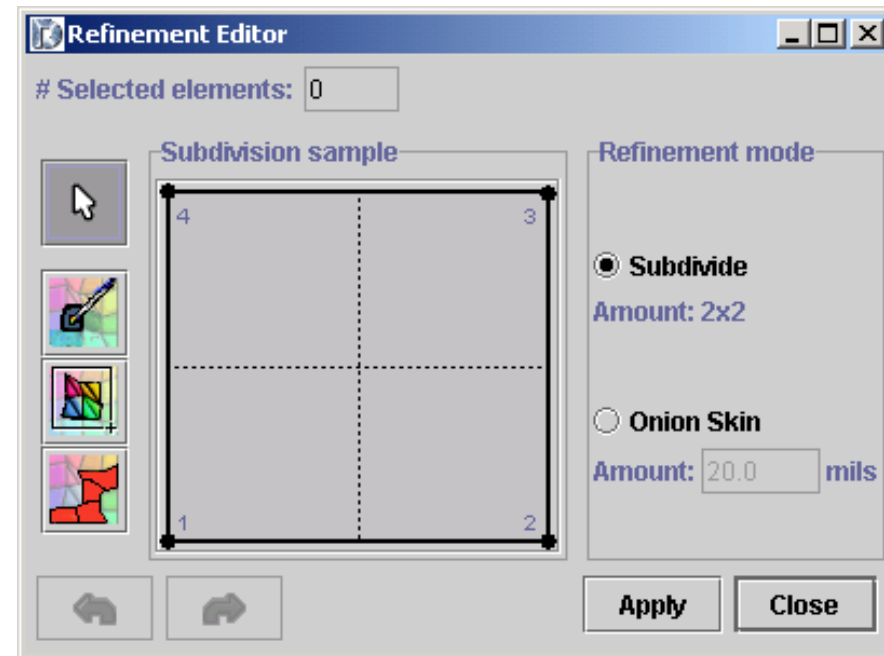
Element Refinement Examples:

Element Subdivision example:

Before Element Subdivision (9 elements) → After Subdivision (12 elements)
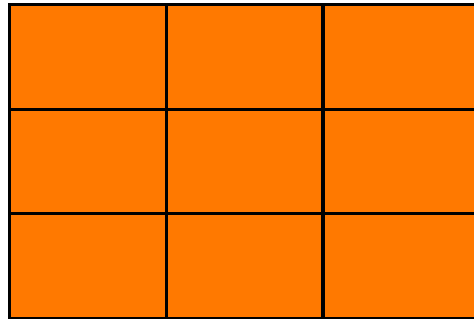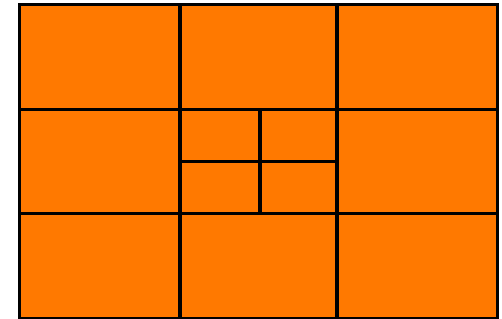
"Onion-skinning" example:

Before "Onion Skinning" process (Surface elements) → After 10 mil "onion skin" application
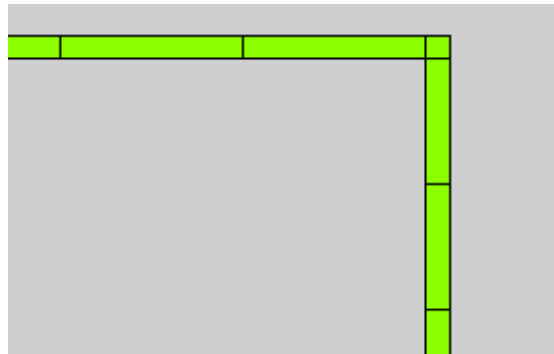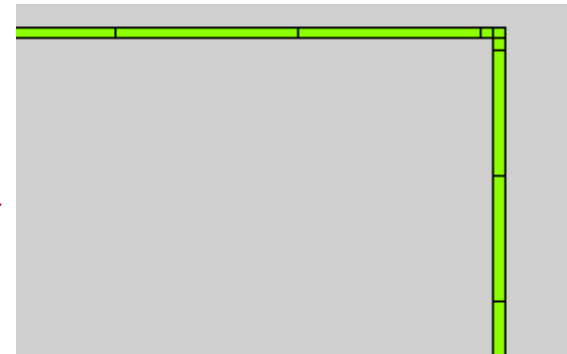
Zone Refinement

Disk failure probability predictions are dependent on the zone discretization selected by the user. In general, as the number of zones is increased, the disk failure probability decreases, and the computation time increases. For some zones, the contribution to disk risk is insignificant, so further discretization is not necessary.

Darwin™ provides an efficient procedure for identifying and subdividing zones that have a significant contribution to disk risk:

1. View Zone Risk Contribution Factors (RCF) to identify risk critical zones.
2. Subdivide risk critical zones using the zone subdivide button in the Zone Editor window.
3. Re-compute risk of newly created sub-zones.
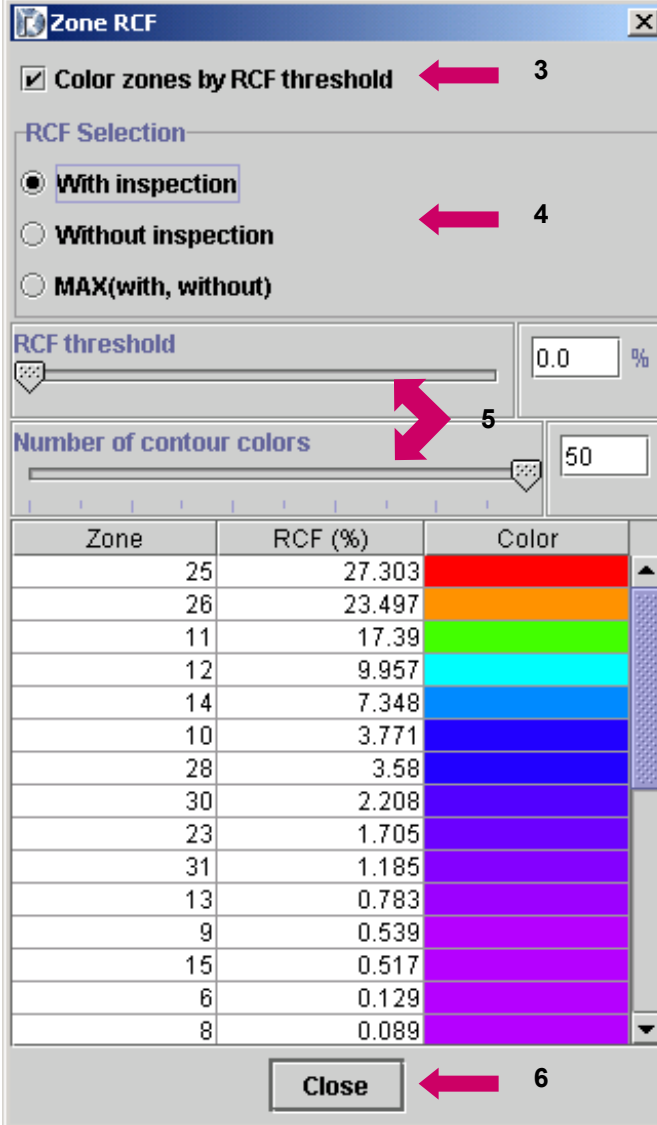
This procedure is outlined in detail on the following pages.

Zone Refinement (cont.)

1. To view Zone Risk Contribution Factors (RCF):

1. Open your project's *.ddb file (created after an initial Darwin™ run).
2. Under the Assessment Definition, click on Stress and Zones, and then click on the (RCF) icon. The Zone RCF window will appear.
3. Click in the **Color zones by RCF threshold** box to start using the RCF utility. Click again if you wish to turn off the RCF utility.
4. Select the data for RCF calculation to be either the Probability of Fracture With Inspection, Without Inspection, or the greater of the two values (MAX).
5. Adjust the **RCF threshold** and the **Number of Contour Colors** by dragging the appropriate slider bar.
6. Click on **Close** to save the changes and exit the Zone RCF utility.

Zone Refinement (cont.)

Zone Risk Contribution Factors Viewing Example:

Before Activation of Zone RCF Utility

After Activation of Zone RCF Utility

RCF identifies zones with highest risk
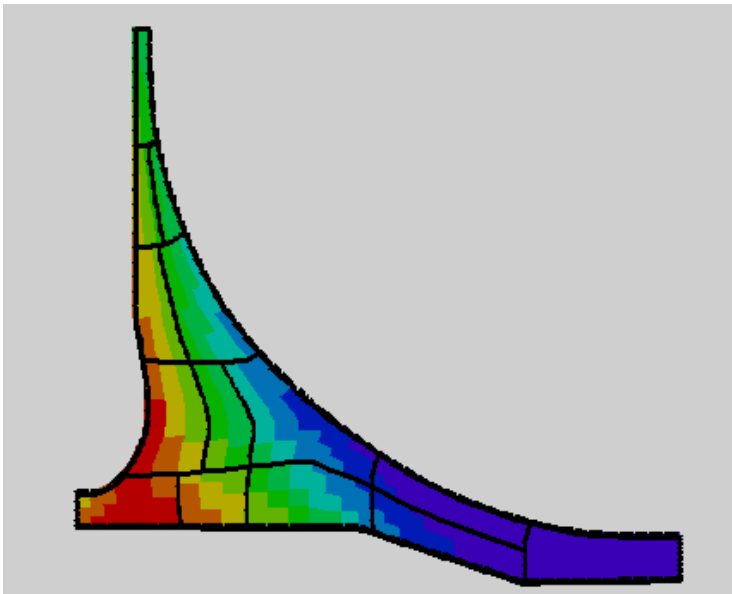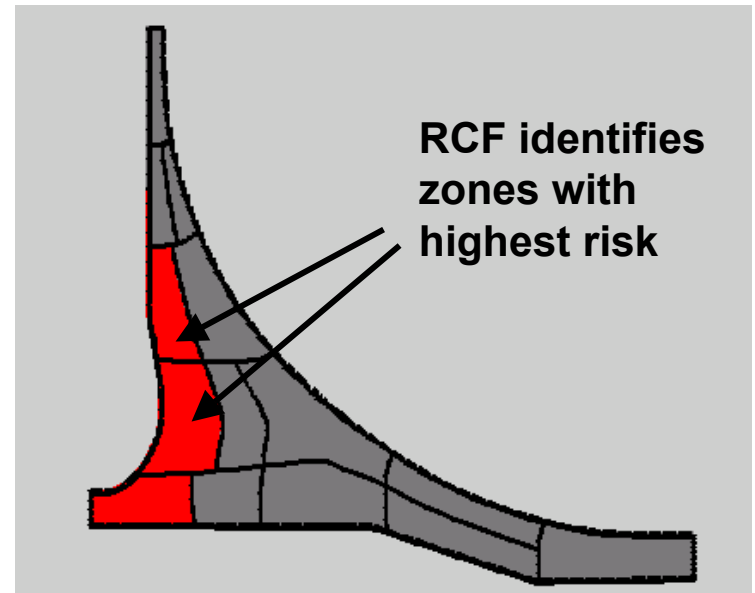
Indicates Zone RCF Utility "OFF".

**Indicates Zone RCF Utility "ON".**
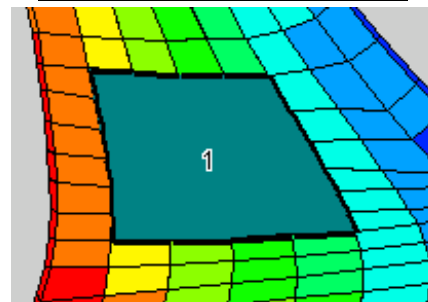
Zone Refinement (cont.)

2. To Subdivide a zone:

1. Click on the [icon] button in the Stress and Zones section to activate the Zone Editor window.

2. Select the zone you wish to subdivide and click on the [Subdivide Zone] button.

3. Confirm zone subdivision by clicking **OK**.

NOTE: Zone subdivision will occur at the centroid of the zone. There is no "undo" option.

EXAMPLE:
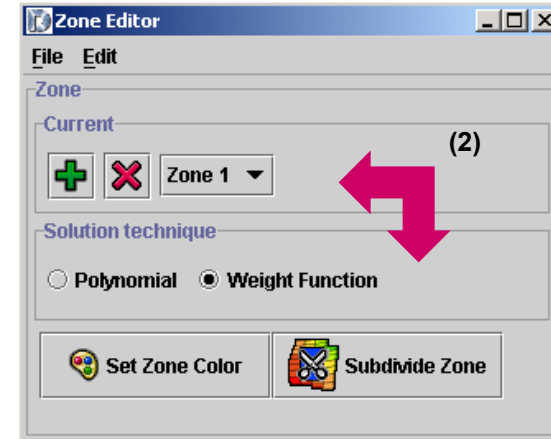
Before Zone Refinement

After Zone Refinement
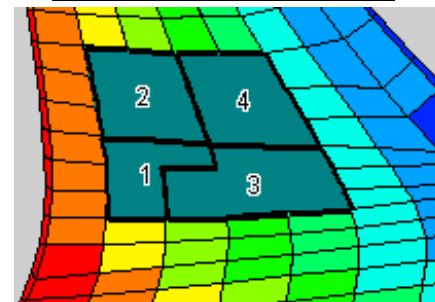
Zone Refinement (cont.)

Zone Refinement concludes with restarting the Darwin™ analysis. To increase the execution efficiency, Darwin™ uses information found in the results database file from the previous run.

3. To re-compute risk:

1. In the Analysis Options section of the Assessment Definition, click on the Select option under the **Restart results database** heading.

2. Click [Browse...] to find the results .ddb file from the previous Darwin™ execution.

Surface Damage - Gradients

Gradients are defined by inserting stress points (X,σ). A polynomial fit and associated equation based on these stress points is shown.
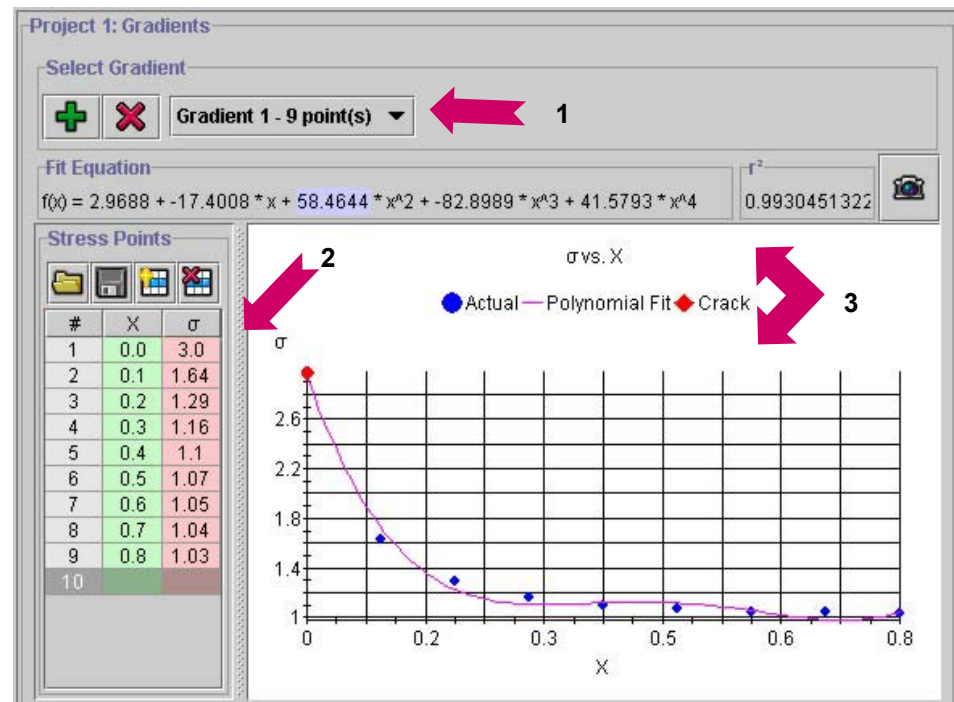
To add a gradient:
1. Click [+] .

To edit a selected gradient:
2. Enter stress points (X,σ) in the Stress Points table.

• To add another point, click [icon].
• To remove a selected point, click [icon].
• To save the current stress point data, click [icon].
• To load stress point data from a file, click [icon].

3. Darwin™ displays the fit equation that is generated by the stress point data. The plot on the right hand side of the screen is a visual representation of the fit equation.


Project 1: Gradients

Select Gradient

Gradient 1 - 9 point(s) ← 1

Fit Equation

f(x) = 2.9688 + -17.4008 * x + 58.4644 * x^2 + -82.8989 * x^3 + 41.5793 * x^4

r² 0.9930451322

Stress Points

#	X	σ
1	0.0	3.0
2	0.1	1.64
3	0.2	1.29
4	0.3	1.16
5	0.4	1.1
6	0.5	1.07
7	0.6	1.05
8	0.7	1.04
9	0.8	1.03
10		

σ vs. X

Actual — Polynomial Fit ◆ Crack

Zones are created in a two-step process. The first step is to add a zone and define its characteristics (properties).

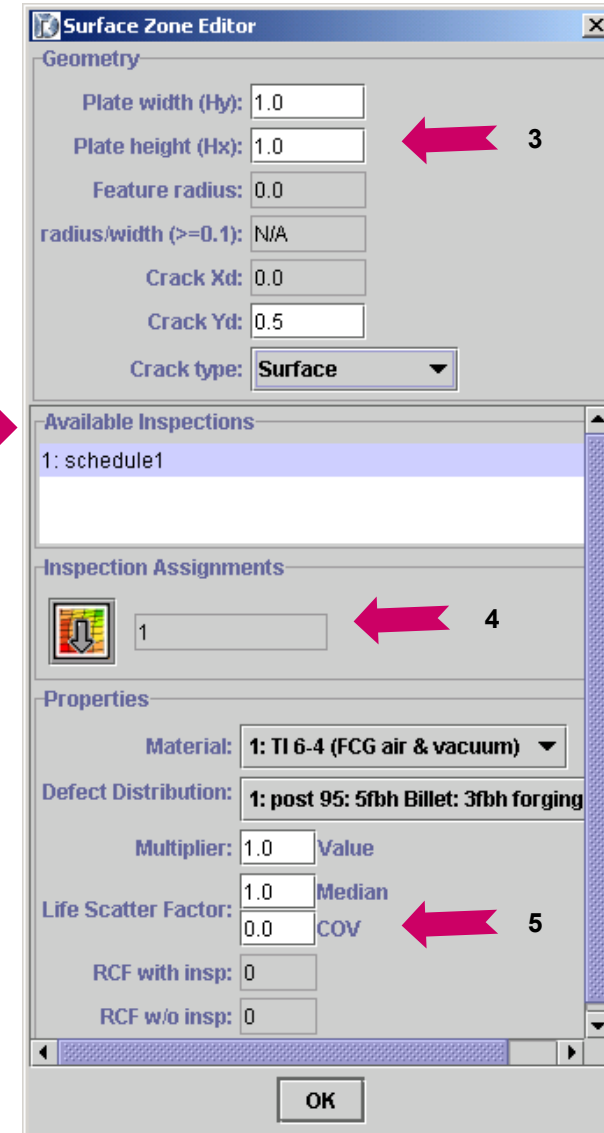**To add a zone:**
1. Click ➕ .
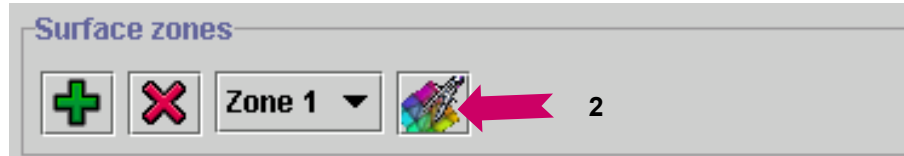
To edit zone properties:
2. Click 🖌️ . The **Surface Zone Editor** window will appear as shown:

3. Insert zone geometric properties under the **Geometry** subsection of the **Surface Zone Editor.** It is recommended that the **Crack type** be chosen first, as some information (I.e. radius/width, etc.) becomes available based on the choice. The three types of cracks available are **Surface**, **Surface at Hole**, and **Corner at Hole**.

4. Select an available inspection schedule to associate with this zone. To see instructions on how to use this feature, see the *Stress and Zones (13 of 18)* page of this tutorial.

5. Select valid properties for this zone under the **Properties** subsection of the **Surface Zone Editor**. To see instructions on how to use this feature, see the *Stress and Zones (14 of 18)* page of this tutorial.

Surface zones

Zone 1

Surface Zone Editor

Geometry

Plate width (Hy): 1.0
Plate height (Hx): 1.0   3
Feature radius: 0.0
radius/width (>=0.1): N/A
Crack Xd: 0.0
Crack Yd: 0.5
Crack type: Surface

Available Inspections
1: schedule1

Inspection Assignments
1   4

Properties
Material: 1: TI 6-4 (FCG air & vacuum)
Defect Distribution: 1: post 95: 5fbh Billet: 3fbh forging
Multiplier: 1.0   Value
Life Scatter Factor: 1.0   Median
0.0   COV   5
RCF with insp: 0
RCF w/o insp: 0

OK

The second step in zone creation is defining load steps. Load steps are defined by a step number, stress value, gradient selection, and temperature value. The three plots on this page reflect the zone state in each load step.
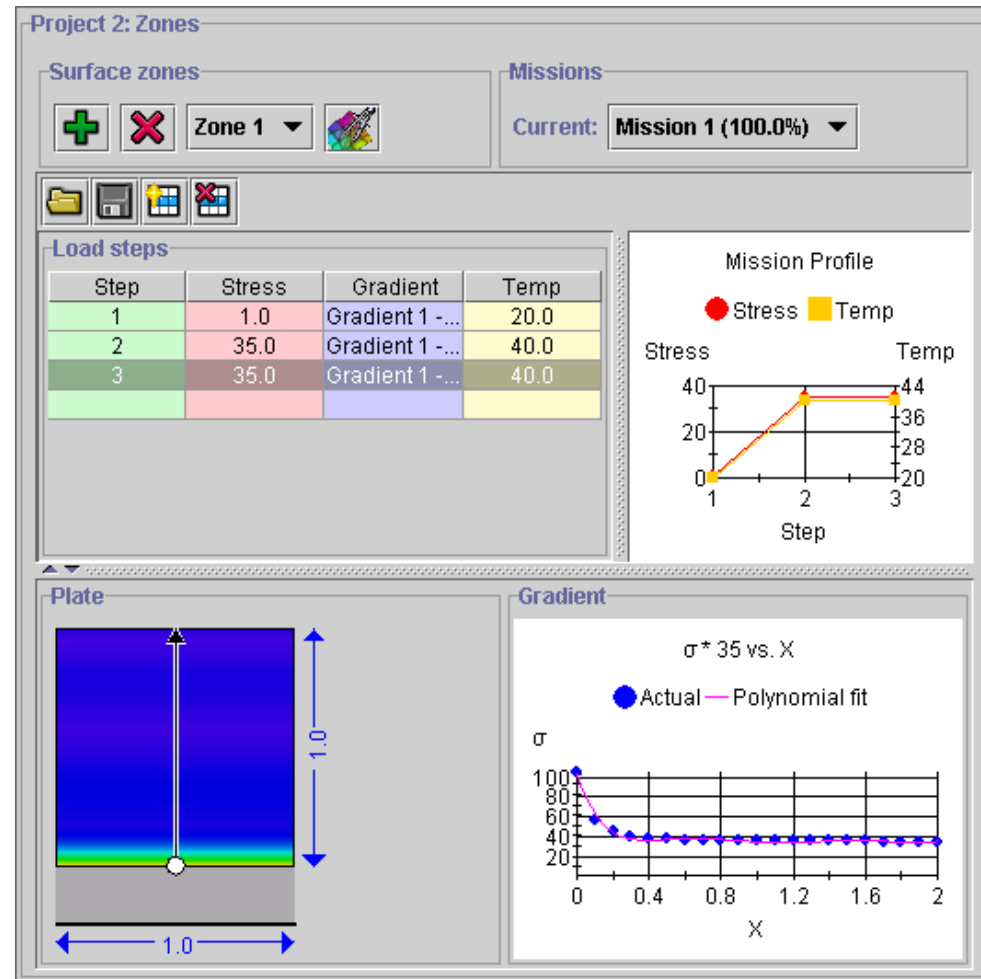
To edit load steps:

• To add another step, click [icon] .
• To remove a selected step, click [icon] .
• To save the current load step data, click [icon] .
• To load load step data from a file, click [icon] .

View Completed Zones

The completed zone can be viewed in the zone editor:

• Stress and temperature versus load step
• Plate with color coded stress contours
• Stress gradient
  (to view a gradient, click on it in the "Load Steps" section)

Probability Method

You can select Monte Carlo method or Importance Sampling for probabilistic computations. You can define the number of samples for each zone individually, or specify that all zones use the same number of samples. Importance Sampling should be used whenever possible for efficient analysis (see DARWIN™ User Guide for details).

1. Method: Monte Carlo or Importance Sampling. Monte Carlo simulation is very robust but computationally expensive. Select the method using the drop-down menu.

2. Data: Specify the number of samples (i.e. Monte Carlo or Importance Sampling simulations) for each zone. You can specify the number of samples for each individual zone by clicking in the table next to each zone, or you can set the number of samples in all of the zones to be the same number using the Set all button at the bottom of the screen.

Analysis Options

The final part of your assessment definition is Analysis Options.
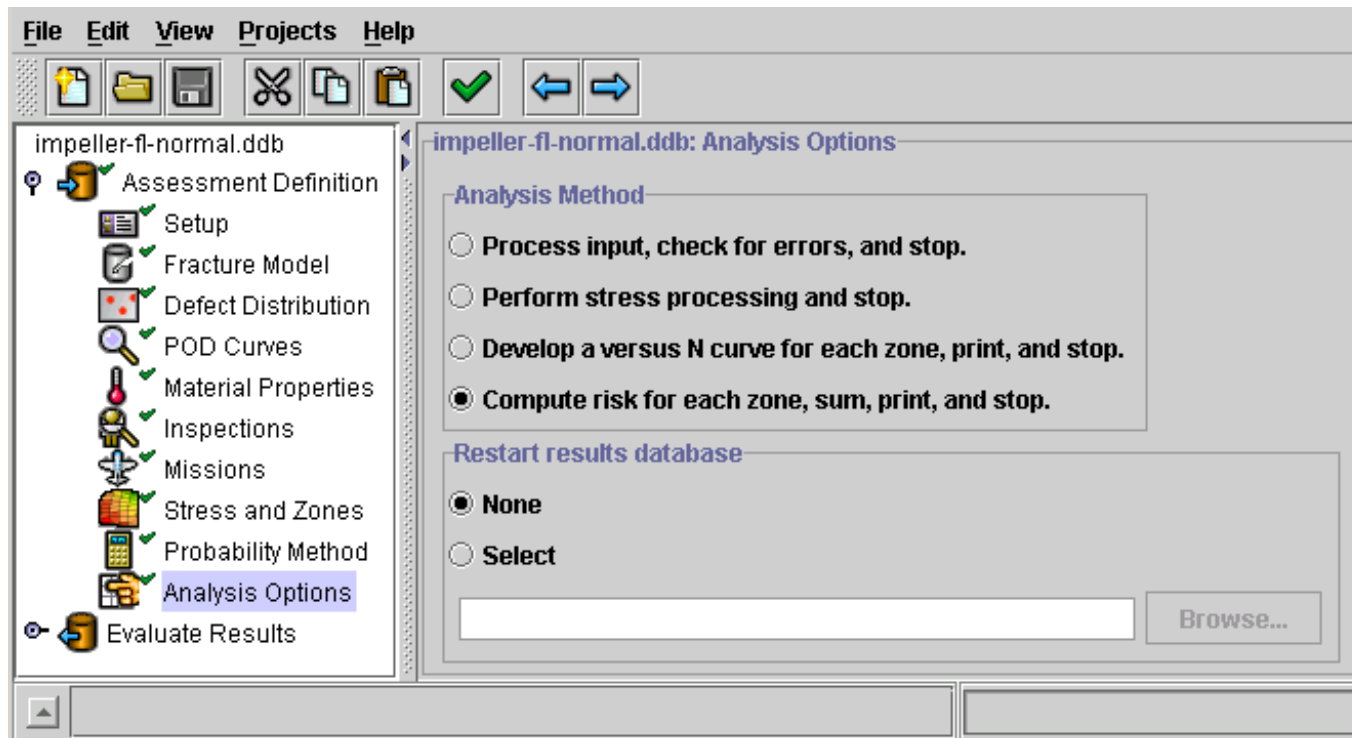
To change options:
Click on the circle next to the option you wish to use, and the circle will darken and become activated.*

*NOTE: You may choose only one Analysis Method.

Executing the
Risk Assessment Code

1. After completing Analysis Options (which concludes the Assessment Definition), you must save your project.

To Save a Project: Click on [floppy disk icon] , and choose a filename.   <u>NOTE: The file generated is a text file (*.dat).</u>

2. You must now execute the Risk Assessment Code using the file generated by your assessment definition.

To Execute the Risk Assessment Code: *See the DARWIN™ User's Guide for details.*

3. The Risk Assessment Code will generate results in the form of an XML database file (.ddb). This file can then be opened in the DARWIN™ GUI. Four new sections will appear under **Evaluate Results** (see "The Outline" near the beginning of this tutorial).*

To Open a *.ddb file: Click on [folder icon] , and select the file you wish to open. Then click on **Evaluate Results** in the outline.

Evaluate Results will help you interpret the analysis results using interactive charts and tables. The results are presented in three sections: *Risk Assessment*, *Fracture Mechanics*, and *Stress Processing*.

Evaluate Results
Job Files
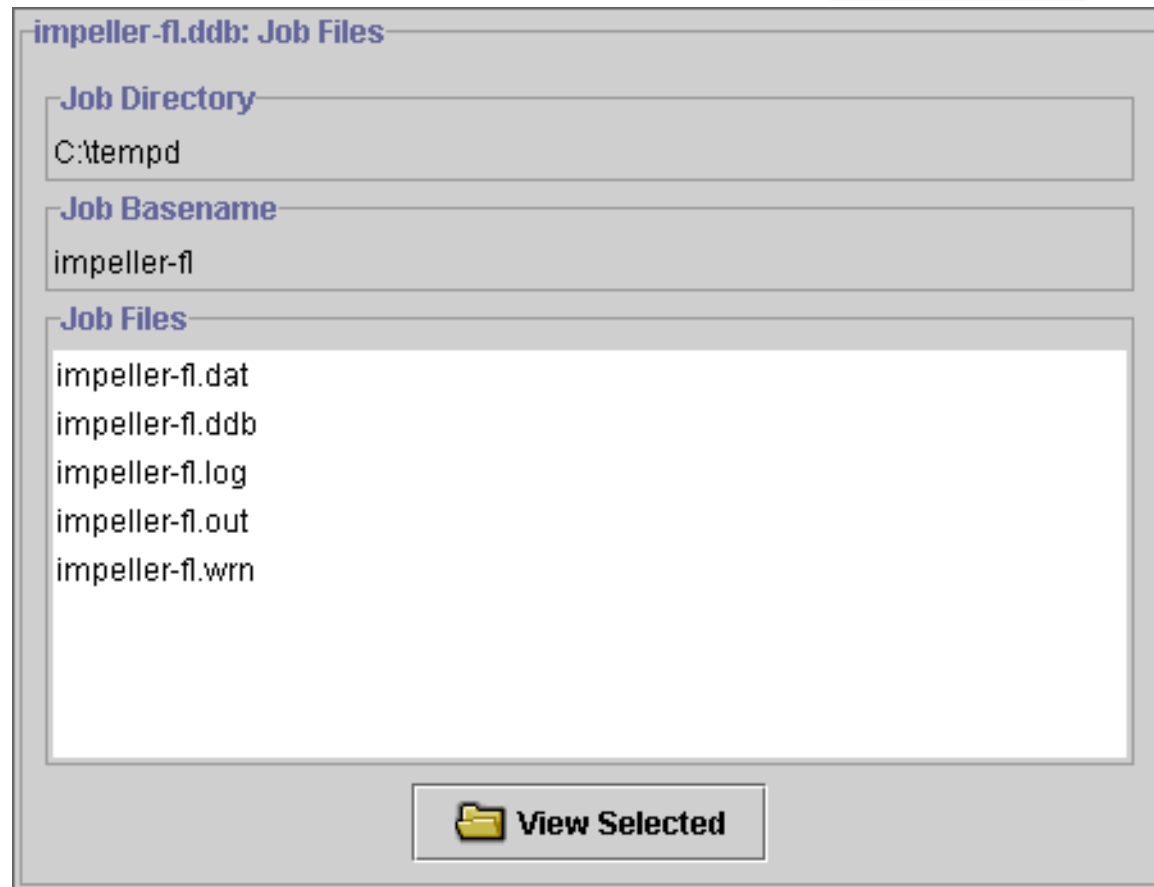*pf* Risk Assessment
Fracture Mechanics
Stress Processing
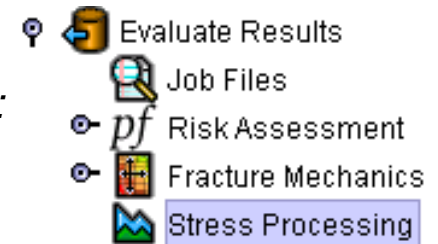
Once the output data file has been loaded into the GUI, it can be viewed by clicking on "Evaluate Results". All data files relating to your project will appear on the main screen as shown.

To view the contents of the output file, click on the *.out file, and click **View Selected**.

A summary of failure probability results for the disk can be found near the end of the file as shown.

All the data defined in the assessment definition will be re-posted onto the GUI so that you can view the choices you made prior to the execution of the risk-assessment code.

impeller-fl.ddb: Job Files

Job Directory
C:\tempd

Job Basename
impeller-fl

Job Files
impeller-fl.dat
impeller-fl.ddb
impeller-fl.log
impeller-fl.out
impeller-fl.wrn

📁 View Selected
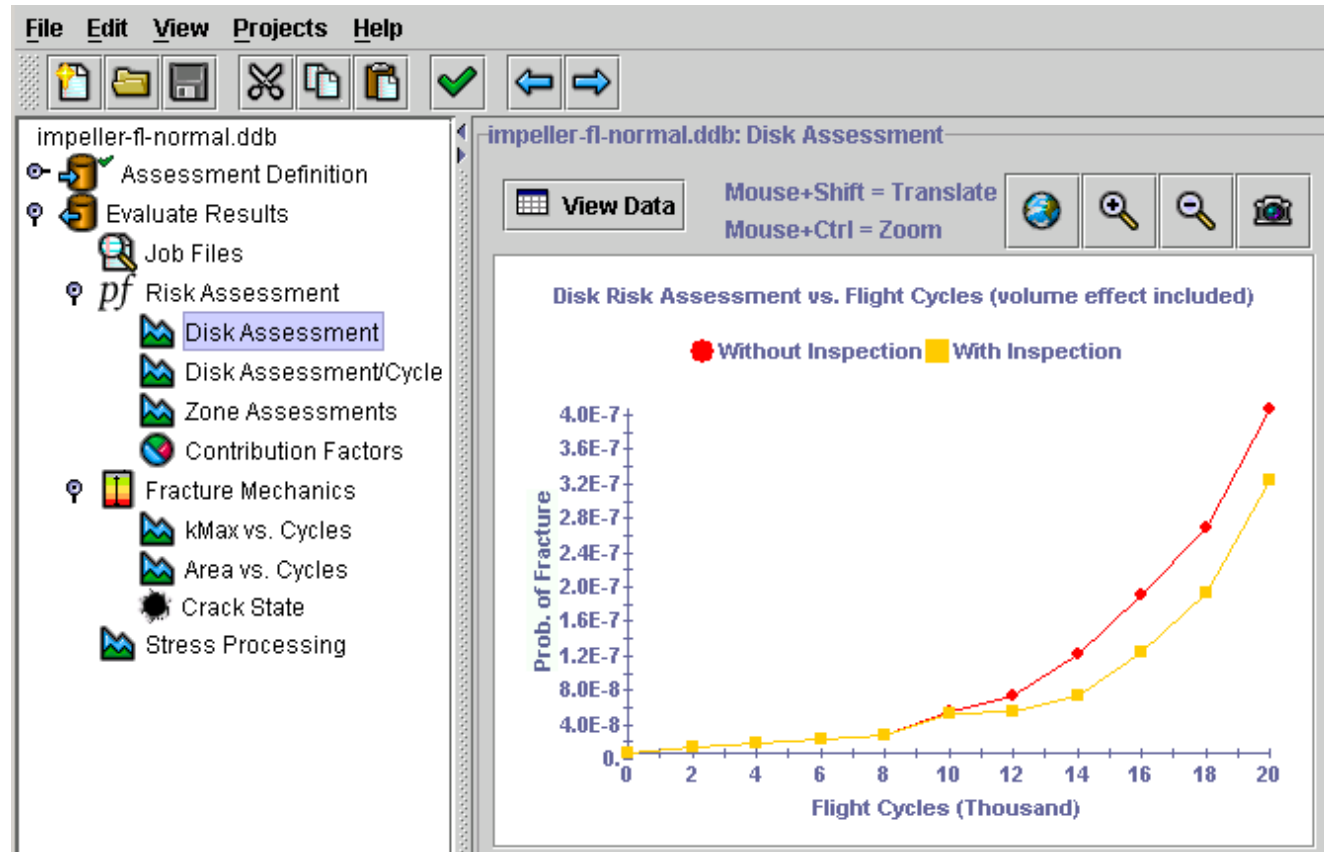
Risk Assessment

Click on this section to access the Risk Assessment plots. The first two plots, <u>Disk Risk Assessment vs. Flight Cycles</u> and <u>Disk Assessment/Cycle</u> have identical functionality, shown below.

To view the graphs:

1. Click on the graph you wish to view.

Optional:
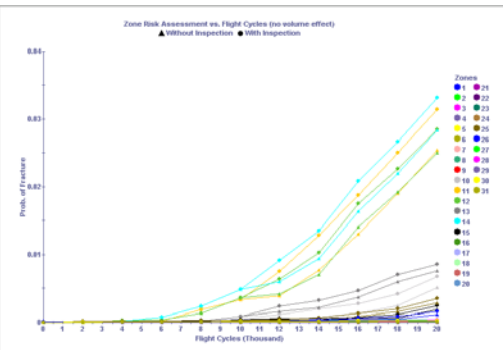To view the numeric data in tabular format, click on the 📊 **View Data** box.

Risk Assessment (continued)

The other two graphs, Zone Assessments and Contribution Factors, allow you to view the failure probabilities associated with each zone and the contribution of each zone to total (disk) failure probability, respectively. The "zones" legend on the right side of these graphs indicates the color of each zone included in the graph. To remove a zone from the graph, click on the colored circle next to the zone number.

Graph Options: (specific to these 2 graphs)

1. Click on **Hide all zones** to turn off all zones.

2. Click on **Show all zones** to turn on all zones.

3. Click on **View Data** to view the numeric data in tabular form.
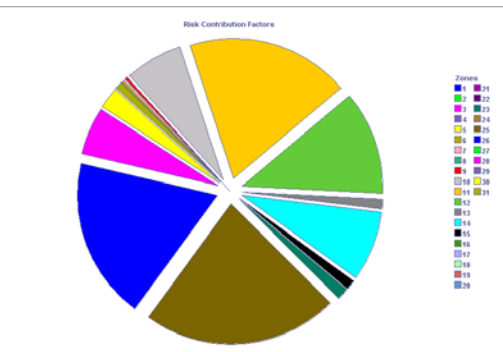
Option specific to *Zone Assessment:*
Click on **Volume Effect** include the influence of zone volume on probability of fracture results for each zone.

Option specific to Contribution Factors:
Click on each individual section of the pie chart to view the contribution percentage associated with an individual zone.
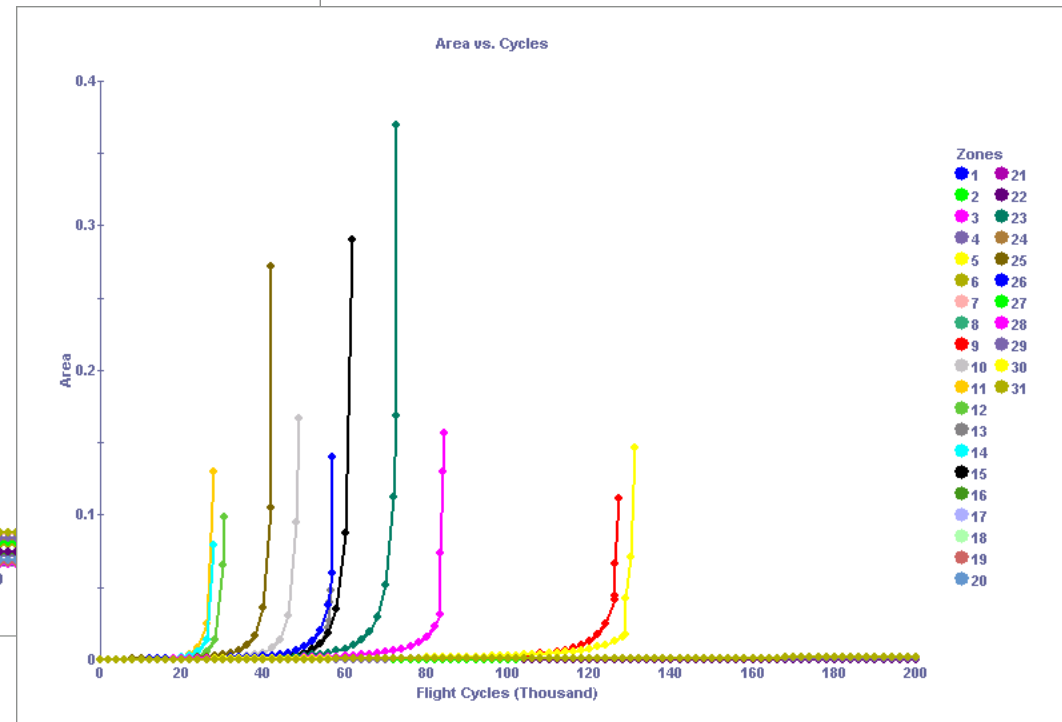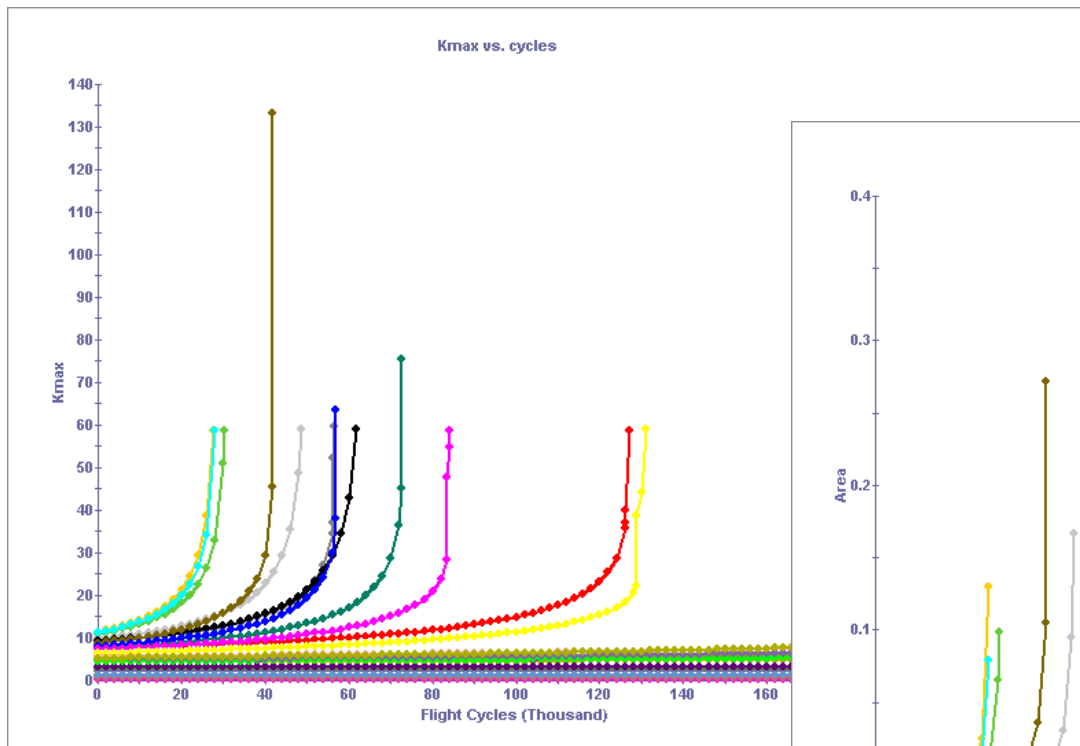
Fracture Mechanics

The 2 graphs in this section are <u>Kmax vs. Cycles</u> and <u>Area vs. Cycles</u>. Zones can be removed from or added to these graphs using the same features shown for the <u>Zone Assessments</u> and <u>Contribution Factors</u> graphs in the Risk Assessment section (see previous page).
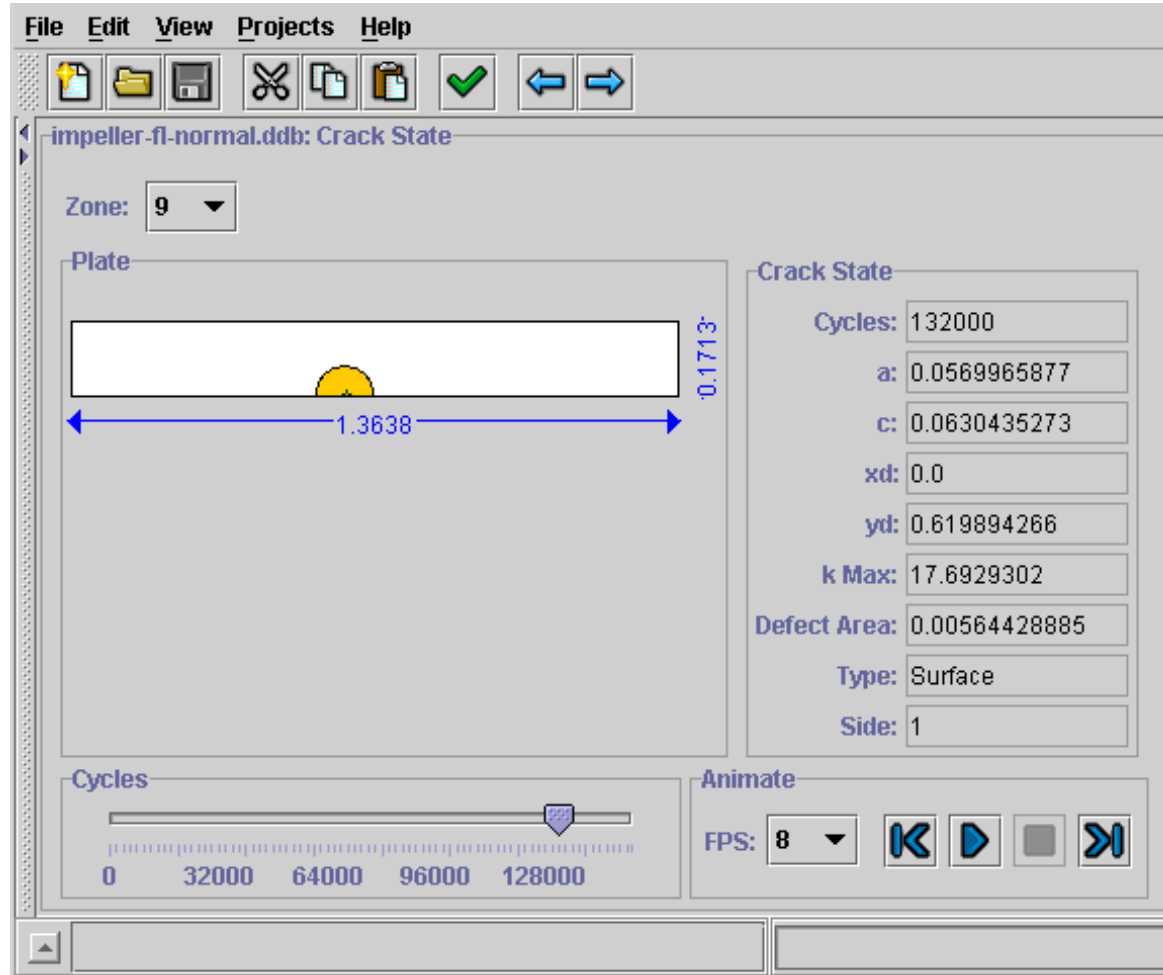
Fracture Mechanics (cont.)

*The **Crack State** section animates crack growth as a function of flight cycles. Darwin™ provides this feature to give users valuable information about crack state transition.*

To View Crack Growth Animation:

1. Select a Zone by clicking on the Zone selection box.

2. Click the ▷ button.

Extra Features:

• Click on **FPS** [8 ▼] box to change the "Frames-Per-Second" (speed) of the animation.

• Click ⬛ to stop the animation.

• Click ◀◀ to go to the beginning.

• Click ▶▶ to go to the end.

impeller-fl-normal.ddb: Crack State

Zone: 9

Plate

1.3638

0.1713

Crack State

Cycles:	132000
a:	0.0569965877
c:	0.0630435273
xd:	0.0
yd:	0.619894266
k Max:	17.6929302
Defect Area:	0.00564428885
Type:	Surface
Side:	1

Cycles

0   32000   64000   96000   128000

Animate

FPS: [8 ▼]

Crack Animation Sample:

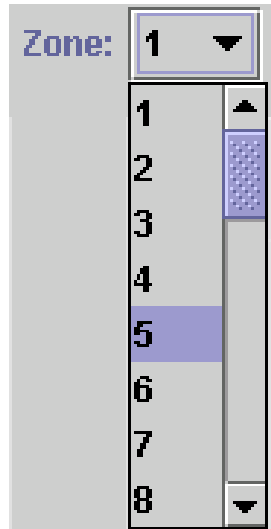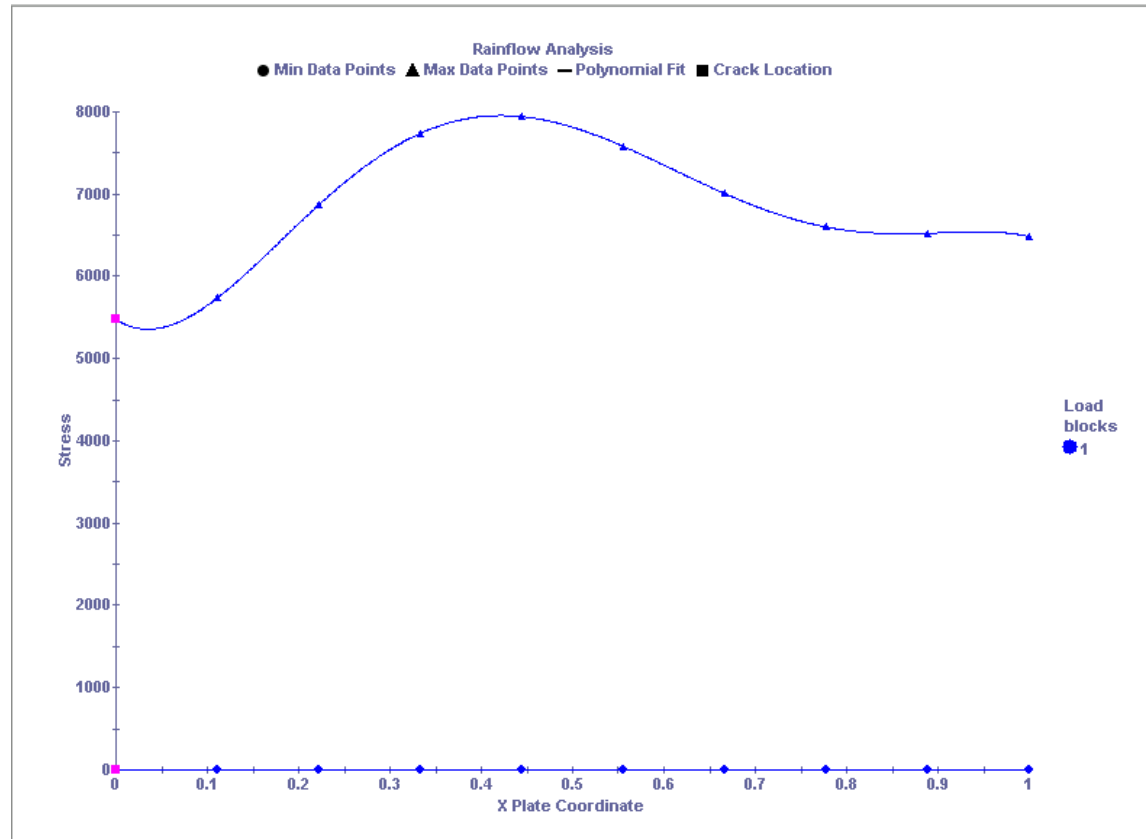## Stress Processing

The stress values (used for fracture mechanics analysis) are shown for each zone in this section. The "Load blocks" legend on the right side of the graph indicates the color of each load block included in the graph.

To change zones:

1. Click on the drop-down menu to select a zone.

Tutorial Summary

A. In Assessment Definition, you have learned how to define the input data required for a disk risk assessment, including the following:

   - Create a project
   - Define general information (units, etc.)
   - Specify fracture model options (including solution method)
   - Add defect distributions
   - Add POD curves
   - Define inspection schedules
   - Add material property data
   - Import finite element data
   - Create zones and specify zone related information
   - Specify the probabilistic computational method
   - Select final analysis options

B. In the Evaluate Results section (post processing) you have learned to view results in 3 different categories, including Risk Assessment, Fracture Mechanics, and Stress Processing.
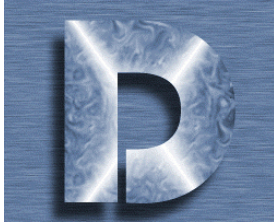
This concludes the

DARWIN™ Interactive Tutorial

We hope you enjoyed this tutorial. For more information on DARWIN™ or the Turbine Rotor Material Design (TRMD) project, please visit our web site.

DARWIN WEB SITE: *www.darwin.swri.org*

This tutorial was created by Southwest Research Institute as part of the ongoing Turbine Rotor Material Design effort funded by the Federal Aviation Administration.