



SINTAP SOFTWARE MANUAL

April 1999

Document: SINTAP/MCS/004

MCS International
Merchants Road
Galway
Ireland

TABLE OF CONTENTS

1	GETTING STARTED.....	1
1.1	INTRODUCTION	1
1.2	INSTALLATION AND RUNNING	1
1.2.1	Hardware & Software Requirements.....	1
1.2.2	Standard Installation.....	2
1.2.3	Server Installation.....	2
1.2.4	Running SINTAP	2
1.3	MANUAL ORGANISATION.....	3
2	GUI BASIC CONCEPTS.....	4
2.1	LIST-TYPE DIALOG	4
2.2	SPREADSHEET-TYPE DIALOG	5
2.3	CLIPBOARD COPY	7
3	ARCHITECTURE.....	8
3.1	INTRODUCTION	8
3.2	GRAPHICAL USER INTERFACE	8
3.2.1	Data-entry.....	8
3.2.2	Analyses.....	9
3.2.3	Results Presentation.....	9
3.2.4	Technical Details	9
3.3	SINTAP ENGINE.....	9
3.3.1	Services.....	9
3.3.2	Source Code.....	10
3.3.3	Technical Details	10
3.4	USER GEOMETRY EXTENSION	10
3.4.1	Services.....	10
4	GEOMETRY SECTION.....	11
4.1	INTRODUCTION	11
4.2	GEOMETRY SELECTION	13
4.3	SPECIFYING DIMENSIONS	15
4.4	SPECIFYING CONSTRAINT PARAMETERS	16
5	ASSESSMENT SECTION	17
5.1	INTRODUCTION	17
5.2	LEVELS	17
5.2.1	Level 0	17
5.2.2	Level 1	18
5.2.3	Level 2	18
5.2.4	Level 3	19
5.2.5	Level 4	19
5.3	YIELD PLATEAU OPTIONS	20
5.4	PLANE STRESS/PLANE STRAIN.....	20
6	MATERIALS PROPERTIES SECTION.....	21
6.1	INTRODUCTION	21
6.2	ELASTIC DEFORMATION.....	21
6.3	PLASTIC DEFORMATION.....	22
6.4	MATERIAL TOUGHNESS INPUTS.....	23
6.4.1	Initiation Toughness, K_{mat}	24
6.4.2	CTOD	24
6.4.3	Charpy Values.....	24
6.4.4	J_{mat}	25
7	J-INTEGRAL	26

7.1	INTRODUCTION	26
8	APPLIED LOADING.....	28
8.1	INTRODUCTION	28
8.2	STRESS PROFILES.....	28
8.3	SPECIFIC LOADING PARAMETERS	30
9	SECONDARY STRESSES.....	31
9.1	INTRODUCTION	31
10	ASSESSMENT	33
10.1	INTRODUCTION	33
10.2	DATA VALIDATION	33
10.3	CALCULATION OF J_E	33
10.4	TREATMENT OF STRESS INPUTS	33
10.5	CALCULATION OF K_R , L_R	34
10.6	SENSITIVITY ANALYSIS.....	35
11	OUTPUT.....	36
11.1	INTRODUCTION	36
11.2	FAILURE ASSESSMENT DIAGRAM	36
11.3	CRACK DRIVING FORCE	37
11.4	GRAPH OPTIONS	38
11.5	TEXT OUTPUT	39
11.6	ERROR MESSAGES	41
12	USER GEOMETRY EXTENSION	42
12.1	INTRODUCTION	42
12.2	CONCEPT.....	42
12.3	REQUIREMENTS	42
12.4	FILES	43
12.5	CREATING A USER GEOMETRY SOLUTION.....	43
12.5.1	<i>Preliminary.....</i>	43
12.5.2	<i>Creating Your Custom Geometry Functions.....</i>	44
12.6	INSTALLING THE COMPLETED USER GEOMETRY EXTENSION	47

1 Getting Started

Welcome to the manual for **SINTAP**, a Windows-based fracture risk assessment software package. This chapter, *Getting Started*, provides information about setting up and running **SINTAP**. Specifically, *Getting Started* is divided into the following sections:

- "Introduction" gives a brief introduction to **SINTAP**.
- "Installation and Running" tells you where to get information on installing and running **SINTAP**.
- "Manual Organisation" gives an overview of the chapters comprising this manual.

1.1 Introduction

SINTAP, an acronym for Structural Integrity Assessment Procedures For European Industry, was a Brite-Euram project undertaken with the objective of providing a unified structural integrity evaluation method for European industry.

Structural integrity is the demonstration of the fitness-for-purpose of engineering components and structures throughout their operational lifetime. It involves a wide range of disciplines and is applicable to a range of industries such as power generation, transport, offshore construction and processing. Used correctly, such an approach can provide a balance between economy and concern for safety and can prevent unnecessary repairs during the structure's lifetime.

There have been significant improvements in recent years in the accuracy and scope of Structural Integrity assessment methods. However, problems still exist and the absence of a single unified method in Europe which can be used in industry is a barrier to a wider acceptance of such approaches. The Brite-Euram project **SINTAP** therefore has addressed this problem.

The project has been completed by a consortium of members from wide ranging backgrounds, each with their own specific interest in the area of fracture risk assessment. The consortium comprises a material supplier (British Steel), an electricity generator (Nuclear Electric Ltd. with AEA), an oil and gas supplier (Shell), a chemical processor (EXXON), safety assessors (Health and Safety Executive and SAQ Inspection Ltd.), Research Institutes (GKSS, Fraunhofer IWM, Institut de Soudure, TWI, VTT, JRC (IAM)), two Universities (Cantabria and Gent), a software developer (Marine Computation Services), and a consultancy (Integrity Management Services).

The **SINTAP** software package produced as a result of the project is a powerful analytical tool capable of complex fracture risk analysis, with a wide range of applications in industry.

1.2 Installation and Running

1.2.1 Hardware & Software Requirements

The SINTAP software has the following hardware and software requirements

- PC with 100 MHz Pentium CPU or better.
- A 32-bit version of the Microsoft Windows operating system. Windows 95, Windows NT 4.0 and later versions of these operating systems are supported.

- A minimum of 16 MB of memory under Windows 9x and a minimum of 32 MB under Windows NT.
- Hard disk with at least 12 MB free space.
- CD-ROM drive (for installation only).
- Any Windows compatible SVGA graphics card.
- Minimum screen resolution: 800 x 600 pixels.
- SVGA monitor.
- Any Windows compatible pointing device.

1.2.2 Standard Installation

The **SINTAP** software is supplied on CD-ROM. The installation CD-ROM is configured for AutoPlay under Windows 9x and Windows NT 4.0. Under most circumstances, the installation program should launch automatically on insertion of the CD-ROM into the drive. If the setup program does not start automatically, please run **MASTER.EXE** from the root of the CD-ROM.

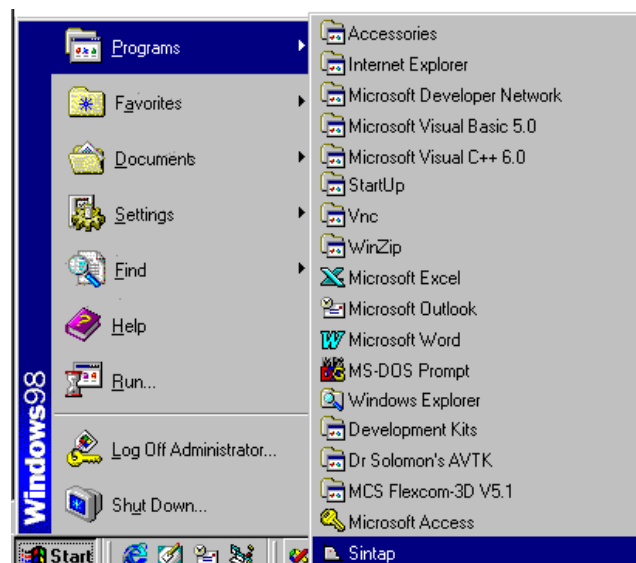
1.2.3 Server Installation

The **SINTAP** software may be installed on a server. The CD-ROM contains a **\SERVER** subdirectory. The contents of this directory may be copied onto a network file server and shared across a network without the need for a separate user installation step.

If this option is chosen, links to the **SINTAP** executable must be created manually in the Start Menu.

1.2.4 Running SINTAP

Run the **SINTAP** software by simply selecting it in the Windows Start Menu.



1.3 Manual Organisation

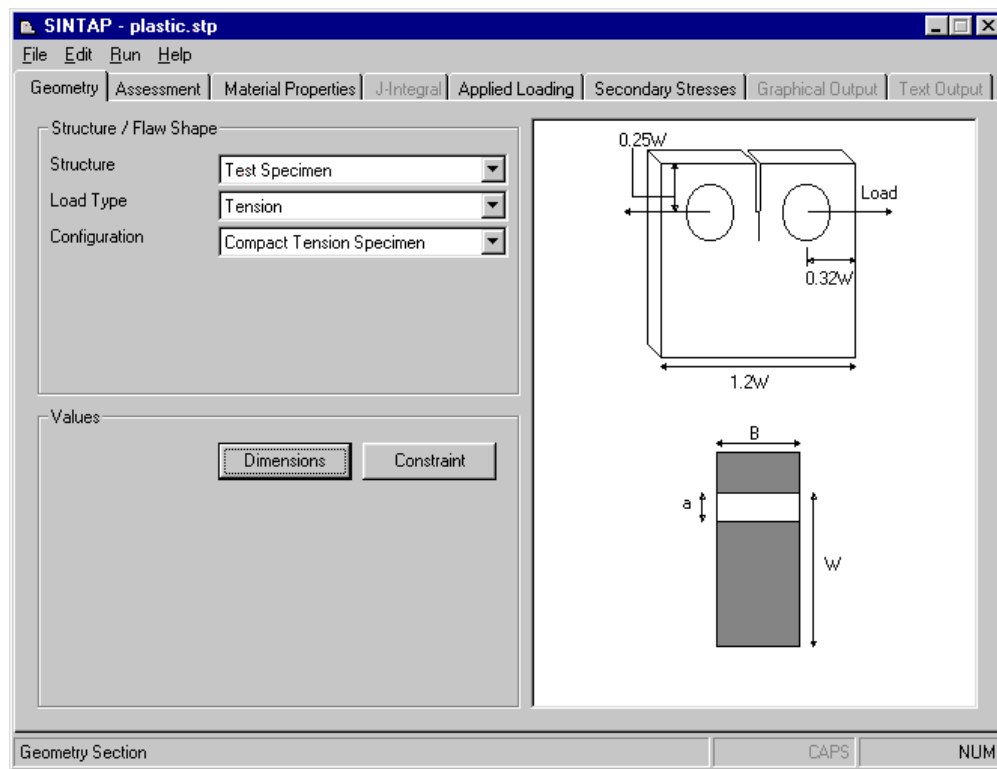
The following is a list of the chapters comprising this manual, together with a brief description of the contents of each.

- GUI basic concepts. This chapter gives you a first glimpse of the **SINTAP** GUI, including all the basic information needed to begin working with the program.
- Architecture. This chapter gives an overview of the layout of the software, together with a brief description of the **SINTAP** GUI and the **SINTAP** engine.
- Geometry. This chapter gives a more detailed description of the Geometry section of the GUI, describing the various geometries available, the required user inputs, and the graphical display during geometry selection.
- Assessment. This chapter gives a more detailed description of the Assessment section of the GUI, describing the different assessment levels available to the user, along with guidance regarding their use.
- Material Properties. This chapter gives a more detailed description of the Material Properties section of the GUI, describing the different tensile and toughness inputs necessary for each assessment level.
- Applied Loading. This chapter gives a more detailed description of the Applied Loading section of the GUI, describing the various applied loading configurations available to the user. Depending on the geometry selected, the user may be allowed to specify a through-section stress profile. Alternatively, a set of loading parameters are requested.
- Secondary Stresses. This chapter gives a more detailed description of the Secondary Stresses section of the GUI, taking the user through the secondary stress library implemented in the software.
- Assessment. This section describes the actions the software performs to conduct an assessment including input validation, the treatment of stress inputs, both primary and secondary and the calculation of the assessment point or assessment locus.
- Output. This chapter describes the outputs of the fracture risk assessment including the FAD, CDF and the text output produced by the software.
- Programming the User Geometry Extension: This chapter describes the use of the User Geometry Extension to add solutions for additional geometries to **SINTAP**.

2 GUI Basic Concepts

This section provides a brief introduction to the operation of the **SINTAP** Graphical User Interface. In most respects, **SINTAP** behaves as a standard Windows application and requires no special training to begin using it. There are however, a few features which merit discussion.

The picture below shows the **SINTAP** main screen.

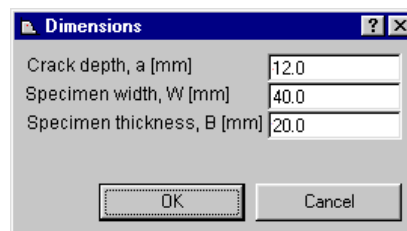


Most of the **SINTAP** screens contain buttons that provide access to further data-entry dialogs for inputting relevant parameters. For example, the *Dimensions* and *Constraint* buttons of the above screen.

The data-entry dialogs are of two main types, the simple list type and the spreadsheet type. These are discussed in more detail in the next two sections.

2.1 List-Type Dialog

These dialogs are for the entry of simple lists of parameters. An example is shown below.



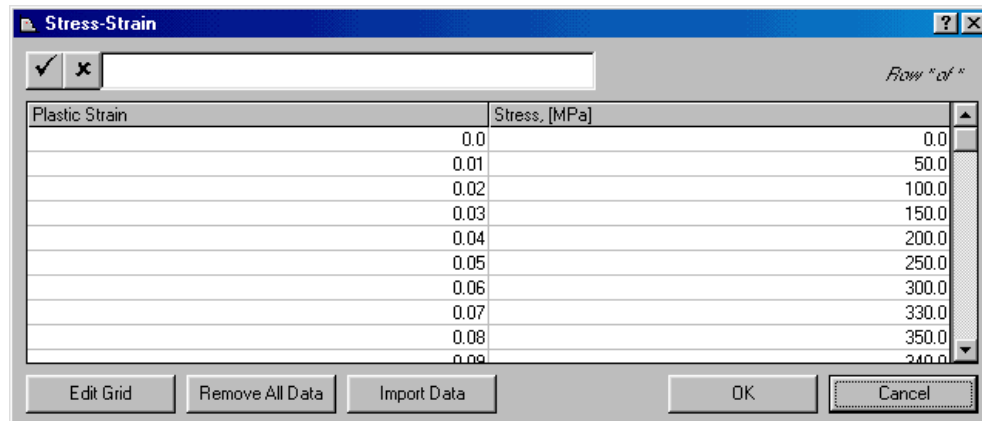
To set the value of a parameter, it is first selected by clicking on its box. Its value may then be changed by typing the new value into the box. The *OK* and *Cancel* buttons may be used to respectively confirm or cancel the entry. Pressing the return key will also confirm the entry.

The buttons at the bottom of the dialog are used as follows:

<i>OK</i>	Confirm the data entered and close the dialog.
<i>Cancel</i>	Close the dialog, discarding any changes made.

2.2 Spreadsheet-Type Dialog

The spreadsheet type dialogs are more complex than the simple list type dialogs. The major difference is that this dialog allows several sets of data to be input. An example is shown below.

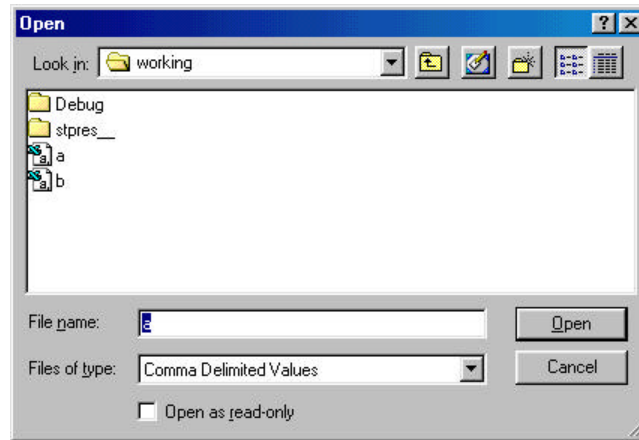


To change the value of a parameter, select the relevant spreadsheet cell by clicking on it and then changing the value at the top of the dialog.

The buttons at the bottom of the dialog also perform the same function as for the simple list-type dialog. There are also some extra buttons as follows:

<i>Remove All Data</i>	Clear the contents of the dialog, restoring all the parameters to their default values.
<i>Import Data</i>	Allows the user to import a previously stored array of values. Described in greater detail below.
<i>Edit Grid</i>	Allows the user to edit the spreadsheet dialog. Described in greater detail below.

The Import Data Button displays the following dialog:

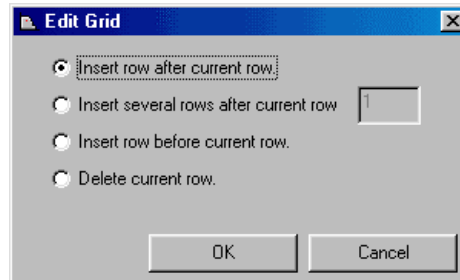


This feature allows the user to import previously stored array of values into the spreadsheet dialog. It is particularly useful when large arrays of data need to be inputted. For example, a detailed stress/strain curve of a material may be stored once in a database and can subsequently be introduced into several **SINTAP** fracture risk assessments.

The import files accepted are .CSV files, as created by Excel. These files contain rows of comma separated values. Periods (.) rather than commas must be used as decimal separators in these files to avoid them been mistaken for delimiters. For example:

0.0,	0.5,	1.2
0.0,	12.4,	3.4
1.0,	21.5,	4.5
13.2,	2.5,	2.3
4.0,	2.5,	3.9

The Edit Spreadsheet Button displays the following dialog:



This allows the user to add/remove rows from the spreadsheet as required. The following keyboard shortcuts are also provided:

<i>Insert</i>	Insert a row after the current row.
<i>Shift+Insert</i>	Insert a row before the current row.
<i>Delete</i>	Clear contents of current cell.

2.3 Clipboard Copy

Many of the screens in **SINTAP** support coping to the Windows Clipboard. This can be very convenient when compiling reports.



Details of what can be copied from each screen are as follows:

Section	Item Copied
Material Properties	Stress strain graph may be copied to the clipboard as a picture.
J-Integral	J-Integral curve may be copied to the clipboard as a picture.
Applied Loading	Through section stress distribution may be copied to the clipboard as a picture.
Secondary Stress	Secondary stress graph may be copied to the clipboard as a picture.
Graphical Output	FAD and CDF may both be copied to the clipboard as a picture.
Text Output	Report may be copied to the clipboard as formatted text.

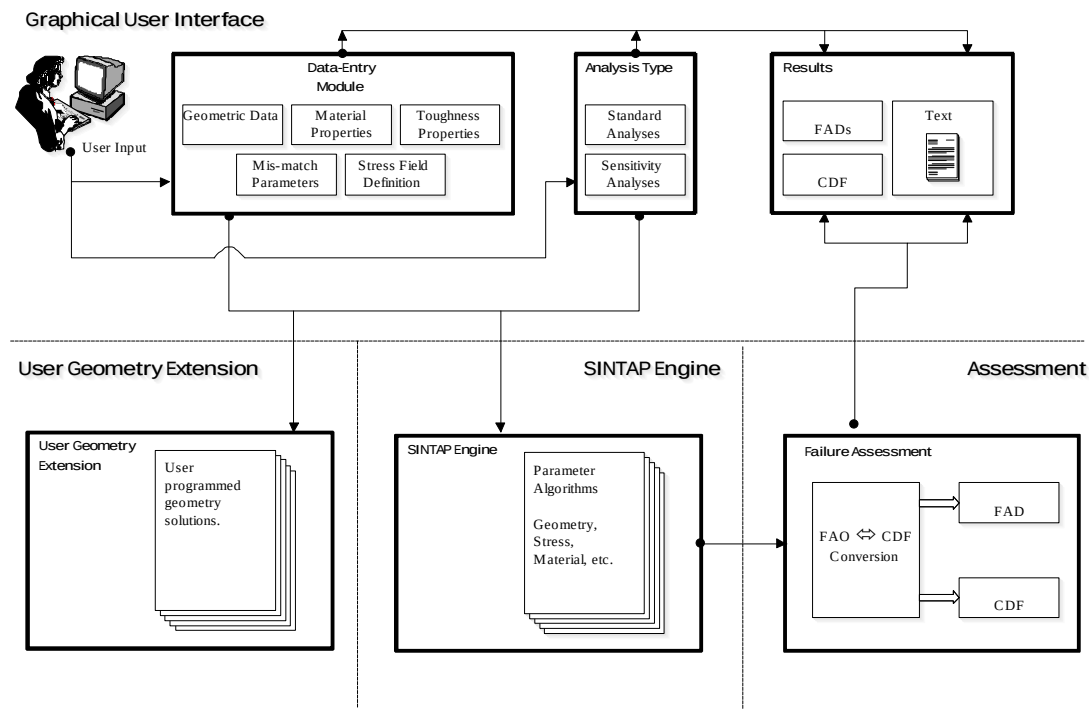
3 Architecture

3.1 Introduction

The **SINTAP** software is composed of three distinct modules:

- Graphical User Interface
- **SINTAP** Engine
- User Defined Geometry Extension

A graphical representation of the architecture is shown below.



3.2 Graphical User Interface

The graphical user interface is responsible for the areas of data-entry, running assessments and presenting results.

3.2.1 Data-entry

The graphical user interface allows the user to define an assessment by entering data in the following areas:

- Geometry Data: *Dimensions, angles, etc.*
- Assessment Level & Options: *Level, mis-match effects, Plane Stress/Plane Strain assumption, etc.*
- Material Tensile Properties: *Yield Strength, Ultimate Strength, Stress/Strain Curves., etc.*
- Material Toughness Properties: *Kmat, CTOD, Charpy and Jmat inputs.*

- J-Integral Results *J vs. Lr, COD vs. Lr.*
- Definition of Stress Field: *Definition of primary and secondary stresses.*

3.2.2 Analyses

The software supports two types of analyses:

- Standard Analyses *Calculation of a single FAD (Lr, Kr) point and an assessment line on a CDF diagram.*
- Sensitivity Analyses *Calculation of a locus of FAD points.*

3.2.3 Results Presentation

The software can present assessment results in three forms.

- Failure Assessment Diagrams [FADs]
- CDF Diagrams.
- Text Output.

3.2.4 Technical Details

Main GUI

- 32-Bit Windows Program.
- Developed in Microsoft Visual Basic.
- Linked to the Graphing Library, **SINTAP** Engine and the user geometry extension.

Graphing Library

- 32-Bit Windows DLL [Dynamic Link Library].
- Developed in Microsoft Visual C++.

3.3 SINTAP Engine

This module encodes the mathematical methods used in performing a structural integrity assessment according to the **SINTAP**-specified procedures.

3.3.1 Services

Parameters entered in the graphical user interface are passed to this level so that calculations may be performed. The engine is primarily concerned with the calculation of:

- Stress Intensity Factors
- Limit Load Solutions
- Input Data Validation
- Polynomial Curve Fits
- The ρ factor

3.3.2 Source Code

The source files contain the SIF and Limit Load solutions and the engine data validation routines for different geometries. The source code for the calculation of p is also included. See the Geometry Section portion of this document for further details.

3.3.3 Technical Details

- 32-Bit Windows DLL [Dynamic Link Library].
- Developed in C.
- Easily portable to other platforms [Unix, DOS, VMS, etc.].
- Directly accessible from other Windows programs developed with tools such as:
 - C/C++
 - Borland Delphi
 - Visual Basic
 - Applications like Excel via VBA [Macro language]

3.4 User Geometry Extension

This module contains user programmed solutions for an arbitrary number of geometries. It's intention is to allow end users to extend the **SINTAP** software themselves.

3.4.1 Services

This module defines geometries and requests parameters from the main GUI. These parameters are then entered in the graphical user interface and passed to this level so that calculations may be performed. The user geometry extension is concerned with the calculation of:

- Stress Intensity Factors
- Limit Load Solutions
- Mis-match Limit Load Ratios

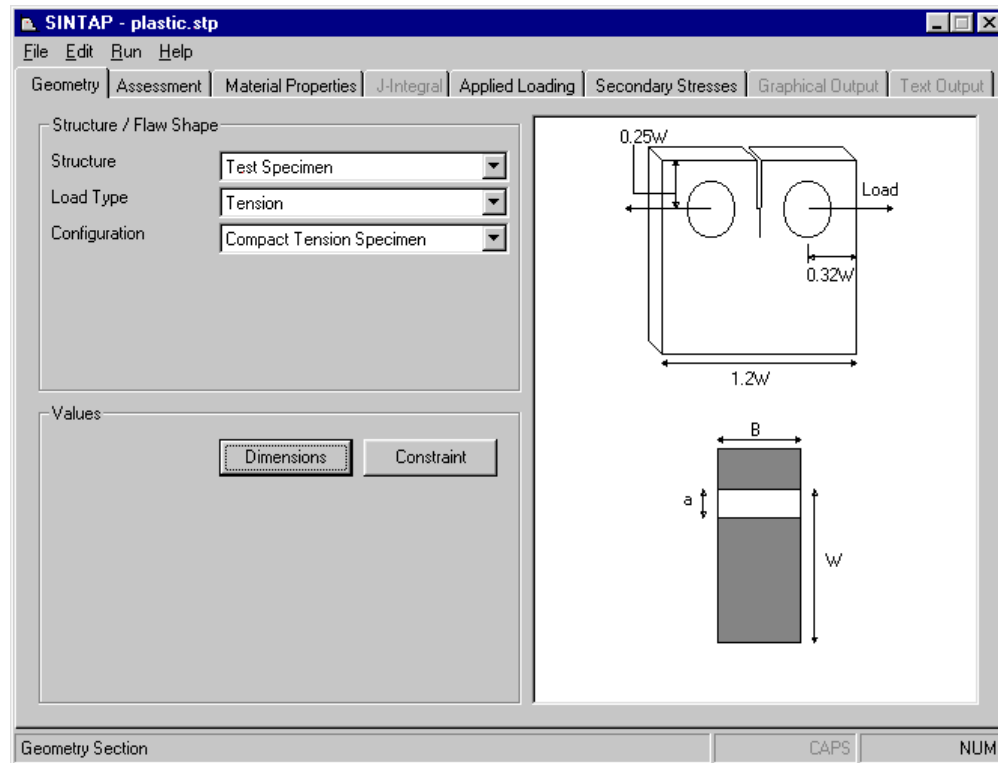
and the performing of data validation on the inputted parameters.

Further details on this module may be found in Chapter 12, *User Geometry Extension*.

4 Geometry Section

4.1 Introduction

The Geometry Section of the **SINTAP** GUI is accessed at any time by clicking on the Geometry tab. By default, it is displayed when the **SINTAP** program is run as geometry selection is deemed to be a suitable starting point for all fracture risk analysis. A typical completed Geometry section of the **SINTAP** GUI is shown below.



The complete list of structures shown on the **SINTAP** geometry screen are as follows:

- Plate Geometries
 - Semi-Elliptical Surface Crack in an Infinite Flat Plate
 - Single Infinite Surface Crack in an Infinite Flat Plate
 - Double Infinite Surface Crack in an Infinite Flat Plate
 - Semi-Circular Surface Crack in an Infinite Flat Plate
 - Embedded Elliptical Crack in an Infinite Flat Plate
 - Embedded Circular Crack in an Infinite Flat Plate
 - Through-Thickness Crack in an Infinite Flat Plate
 - Through-Thickness Crack in a Finite Flat Plate
- Cylinder Geometries
 - Semi-Elliptical Axial Internal Surface Crack in a Cylinder
 - Infinite Axial Internal Surface Crack In a Cylinder
 - Semi-Elliptical Axial External Surface Crack in a Cylinder

- Infinite Axial External Surface Crack In a Cylinder
 - Axial Through-thickness Crack In a Cylinder
 - Semi-Elliptical Circumferential Internal Surface Crack in a Cylinder
 - Infinite Axial Circumferential Surface Crack In a Cylinder
 - Semi-Elliptical Circumferential External Surface Crack in a Cylinder
 - Infinite Circumferential External Surface Crack In a Cylinder
 - Circumferential Through-thickness Crack In a Cylinder
- Sphere Geometries
 - Through-thickness Crack in a Sphere
- Test Specimen Geometries
 - Single Notched Tension Test Specimen
 - Double Notched Tension Test Specimen
 - Compact Tension Test Specimen
 - Pure Bending Test Specimen
 - 3-Point Bending Test Specimen
- Joint Geometries
 - T-Joint Under Axial Loading
 - T-Joint Under In-Plane Bending Moment
 - T-Joint Under Out-of-Plane Bending Moment
 - K-Joint Under Axial Loading
 - K-Joint Under In-Plane Bending Moment
 - K-Joint Under Out-of-Plane Bending Moment
 - X-Joint Under Axial Loading
 - X-Joint Under In-Plane Bending Moment
 - X-Joint Under Out-of-Plane Bending Moment
- User Defined Geometries

Because of the time constraints on the development of the SINTAP software, only a limited number of these geometries are actually fully available for performing analyses. These are:

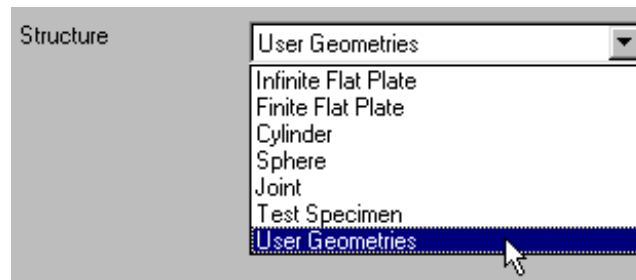
1. Finite Surface Crack in an Infinite Plate
2. Infinite Surface Crack in a Infinite Plate
3. Embedded Elliptical Crack in a Infinite Plate
4. Through-Thickness Crack In a Infinite Plate
5. Through-Thickness Crack In a Finite Homogeneous Plate
6. Through-Thickness Crack In a Finite Homogeneous Plate
7. Through-Thickness Crack In a Finite Welded Plate –Crack in Centre of Weld
8. Through-Thickness Crack In a Finite Welded Plate –Crack in Interface of Weld and Base Metals
9. Through-Thickness Crack In a Finite Welded Plate –Crack in Interface of Bimaterial Joint
10. Finite Axial Internal Surface Crack in a Cylinder
11. Infinite Axial Internal Surface Crack In a Cylinder
12. Finite Axial External Surface Crack in a Cylinder
13. Infinite Axial External Surface Crack In a Cylinder
14. Through-thickness Crack in a Sphere
15. Compact Tension Test Specimen
16. Homogeneous Three Point Bend Specimen
17. Welded Three Point Bend Specimen – Crack in Centre of Weld – $S/W = 8$
18. Welded Three Point Bend Specimen – Crack in Interface of Weld and Base Metals – $S/W = 8$
19. Welded Three Point Bend Specimen – Crack in Interface of Bimaterial Joint – $S/W = 8$

- 20. Welded Three Point Bend Specimen – Crack in Centre of Weld – $S/W = 4$
- 21. Welded Three Point Bend Specimen – Crack in Interface of Weld and Base Metals – $S/W = 4$
- 22. Welded Three Point Bend Specimen – Crack in Interface of Bimaterial Joint – $S/W = 4$

Plus any user geometries added through the user geometry extension.

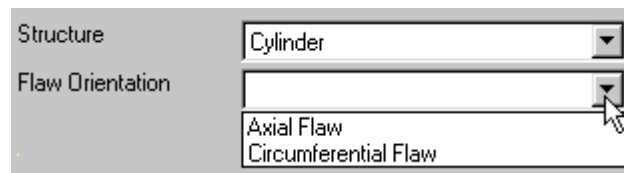
4.2 Geometry selection

This section describes in detail how the user may select a geometry and enter the various parameters relating to that geometry. The structure is selected from the list box of structures by clicking on the down arrow on the right hand side of the list box as shown below.

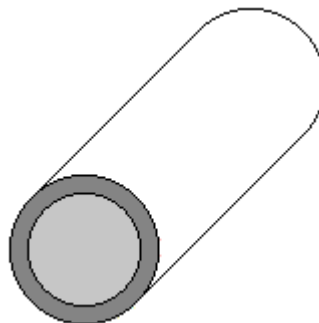


Once the structure has been selected, further list boxes appear below the structure list box so the user may further define the required geometry. The hierarchy of the list boxes is such that the contents of each box depends on the value selected in the previous box.

As an example of a complete geometry specification, let's select "Cylinder" from the list of available structures in the structure list box shown above. The flaw orientation is the next parameter to be specified. By clicking on the down arrow on the right hand side of the list box, the flaw orientation may be selected from the list as shown below.



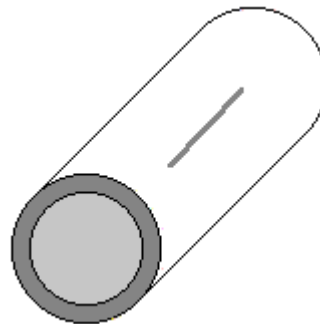
Also, once "Cylinder" has been selected from the list of available structures, the top half of the geometry preview box is updated to display the cylinder geometry as follows:



By selecting “Axial Flaw” from the list of flaw orientations available in the list box shown above, the flaw location is the next parameter to be specified. By clicking on the down arrow on the right hand side of the list box, the flaw location may be selected from the list as shown below.

Structure	Cylinder
Flaw Orientation	Axial Flaw
Flaw Location	Internal External Through Thickness Flaw

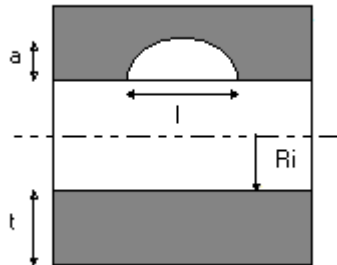
Also, once “Axial Flaw” has been selected from the list of available flaws, the top half of the geometry preview box is updated to display the axial flaw on the cylinder geometry as follows:



By selecting “Internal” from the list of flaw locations available in the list box shown above, the flaw type is the next parameter to be specified. By clicking on the down arrow on the right hand side of the list box, the flaw type may be selected from the list as shown below.

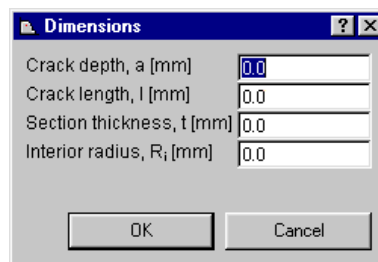
Structure	Cylinder
Flaw Orientation	Axial Flaw
Flaw Location	Internal
Flaw Type	Semi-elliptical Flaw Infinite Flaw

By selecting “Semi-elliptical Flaw” from the list of flaw types available in the list box shown above, the flaw geometry is now fully specified, and the lower half of the geometry preview box is updated to display a detailed view of the completed geometry, showing all the dimensions which must be specified for the fracture risk assessment as follows:



4.3 Specifying Dimensions

Having fully specified the flaw geometry, the specimen and flaw dimensions must now be specified. This is achieved by clicking on the *Dimensions* button which displays the following dialog for the cylinder geometry as specified above.

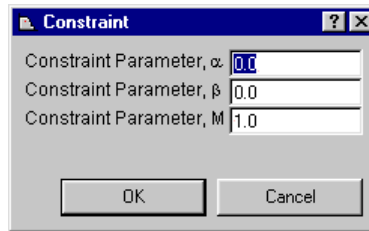


The required dimensions correspond to those shown in the geometry preview box above and these may be inputted as described earlier in the section “GUI basic concepts”. The parameters required to be inputted by the user for the cylinder geometry specified above are:

- Crack depth.
- Crack length.
- Section thickness.
- Interior radius.

4.4 Specifying Constraint Parameters

Constraint parameters may optionally be inputted by clicking on the *Constraint* button which displays the following dialog for the cylinder geometry as specified above. These may be inputted as described earlier in the section “GUI basic concepts”.



The constraint parameters required to be inputted by the user are:

- Constraint Parameter α .
- Constraint Parameter β .
- Constraint Parameter M .

The choice of these parameters is described in R6 Procedure Revision 3, and they have the effect of modifying the failure assessment diagram which is now briefly described.

The FAD is initially constructed as follows:

$$K_r = f(L_r) \quad \text{for } L_r \leq L_r^{\max}$$

$$K_r = 0 \quad \text{for } L_r > L_r^{\max}$$

where $f(L_r)$ depends on the assessment level. The modified FAD, taking account of these constraint parameters is then:

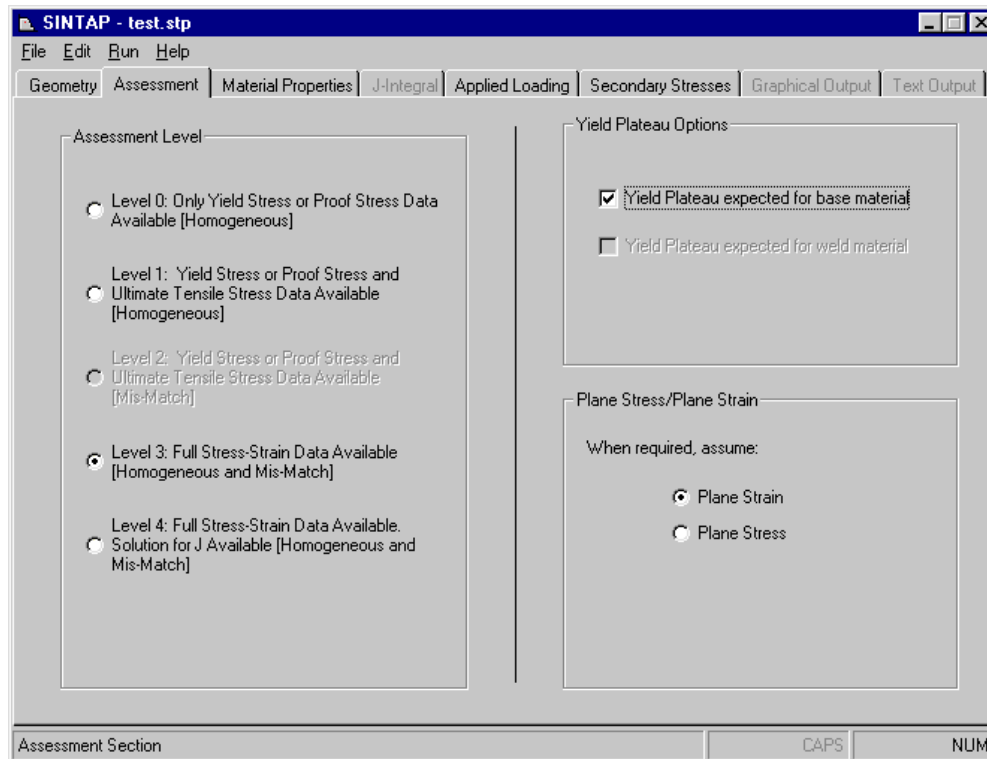
$$K_r = f(L_r) (1 + \alpha (-\beta L_r)^m) \quad L_r \leq L_r^{\max}$$

Once the geometry has been completely specified, including dimensional and constraint parameters, the user may now proceed to the next section of the **SINTAP** GUI, Assessment.

5 Assessment Section

5.1 Introduction

The Assessment Section of the **SINTAP** GUI is accessed at any time by clicking on the Assessment tab. Usually, it is logical to follow from the Geometry section to the Assessment section in a fracture risk analysis. The Assessment section of the **SINTAP** GUI is shown below.



5.2 Levels

The **SINTAP** software has five assessment levels as described below. The level selected by the user will be determined by the complexity of analysis required and also by the detail of information available to the user regarding material properties, loading etc. Usually the user will start off with a low assessment level, and move on to progressively higher levels if the fracture risk assessment does not prove that the flaw is acceptable. The desired assessment level may be selected by clicking on one of the option buttons, Level 0 to Level 4, in the Assessment section.

5.2.1 Level 0

Level 0, also known as the default level, is the simplest level of analysis and only requires input of yield or proof stress data. Mis-match is not supported. This option is disabled if the selected geometry is mis-matched. The user inputs required for a Level 0 assessment are as follows:

- Geometry dimensions.
- Yield or proof strength of the material.
- Young's Modulus of the material.
- Poisson's Ratio of the material.
- Toughness data entered as a Kmat, CTOD, Charpy or Jmat value.

The equations for the Level 0 Failure Assessment Diagrams are taken from the **SINTAP Procedure**, Rev. 5.0, Annex 4.1. The equations for the Level 0 FADs are as follows:

- Level 0 FAD, yield plateau expected: Eqn. 1.
- Level 0 FAD, yield plateau not expected: Eqn. 2.
- L_{rmax} Extension: Eqn. 3.

5.2.2 Level 1

Level 1 is the next level of assessment and requires input of yield or proof stress data. Ultimate strength data is also required. Mis-match is not supported. This option is disabled if the selected geometry is mis-matched. The user may specify whether or not the material exhibits a yield plateau. The user inputs required for a Level 1 assessment are as follows:

- Geometry dimensions.
- Yield or proof strength of the material.
- Ultimate tensile strength of the material.
- Young's Modulus of the material.
- Poisson's Ratio of the material.
- Toughness data entered as a Kmat, CTOD or Jmat value.

The equations for the Level 1 Failure Assessment Diagrams are taken from the **SINTAP Procedure**, Rev. 5.0, Annex 4.2. The equations for the Level 1 FADs are as follows:

- Level 1 FAD, yield plateau expected: Eqn. 4.
- Level 1 FAD, yield plateau not expected: Eqn. 11.

5.2.3 Level 2

Level 2 is a mis-match only level. It is disabled if the selected geometry is not mis-matched. Material properties for the weld material as well as the base material must be specified. The user may specify that either one or both of the base and weld materials exhibit a yield plateau. The user inputs required for a Level 2 assessment are as follows:

- Geometry dimensions.
- Yield strength of the base material.
- Ultimate tensile strength of the base material.
- Young's Modulus of the base material.
- Yield strength for the weld material.
- Ultimate tensile strength of the weld a material.
- Young's Modulus for the weld material
- Poisson's Ratio of the base material.
- Toughness data entered as a Kmat, CTOD or Jmat value.

The equations for the Level 2 Failure Assessment Diagrams are taken from the **SINTAP Procedure**, Rev. 5.0, Annex 4.3. The equations for the Level 2 FADs are as follows:

- Level 2 FAD, yield plateau expected in both materials: Eqns. 15,16 & 20
- Level 2 FAD, yield plateau expected in neither material: Eqns. 25 & 29
- Level 2 FAD, yield plateau expected in one material only: Eqns. 34 & 40

5.2.4 Level 3

Level 3 supports mis-match. If mis-match is specified, then material properties for the weld as well as the base material must be specified. The user inputs required for a Level 3 assessment are as follows:

- Geometry dimensions.
- Yield strength of the base material.
- Ultimate tensile strength of the base material.
- Young's Modulus of the base material.
- Poisson's Ratio of the base material.
- A Stress/Plastic Strain curve for the base material
- Toughness data entered as a Kmat, CTOD or Jmat value.

Some additional inputs required for a Level 3 assessment with mis-match are as follows:

- Yield strength for the weld material.
- Ultimate tensile strength of the weld a material.
- Young's Modulus for the weld material
- Poisson's Ratio of the weld material.
- A Stress/Plastic Strain curve for the weld material

The equation for the case without mis-match is taken from the **SINTAP Procedure**, Rev. 5.0, Annex 4.4.

- Level 3 FAD Eqn. 45.

The equation for the case with mis-match are taken from the **SINTAP Procedure**, Rev. 5.0, Annex 4.4.

- Level 3 FAD: Eqn. 47.

5.2.5 Level 4

Level 4 is the most detailed assessment level and supports mis-match. If mis-match is specified, then material properties for the weld as well as the base material must be specified. The user inputs required for a Level 4 assessment are as follows:

- Geometry dimensions.
- Yield strength of the base material.
- Ultimate tensile strength of the base material.
- Young's Modulus of the base material.
- Poisson's Ratio of the base material.
- A Stress/Plastic Strain curve for the base material
- J-Integral analysis results for the body
- Toughness data entered as a Kmat, CTOD or Jmat value.

Some additional inputs required for a Level 4 assessment with mis-match are as follows:

- Yield strength for the weld material.
- Ultimate tensile strength of the weld a material.
- Young's Modulus for the weld material
- A Stress/Plastic Strain curve for the weld material

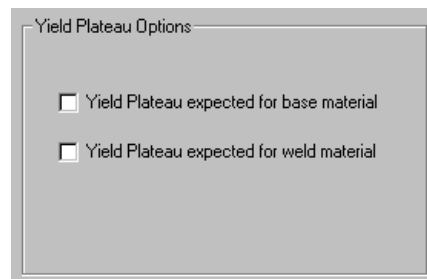
The equation the Level 4 FAD used is:

$$K_r = \left(\frac{J}{J_e} \right)^{-\frac{1}{2}}$$

Which can be found in the **SINTAP Procedure**, Rev. 5.0, Annex A.

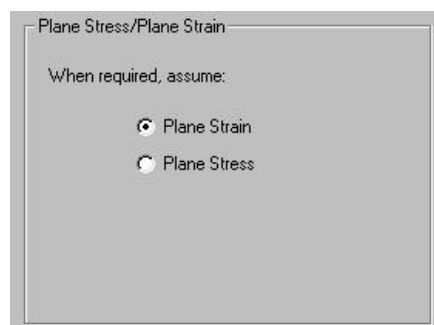
5.3 Yield Plateau Options

The Yield Plateau option boxes allow the user to specify whether or not the material (or the base and weld material in the case of mis-match) exhibits a yield plateau. This option is included on the Assessment screen because of its profound effect on the FAD shape. The Weld material option is only enabled when a mis-matched geometry is chosen. The default is no yield plateau expected.



5.4 Plane Stress/Plane Strain

The software has to make occasional assumptions of plane stress or plane strain. This effects some SIF and Limit Load solutions and also the calculation of J values from K values. The default is to assume plane strain when required.

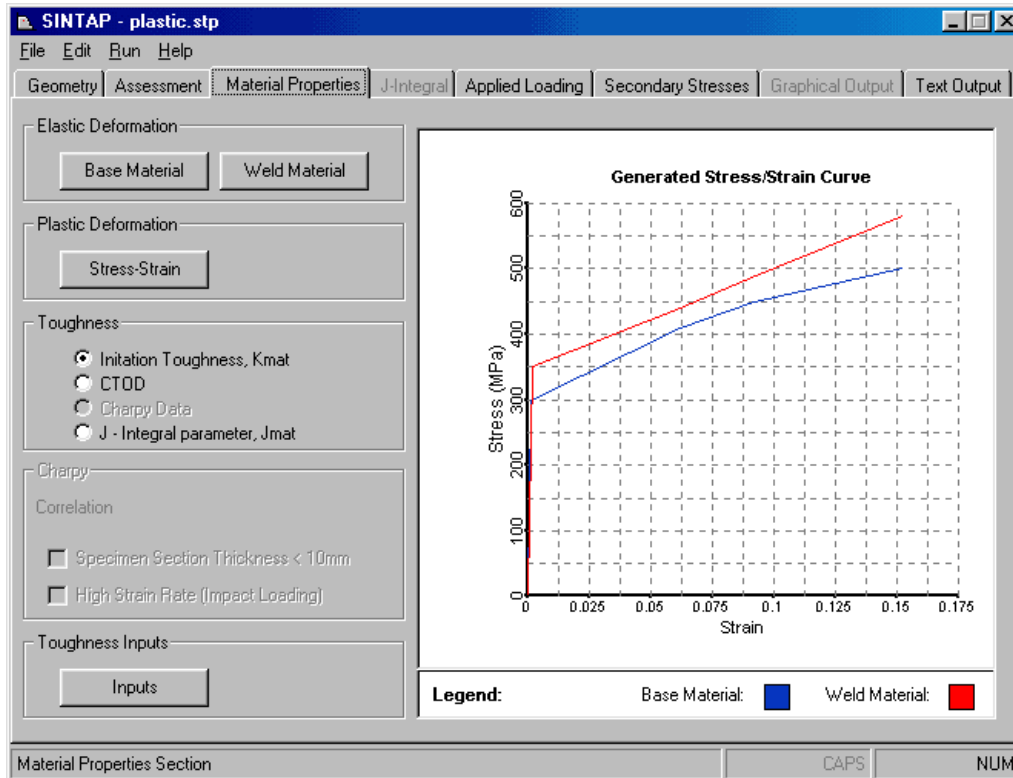


Once the assessment level has been specified, the user may now proceed to the next section of the **SINTAP** GUI, Material Properties.

6 Materials Properties Section

6.1 Introduction

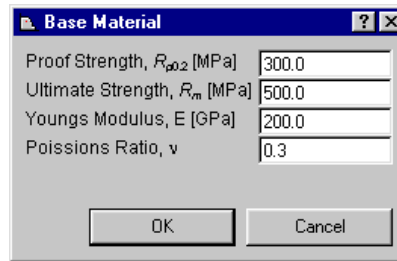
The Materials Properties Section of the **SINTAP** GUI may be accessed at any time by clicking on the Materials Properties tab. Usually, it is logical to follow from the Assessment section to the Materials Properties section in a fracture risk analysis. The Materials Properties section of the **SINTAP** GUI is shown below.



The Material Properties section of the **SINTAP** GUI allows the user to specify tensile material properties for the base and weld materials (with a mis-matched geometry), along with options for specifying toughness properties for the base material. Depending on the geometry selected in the Geometry section previously, the tensile properties to be specified by the user will vary. For example, if the user has selected a geometry which is not mis-matched, then the *Weld Material* button in the *Elastic Deformation* section will appear greyed out as properties for the weld material are not relevant to the assessment.

6.2 Elastic Deformation

The *Elastic Deformation* section allows the user to specify elastic parameters for both the base material and the weld material. By clicking on the *Base Material* button, the following dialog appears:



Base Material

Proof Strength, $R_{p0.2}$ [MPa] 300.0

Ultimate Strength, R_m [MPa] 500.0

Youngs Modulus, E [GPa] 200.0

Poissons Ratio, ν 0.3

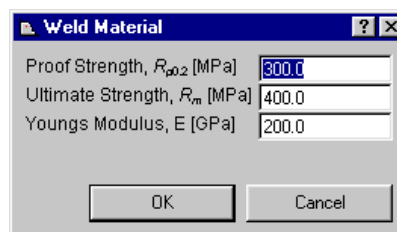
OK Cancel

The following base material parameters are required to be inputted by the user and these may be inputted as described earlier in the section “GUI basic concepts”.

- Proof Strength.
- Ultimate Strength.
- Young’s Modulus.
- Poisson’s Ratio.

The only exception is for Level 0. Ultimate Strength is not required as an input at Level 0 and does not appear on the dialog.

By clicking on the *Weld Material* button, the following dialog appears:



Weld Material

Proof Strength, $R_{p0.2}$ [MPa] 300.0

Ultimate Strength, R_m [MPa] 400.0

Youngs Modulus, E [GPa] 200.0

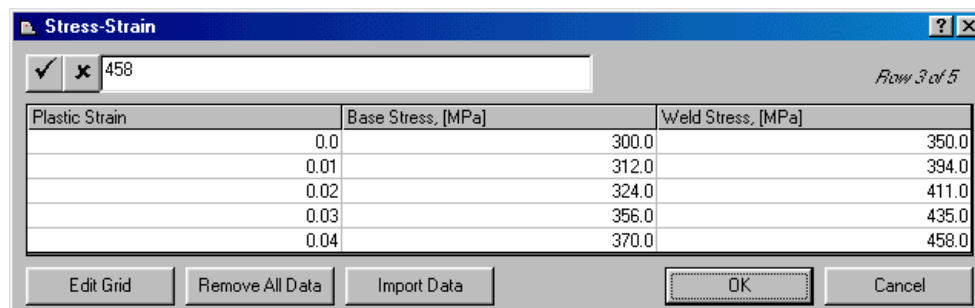
OK Cancel

The following weld material parameters are required to be inputted by the user and these may be inputted as described earlier in the section “GUI basic concepts”.

- Proof Strength.
- Ultimate Strength.
- Young’s Modulus.

6.3 Plastic Deformation

The *Plastic Deformation* section allows the user to specify plastic parameters for the base and optionally the weld materials. For a mismatched geometry, clicking on the *Stress Strain* button produces the following dialog:



Stress-Strain

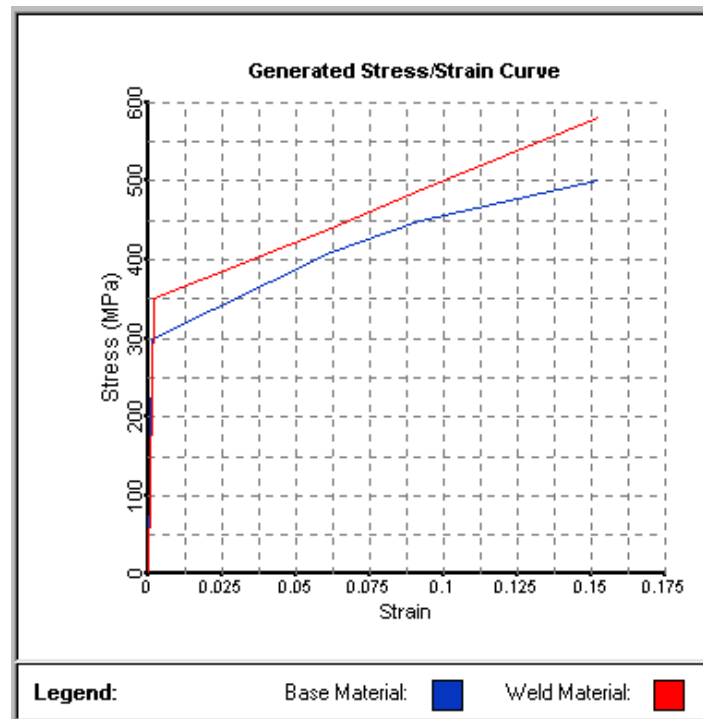
✓ x 458 Row 3 of 5

Plastic Strain	Base Stress, [MPa]	Weld Stress, [MPa]
0.0	300.0	350.0
0.01	312.0	394.0
0.02	324.0	411.0
0.03	356.0	435.0
0.04	370.0	458.0

Edit Grid Remove All Data Import Data OK Cancel

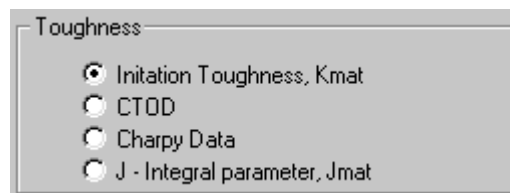
For the case of a homogeneous geometry, the dialog is the same except that the Weld Stress column is hidden.

Pairs of Plastic Strain and Stress values are required to be inputted by the user to define the stress/strain curve for the base material. These may be inputted as described earlier in the section “GUI basic concepts”. If the assessment level is Level 3 or greater, and a stress/plastic strain curve has been inputted, a generated stress/strain curve for the base and/or the weld material will be graphed on the right hand side of this screen as shown below. The generated stress strain curve calculates the total strain as the elastic strain, calculated using the stress and Young’s Modulus, and the values of plastic strain entered in the table above.



6.4 Material Toughness Inputs

There are four options when inputting toughness values. The procedure uses K_{mat} values internally for all K calculations. If a toughness value is input in one of the other three formats then a K_{mat} value is automatically calculated from the parameters input. The method for inputting toughness may be selected by clicking on one of the option buttons as shown below. By default, the option for inputting initiation toughness K_{mat} directly is automatically selected.



6.4.1 Initiation Toughness, K_{mat}

K_{mat} is the intrinsic toughness measure in the **SINTAP** software. All other toughness values are converted to K_{mat} values before being used.

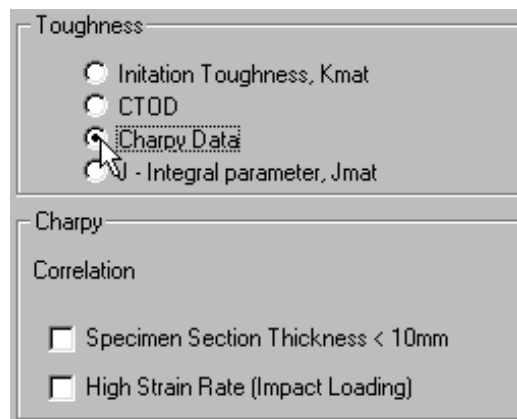
6.4.2 CTOD

CTOD values are converted to K_{mat} values using Eqns. 9 & 10 from the *Sub-Task 3.3 Report* [SINTAP/BS/17]. If the assessment is Level 0 then only Eqn. 9 is used since there is no Ultimate Tensile Strength value available.

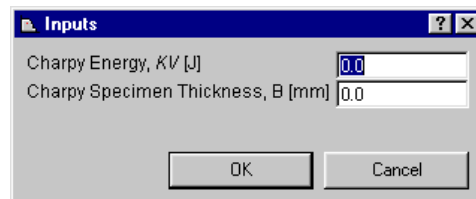
6.4.3 Charpy Values

Charpy values are converted to K_{mat} values using Eqns. 1, 2, 8, 12, 17, 18, 19, 20 & 21 from the *Sub-Task 3.3 Report* [SINTAP/BS/17]. Charpy values are only allowed at Level 0 because they are not considered of sufficient quality for higher level assessment.

For example, if the Charpy Data option is selected then Charpy correlation options are displayed, which appear greyed out if one of the other options is selected, as shown below.



By clicking on the *Toughness* button in the *Toughness Inputs* section, the following dialog appears:



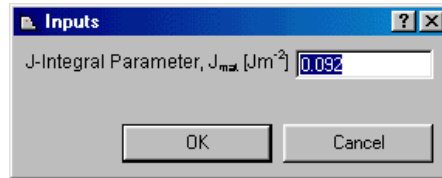
The following Charpy parameters are required to be inputted by the user and these may be inputted as described earlier in the section “GUI basic concepts”.

- Charpy Energy.
- Charpy Specimen Thickness.

6.4.4 J_{mat}

J_{mat} are converted to K_{mat} values using Eqns. 9 & 10 from the *Sub-Task 3.3 Report* [SINTAP/BS/17].

By clicking on the *Toughness* button in the *Toughness Inputs* section, the following dialog appears:



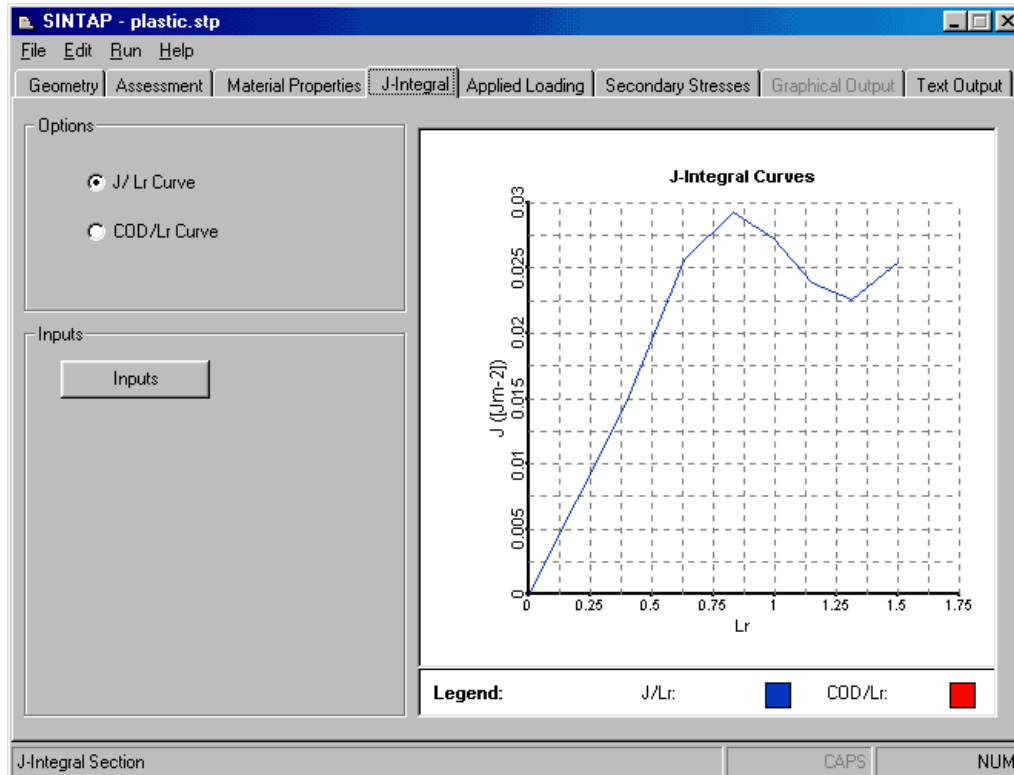
Simply enter the value of J_{mat} in units of Jm^{-2} .

If a Level 4 assessment has been selected, the user may now proceed to an the J-Integral section of the **SINTAP** GUI. This section is not relevant to any other assessment level and the J-Integral tab will appear greyed out should a lower level of assessment be selected. If this is the case, proceed instead to the following section of the **SINTAP** GUI, Applied Loading.

7 J-Integral

7.1 Introduction

The J-Integral Section of the **SINTAP** GUI is accessed at by clicking on the J-Integral tab. This section is only enabled when the assessment level is Level 4. Usually, in a Level 4 analysis, it is logical to follow from the Material Properties section. The J-Integral section of the **SINTAP** GUI is shown below.



This section allows the user to input the J-Integral curve for the material. There are two options for the form of input as follows:

- J as a function of L_r
- COD as a function of L_r

For example if the J/L_r Curve option is selected, clicking on the *Inputs* button displays the following dialog.

J, [Jm-2] (Y-axis)	Lr, (X-axis)
0.0	0.0
0.0146	0.3977
0.0256	0.6363
0.0292	0.8351
0.0272	0.9942
0.0238	1.1533
0.0226	1.3124
0.0255	1.5112

Pairs of J-Integral and L_r values are required to be inputted by the user to define the J/L_r curve for the material. These may be inputted as described earlier in the section “GUI basic concepts”.

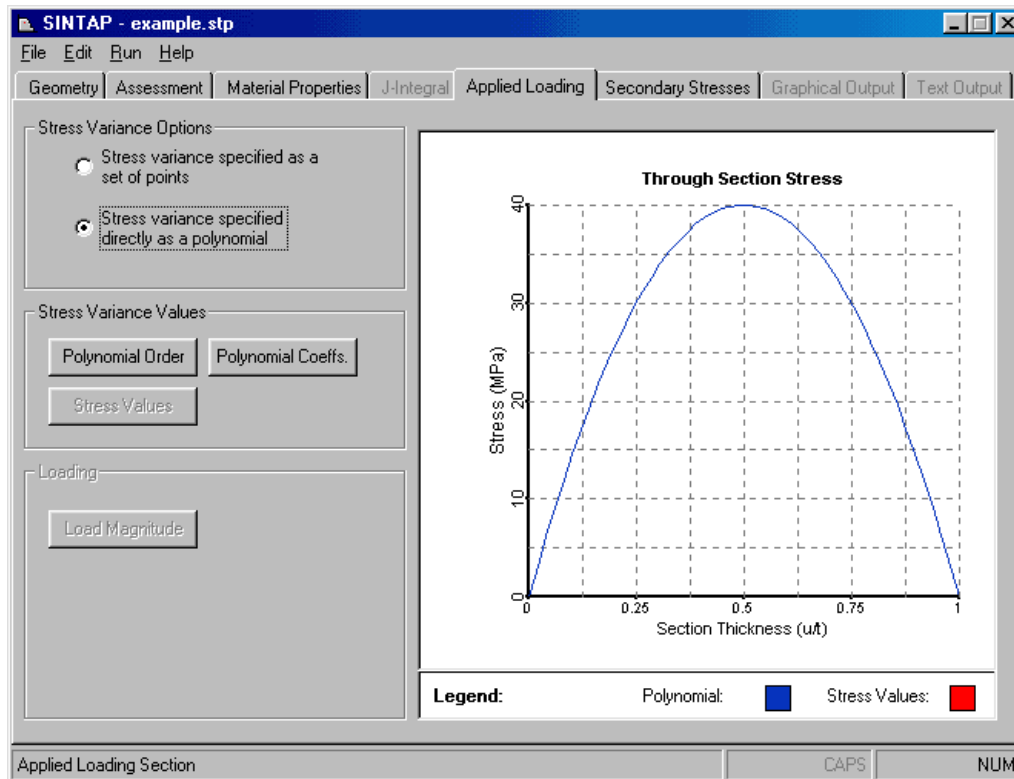
Regardless of how the J-Integral information is specified, CDFs in the graphical output screen are always shown as a function of J.

Once the J-Integral section has been fully completed, the user may now proceed to the next section of the **SINTAP** GUI, Applied Loading.

8 Applied Loading

8.1 Introduction

The Applied Loading Section of the **SINTAP** GUI is accessed at any time by clicking on the Applied Loading tab. The Applied Loading section of the **SINTAP** GUI is shown below.



The Applied Loading section of the **SINTAP** GUI allows the user to specify the loading distribution on the body being assessed. There are different options for specifying the applied loading:

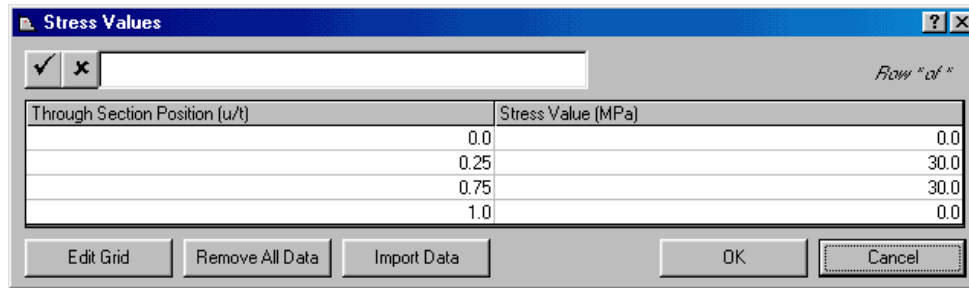
- Stress Profiles
 - Stress variance specified as a set of points.
 - Stress variance specified directly as a polynomial
- Specific Loading Parameters.

Depending on the geometry selected, the user may either be allowed to specify a through-section stress profile or a be requested to enter specific loading parameters.

8.2 Stress Profiles

If a stress profile is called for, the user may input it as either a set of data points or in the form of polynomial coefficients describing the shape of the stress field.

In the case of a set of data points, the user may input the required information by clicking on the *Data Points* button which displays the following dialog.

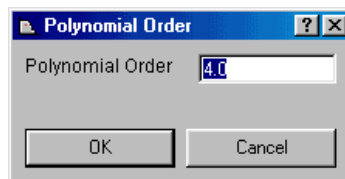


The **Stress Values** dialog box features a table for inputting stress data. It includes a 'Flow * of *' label, a table with two columns ('Through Section Position (u/t)' and 'Stress Value (MPa)'), and buttons for 'Edit Grid', 'Remove All Data', 'Import Data', 'OK', and 'Cancel'.

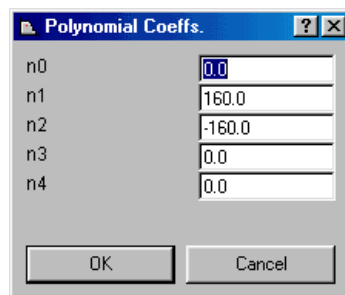
Through Section Position (u/t)	Stress Value (MPa)
0.0	0.0
0.25	30.0
0.75	30.0
1.0	0.0

Pairs of Through Section Position and Stress values are required to be inputted by the user to define the stress profile for the body under assessment. These may be inputted as described earlier in the section “GUI basic concepts”.

In the case of polynomial coefficients, the user may input the required information by clicking on the *Polynomial Order* and *Polynomial Coeffs* buttons which display the following dialogs, respectively.



The **Polynomial Order** dialog box contains a single input field for 'Polynomial Order' set to 4.0, with 'OK' and 'Cancel' buttons.

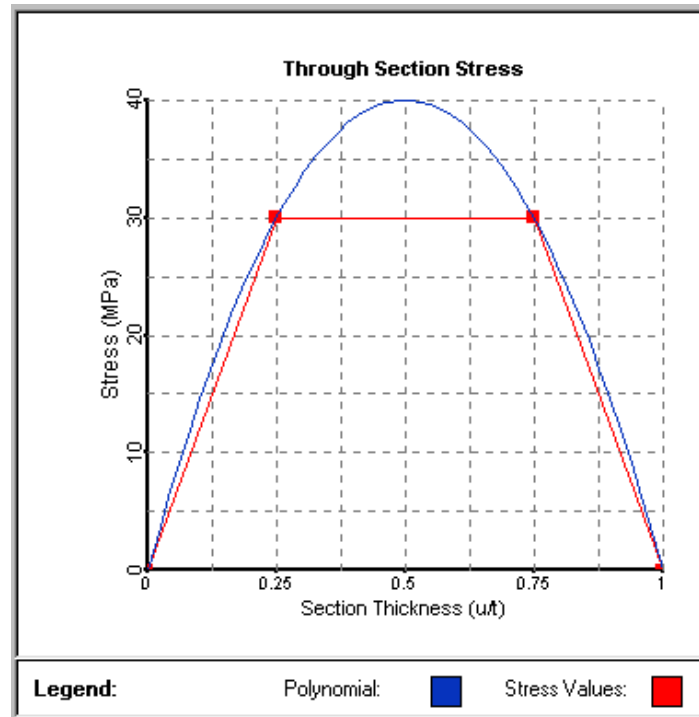


The **Polynomial Coeffs.** dialog box displays five input fields for coefficients n0 through n4. The values are: n0 = 0.0, n1 = 160.0, n2 = -160.0, n3 = 0.0, and n4 = 0.0. It includes 'OK' and 'Cancel' buttons.

The following polynomial parameters are required to be inputted by the user and these may be inputted as described earlier in the section “GUI basic concepts”.

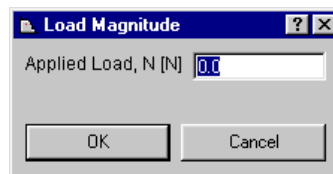
- Polynomial order.
- Polynomial coefficients.

If the user inputs data points, the software will provide a best-fit polynomial to them in the order specified under *Polynomial Order*. The user may adjust the polynomial order upward or downward to provide the most satisfactory fit to their data. The polynomial representation of the stress field is the one used in the software to conduct the assessment so choosing a good fit is important, as shown below.



8.3 Specific Loading Parameters

The SIF and Limit Load Solutions for some geometries do not allow an arbitrary through section stress field to be specified to describe the loading. This section requests the user to supply specific loading parameters such as a single value of applied load, a pressure, a membrane and bending stress, etc. The user may input the required information by clicking on the *Load Magnitude* button in the *Loading* section which displays the following dialog.



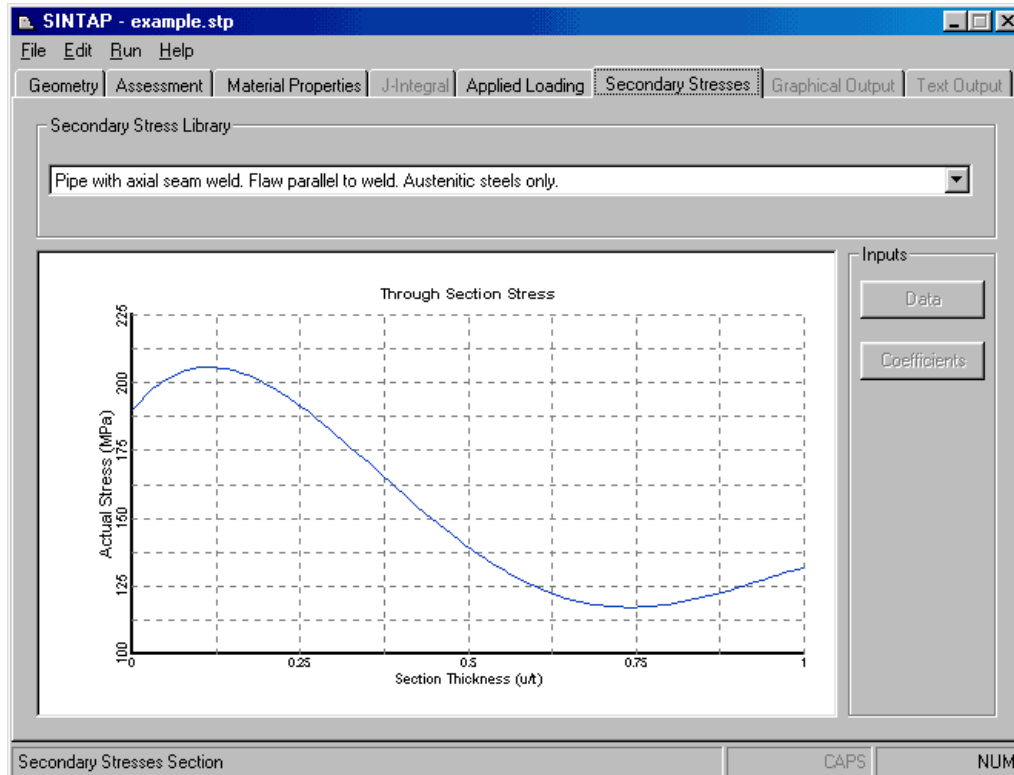
In the case of a user geometry, this dialog will request the loading parameters as specified in the user geometry extension. See Chapter 12, *User Geometry Extension* for more details.

Once the Applied Loading section has been completed, the user may now proceed to the next section of the SINTAP GUI, Secondary Stresses.

9 Secondary Stresses

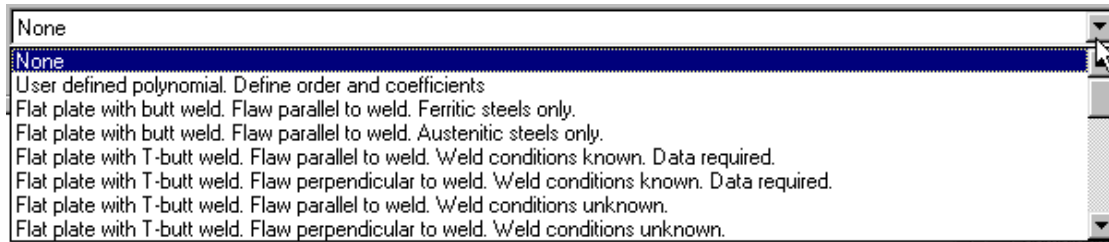
9.1 Introduction

The Secondary Stresses Section of the **SINTAP** GUI may be accessed by clicking on the Secondary Stresses tab. Usually, it is logical to follow from the Applied Loading section to the Secondary Stresses section in a fracture risk analysis. A typical Secondary Stresses section of the **SINTAP** GUI is shown below.



The secondary stresses section shows a library of through section secondary stress profiles. The profiles contained in the secondary stress library come from Version 3 of the *Compendium of Residual Stress Profiles* [BRPR-CT95-0024]. A user-defined profile may also be entered. Several profiles require further input from the user and this is indicated in the name. Since these are through section profiles, they are only available when the geometry in question takes a through section stress profile as an input to its SIF calculations. If this is not the case, the Secondary Stress tab is greyed out and cannot be selected.

The secondary stress profile is selected from the list box of profiles by clicking on the down arrow on the right hand side of the list box as shown below.



A complete list of residual stress profiles in the software is as follows

1. Flat plate with butt weld. Flaw parallel to weld. Ferritic steels only.
2. Flat plate with butt weld. Flaw parallel to weld. Austenitic steels only.
3. Flat plate with butt weld. Flaw perpendicular to weld. All steels.
4. Flat plate with T-butt weld. Flaw parallel to weld. Weld conditions known.
5. Flat plate with T-butt weld. Flaw perpendicular to weld. Weld conditions known.
6. Flat plate with T-butt weld. Flaw parallel to weld. Weld conditions unknown.
7. Flat plate with T-butt weld. Flaw perpendicular to weld. Weld conditions unknown.
8. Pipe with butt weld. Flaw parallel to weld. All steels.
9. Pipe with butt weld. Flaw perpendicular to weld. Ferritic steels only. Low heat weld.
10. Pipe with butt weld. Flaw perpendicular to weld. Ferritic steels only. High heat weld.
11. Pipe with butt weld. Flaw perpendicular to weld. Austenitic steels only.
12. Pipe with T-butt weld. Flaw parallel to weld. Recommended for ferritic steels only.
13. Pipe with T-butt weld. Flaw perpendicular to weld. Recommended for ferritic steels only.
14. Set In Nozzle. Flaw parallel to weld. All steels.
15. Set In Nozzle. Flaw perpendicular to weld. All steels.
16. Pipe with set in nozzle. Flaw parallel to weld. All steels.
17. Pipe with set in nozzle. Flaw perpendicular to weld. All steels.
18. Set On Nozzle. Flaw parallel to weld. All steels.
19. Set On Nozzle. Flaw perpendicular to weld. All steels.
20. Pipe with set on nozzle. Flaw parallel to weld. All steels.
21. Pipe with set on nozzle. Flaw perpendicular to weld. All steels.
22. Pipe with axial seam weld. Flaw parallel to weld. Ferritic steels only.
23. Pipe with axial seam weld. Flaw parallel to weld. Austenitic steels only.
24. Pipe with axial seam weld. Flaw perpendicular to weld. All steels

To illustrate the case of a stress profile which requires data input, clicking on profile no. 4 above causes the normally greyed out *Data* button to become active. By clicking on the *Data* button, the following dialog appears:

Input these parameters as described earlier in the section “GUI basic concepts”.

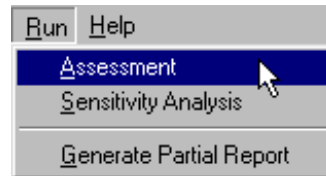
Once the Secondary Stresses section has been completed, the user has now completely specified all the necessary parameters and may now proceed to performing a fracture risk assessment using **SINTAP**.

10 Assessment

10.1 Introduction

This section describes the actions the software performs to conduct an assessment including input validation, the treatment of stress inputs, both primary and secondary and the calculation of the assessment point or assessment locus.

An assessment is run by clicking on the *Assessment* option from the *Run* menu as shown below.



All analyses follow the following steps:

1. Data Validation
2. Calculation of J_e
3. Calculation of K_r , L_r

For sensitivity analyses, step 3 is performed repeatedly.

10.2 Data Validation

The software attempts to validate all the users input pre-run to prevent errors occurring in the engine. The GUI attempts to ensure that all necessary values have been entered while the data validation sections of the engine check that the entered values are within valid ranges.

10.3 Calculation of J_e

The software must calculate J_e as a function of L_r prior to the assessment run so that the CDFs and the Level 4 FAD may be calculated. To accomplish this, the software internally performs a sensitivity analysis with respect to load for the current assessment and calculates a table of values of J_e vs. L_r over the range of $0 \leq L_r \leq 4.0$. This table may then be interpolated over during the assessment to retrieve values of J_e .

10.4 Treatment of Stress Inputs

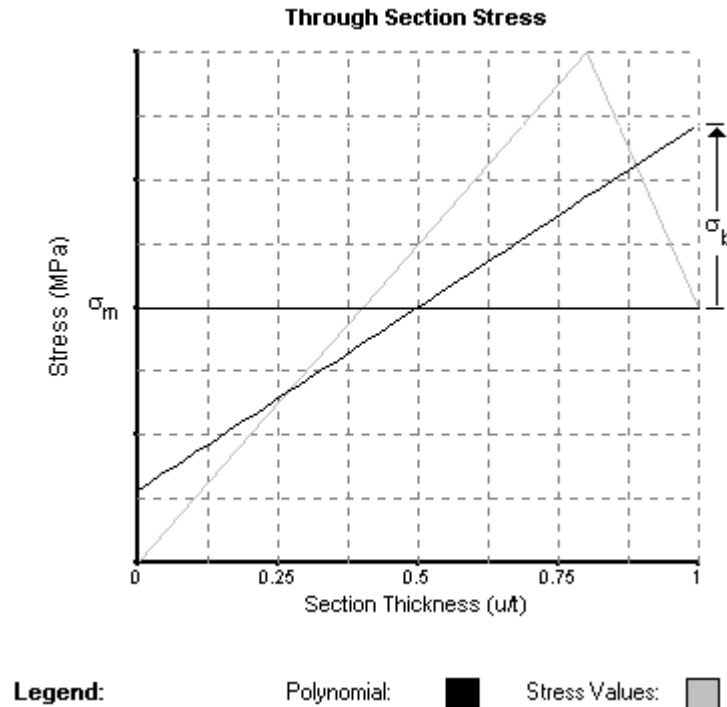
Single value loadings are passed directly to the engine to be used in the calculation of K and L_r values.

The treatment of arbitrary stress fields is more complex. The polynomial stress field entered directly by the user or the polynomial stress field automatically fit to the users data is taken to represent the through section stress in the body. This polynomial is $\sigma = f(u/t)$ where t is the section thickness. The order of this polynomial is referred to here as the given order.

Many calculations take a polynomial stress field as input. The order of this polynomial is referred to here as the requested order. In this situation, the following occurs:

Case	Action
Given order $\leq$ requested order	Use polynomial directly.
Given order $>$ requested order	Sample the polynomial and refit polynomial of requested order to the sampled data.

Secondary stress fields are automatically sampled and refit to the requested order in the calculation of K_I^s .



The majority of other calculations in the engine require σ_m and σ_b as inputs. In this case a first order polynomial is fitted to the data as described above and the values are taken as shown graphically above. This is true of both primary and secondary stress fields.

For K calculations in the engine the stress polynomial is required as $f(u/a)$ rather than $f(u/t)$. In this case each coefficient is multiplied by $(a/t)^n$ to convert the polynomial to $f(u/a)$.

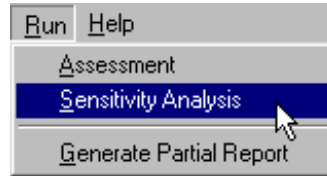
10.5 Calculation of K_r , L_r

Once the input data has been validated and the stress inputs to the K and L_r functions in the engine are calculated, the appropriate functions in the engine are called to calculate L_r , K_I^s , K_I^p , r and hence K_r .

The factor r is calculated following the detailed methodology from R6, Appendix 4. Factors ψ and ϕ which are required in the calculation of r are calculated using the tabular data from tables A4.1 and A4.2. The equivalent efficient stress intensity factor K_p^s is defined using linear elastic approximation outlined in A4.5.2.

10.6 Sensitivity Analysis

A sensitivity analysis is run by clicking on the *Sensitivity Analysis* option from the *Run* menu as shown below.



Sensitivity assessments currently allow a multiplier to be applied to either the applied loading or to the crack length. The software runs the assessment a specified number of times varying the multiplier between user-specified lower and upper limits. The resulting assessment points are plotted as a line on the FAD. Only the first value from a sensitivity assessment is plotted on the resulting CDF.

Once an assessment has been completed by the **SINTAP** software, the user may now proceed to accessing the results of the fracture risk assessment.

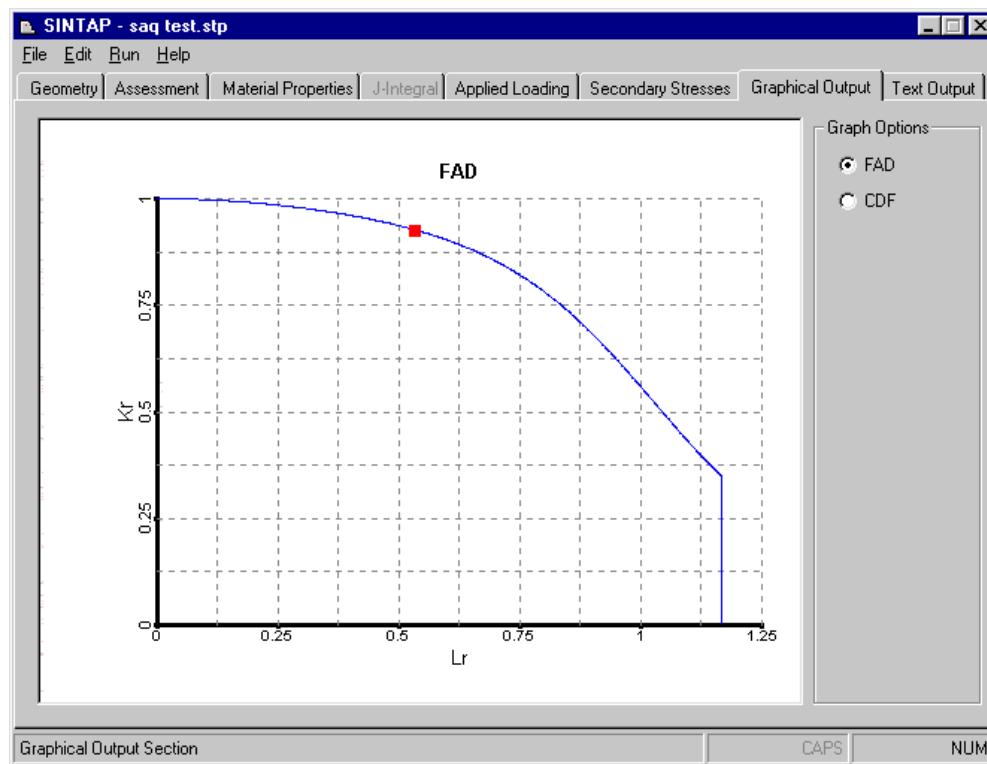
11 Output

11.1 Introduction

Once an assessment has been completed the outputs of the fracture risk assessment may be accessed by the user including the FAD, CDF and the text output produced by the software.

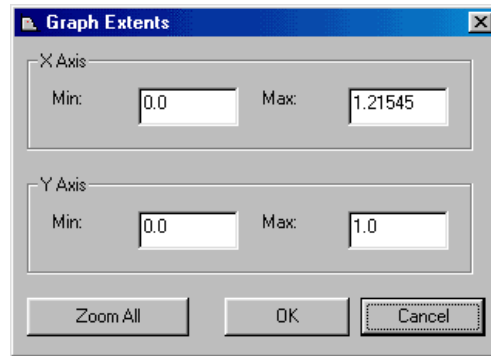
11.2 Failure Assessment Diagram

The failure assessment diagram is drawn on the results screen along with the assessment point $\{L_r, K_r\}$ as shown below. Selection of the FAD shape used has already been covered in detail in the *Assessment* section part of this document.

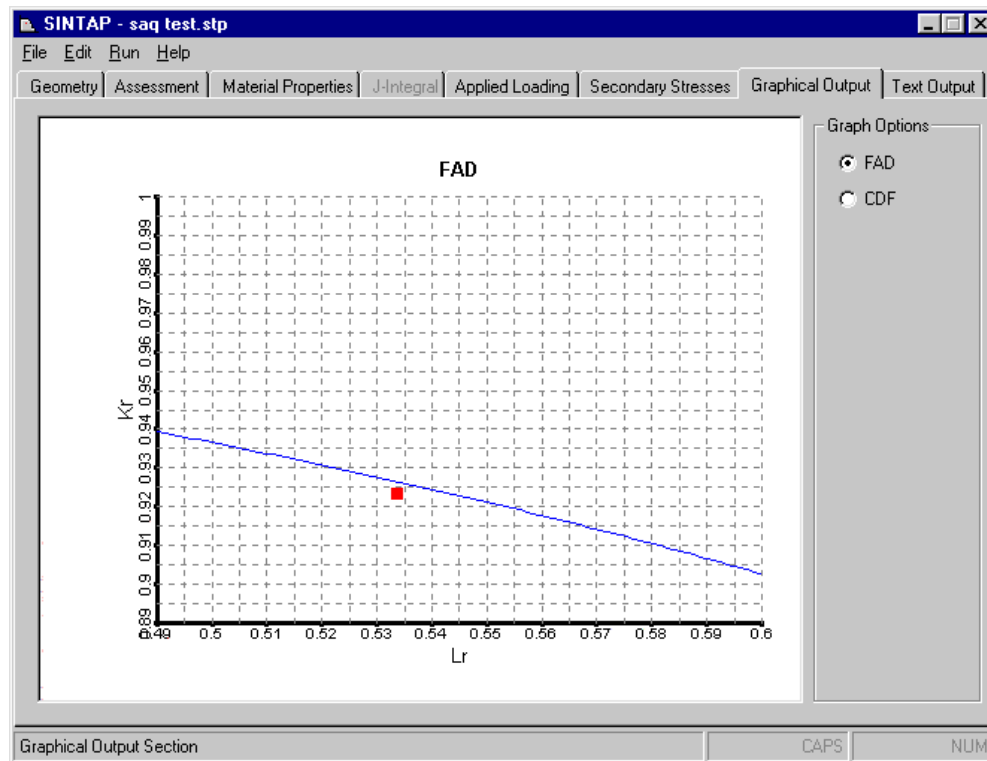


Points which lie within the FAD indicate that the case is acceptable, outside indicate that the case is unacceptable.

If the user wishes to view more closely any particular area of the FAD, it is possible to zoom in on the desired portion. This is particularly useful if the failure assessment point lies close to the failure assessment line as in the above FAD, and it is unclear whether or not the point lies inside the safe region or not. By double clicking on any part of the FAD, the following dialog box is displayed.

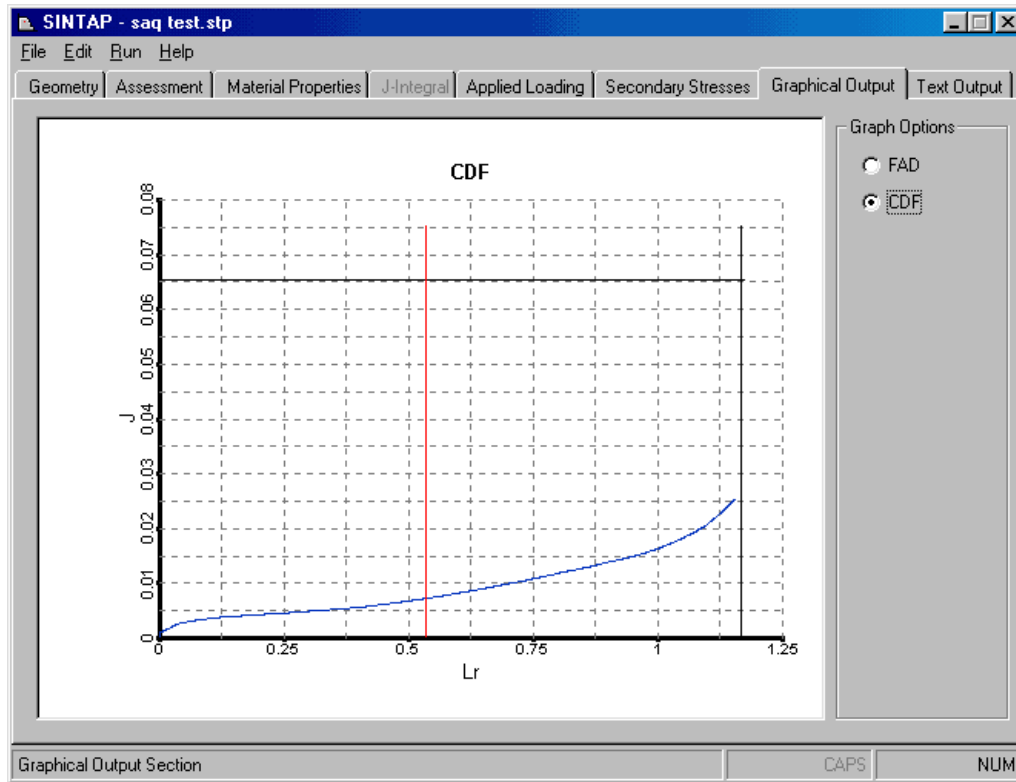


The user may now select boundaries for both the x and y axes to zoom in on a particular area of the diagram. The *Zoom All* button is used to view the FAD in its default original display. For example, the user could set the minimum and maximum for the x axis as 0.5 and 0.6, respectively, while also setting the minimum and maximum for the y axis as 0.9 and 1.0, respectively. This would give an enlarged view of the area around the failure assessment point as shown below. The zoom facility is also available for the CDF diagram.



11.3 Crack Driving Force

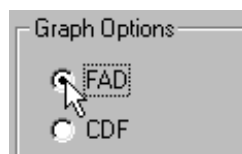
The CDF diagram is also drawn on the results screen. A vertical line is drawn at $L_{r_{max}}$ and a horizontal line is drawn at the limiting value of J . The assessment value of L_r is then shown as another vertical line.



The CDF diagram may be scaled in the same way as the FAD, by double clicking and setting the max and min values for the axes.

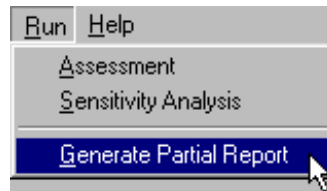
11.4 Graph Options

The user may alternate between the failure assessment diagram and the crack driving force diagram by clicking on one of the option buttons to the right of the results diagram as follows.

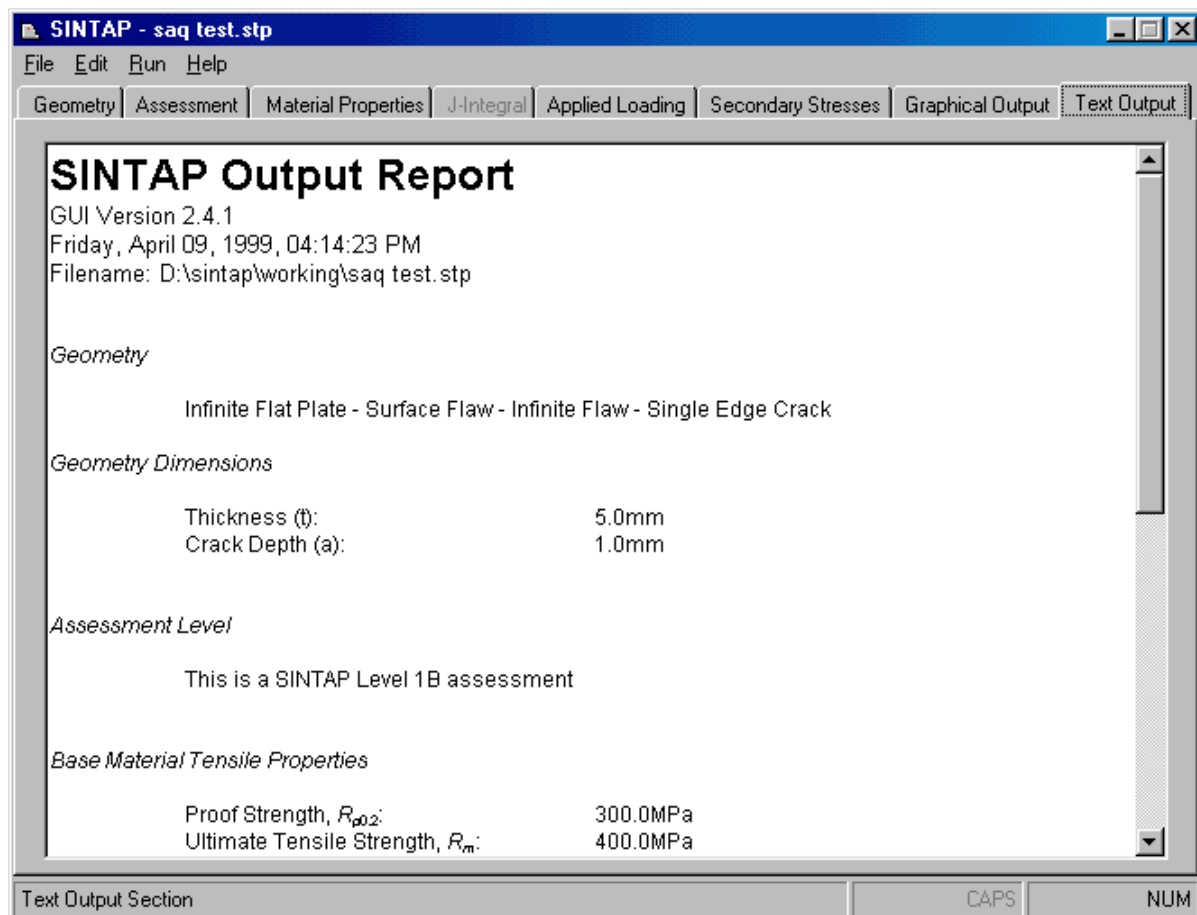


11.5 Text Output

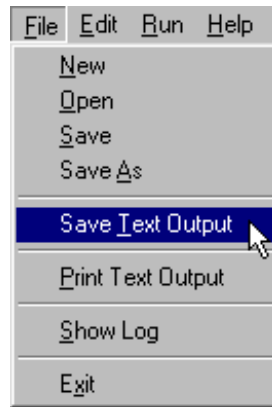
The user may generate a text output by clicking on the *Generate Partial Report* option from the *Run* menu as shown below. The report contains a summary of the user input data along with the results of the fracture risk assessment if an analysis has been run.



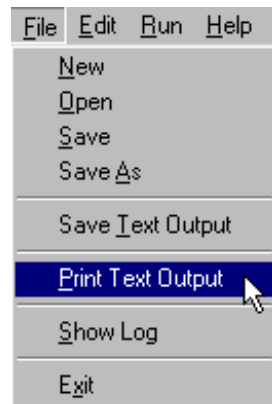
A typical **SINTAP** report would look like the sample text output shown below.



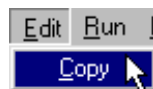
The text output may be saved to a text file (rich text format) by clicking on the *Save Text Output* option from the *File* menu as shown below.



The text output may also be directly printed by clicking on the *Print Text Output* option from the *File* menu as shown below.

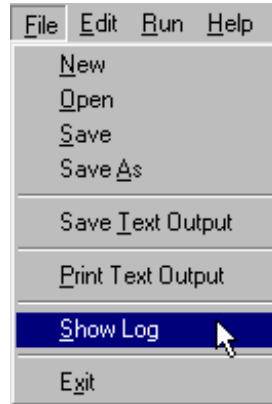


The user also has the option of importing graphical displays from **SINTAP** such as the stress/strain curve, applied loading diagram, secondary stress diagram, FAD, CDF, and Text Output, to other software packages by clicking on the *Copy* option from the *Edit* menu as shown below.

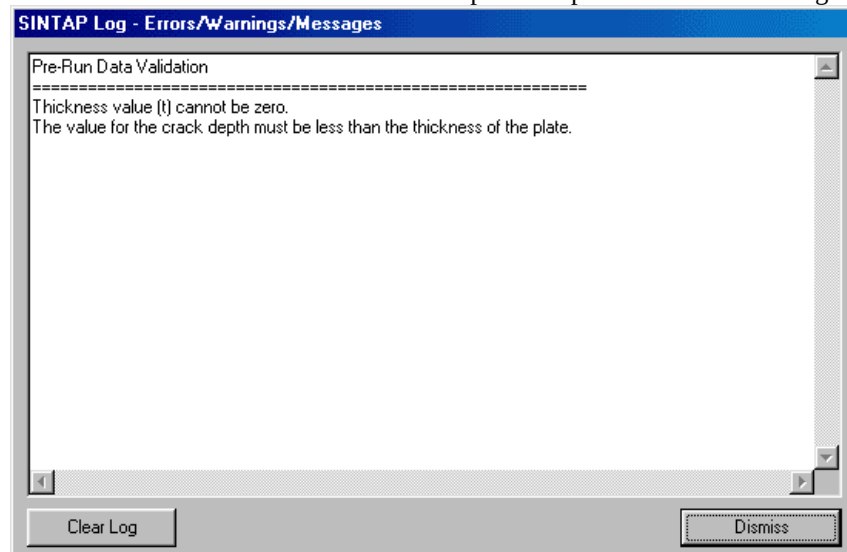


11.6 Error messages

Any error messages generated during an analysis session are stored in a log file and may be viewed by the user by clicking on the *Show Log* option from the *File* menu as shown below.



A typical log entry showing errors that occurred during the user input data validation stage of an analysis is shown below. This is to warn the user of incorrect values inputted to prevent errors occurring in the engine.



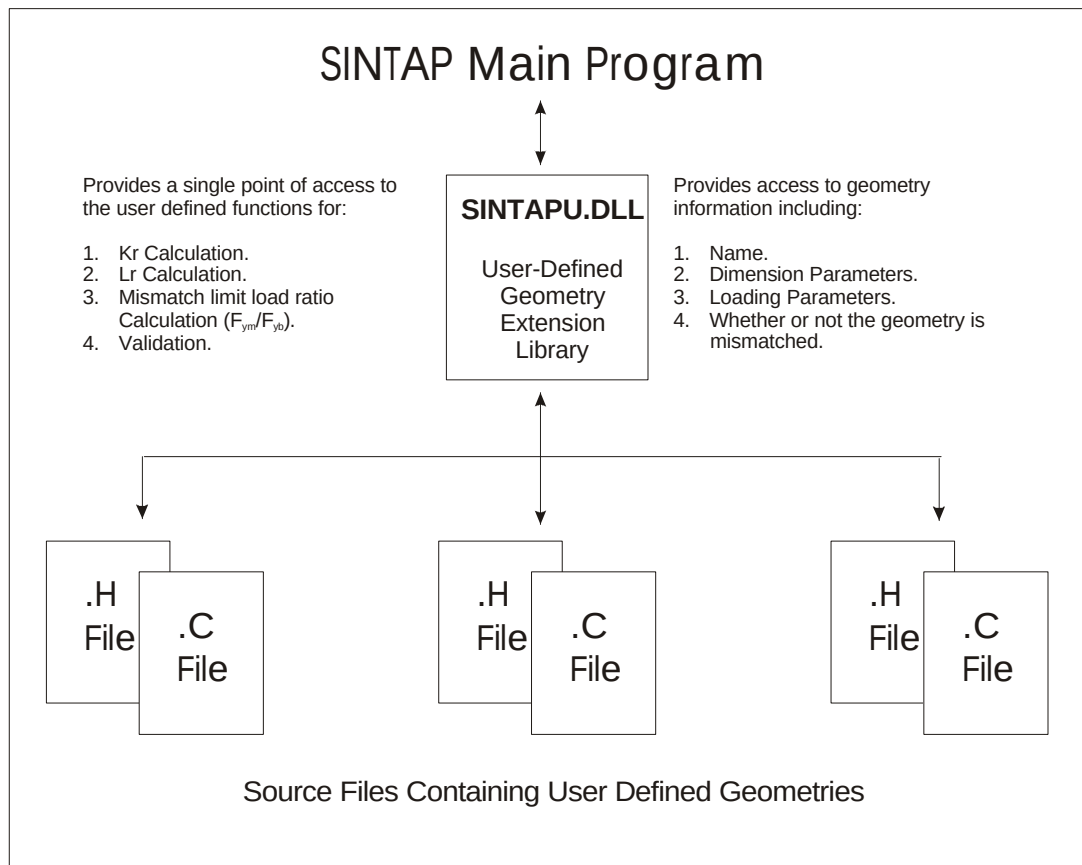
12 User Geometry Extension

12.1 Introduction

The **SINTAP** GUI has been modified so that an arbitrary number of user-defined geometries may be implemented through a user-programmed library.

This library is similar in style and function to the **SINTAP** engine. Its advantage is that it allows the system to be extended with an arbitrary number of additional user-defined geometries without modification to the main Graphical User Interface (GUI).

12.2 Concept



12.3 Requirements

To make use of this extension, the user must have a copy of Microsoft Visual C++ 6.0 or later and a moderate level of competence with the C programming language.

12.4 Files

The user geometry extension project contains several files. These fall into two general groups, those modifiable by the end user and those not intended to be modified.

Project Files Not User Modifiable

SINTAPU.DSP	Visual C++ 6.0 project file.
SINTAPU.C	Main source file for the DLL.
SINTAPU.DEF	Module definition file for the DLL.
GEOMS.H	General declarations.

Project Files User Modifiable

GEOMS.C	Main interface to the user defined geometry functions.
GEOM1.C, GEOM1.H	Template files for the first user geometry.
GEOM2.C, GEOM2.H	Template files for the second user geometry.

Modifying files indicated a “not user modifiable” may cause the **SINTAP** software to stop functioning.

12.5 Creating a User Geometry Solution

Creating a user geometry solution can be broken in two distinct steps:

- The preliminary preparation step of modifying existing files to include another geometry.
- Actually programming the SIF, limit load solution, mis-match ratio solution and verification function.

12.5.1 Preliminary

Open the project by loading the project file, SINTAPU.DSP, into Microsoft Visual C++ 6.0.

If this is the first or second user geometry being implemented, modify one of the existing templates (GEOM1.C and GEOM2.C). Otherwise, duplicate one set of template files. For example, if this is the third user geometry, copy GEOM2.C and GEOM2.H to GEOM3.C and GEOM3.H respectively.

If you have duplicated a set of template files, include the new GEOMx.C file in the project by using the ‘Add to Project’ option in the ‘Project’ menu.

Edit your set of template files. Rename the functions in these files appropriately. For example, if this is the third user geometry, rename KrGeom2 to KrGeom3, LrGeom2 to LrGeom3 and so on. The functions may have any name you like, so long as they are unique. However, it is recommended that you keep to the existing naming scheme for the sake of consistency.

Edit GEOMS.C. Ensure that the header file for your new geometry, GEOMx.H, is included at the top of the file.

Also in GEOMS.C, edit the geomInfo table and include an entry for your geometry. This table holds the following information:

Geometry Name	The name of the geometry.
Dimension Labels	The labels of the dimensions required for the geometry. A newline character separates entries.

	Dimensions relating to crack size must be placed at the start of the list.
Load Labels	The labels to use when requesting loading information for the geometry. A newline character separates entries.
Crack Dimensions	The number of entries in the Dimension Labels which relate to crack size. These entries must be at the start of the list.
Mis-match	A BOOLEAN value that indicates whether or not this is a mis-matched geometry.

Still in `GEOMS.C`, find the function `userKr`. This function contains a switch statement which allows the calling program to access the user programmed Kr solution in one of the user geometry files. Add a call in this function to your SIF function in the `GEOMx.C` file.

Repeat this procedure in the `userLr`, `userLLR` and `userVerify` functions. Add calls to your custom limit load, mis-match limit load ratio and dimensional verification functions. Search for the text "TO DO:" to assist you in locating the portions of the `GEOM.C` file which require modification.

It is advised that you test your modifications at this point by compiling the project. Perform any debugging necessary before proceeding to program your custom functions as described in the next section.

12.5.2 Creating Your Custom Geometry Functions

There are four different custom functions for the user to create for a particular geometry.

- A Kr solution.
- An Lr solution
- An Fym/Fyb solution for mis-match [optional].
- A geometry dimension verification function.

Some general notes on the function inputs received from the program are presented first followed by a description of the custom functions.

Note on dimensional and loading inputs received from program.

The dimensional and loading inputs are received from the calling program in the form of pointers to an array of doubles.

The content of these arrays corresponds to the 'load label' and 'crack dimension' entries specified in the `geomInfo` table in `GEOMS.C`.

For example, if the 'crack dimension' entry was:

```
"Crack depth, a [mm]\nCrack Length, l [mm]\nThickness, t [mm]"
```

remembering that C arrays index from zero, you could write:

```
double a = dims[0];
double l = dims[1];
double t = dims[2];
```

at the top of your custom function. You would then be free to use these variables in the rest of your calculations.

Note on the buf and bufsize arguments

All functions include a pointer to a character buffer in their argument list, `buf`. This buffer is intended to act as a mechanism for passing messages between the DLL and the calling program. The calling program interprets the insertion of any message into this buffer as the indication of an error.

The `bufsize` argument indicates the length of the buffer in characters. Although this buffer is usually reasonably large, no assumptions should be made as to its size. The programmer should also provide for the possibility that the pointer `buf` is NULL and that no buffer has been made available. Failure to account for these possibilities could result in abnormal program termination or corrupted results.

Kr solution

The program allows Kr solutions as a function of the dimensional inputs, the loading inputs and K_{mat} . The general form of a custom Kr function is as follows:

```
double KrGeomN( double * dims,
                double * load,
                double  Kmat,
                char   * buf,
                long    bufsize )
```

dims

A pointer to an array of doubles representing the geometry dimensions.

load

A pointer to an array of doubles representing the loading case.

Kmat

The current value of K_{mat} as determined by the **SINTAP** software.

buf

A pointer to a character array. Used for passing back error strings to the calling program.

bufsize

The size, in characters, of the array pointed to by `buf`.

The return value is a double containing the value of Kr for the geometry as a function of the specified inputs.

Lr solution

The program allows Lr solutions as a function of the dimensional inputs, the loading inputs and the proof or yield strength. The general form of a custom Lr function is as follows:

```
double LrGeomN( double * dims,
                double * load,
                double  YS,
                char   * buf,
                long    bufsize )
```

dims

A pointer to an array of doubles representing the geometry dimensions.

load

A pointer to an array of doubles representing the loading case.

YS

The proof or yield strength of the material.

buf

A pointer to a character array. Used for passing back error strings to the calling program.

bufsize

The size, in characters, of the array pointed to by buf.

The return value is a double containing the value of L_r for the geometry as a function of the specified inputs. An implementation of this function which returns a constant value will cause the calculation of J_e in the **SINTAP** GUI to fail.

Mis-match Limit Load Ratio Solution

The program allows mis-match limit load ratio solutions as a function of the dimensional inputs and the mis-match ratio M . The general form of a custom limit load ratio function is as follows:

```
double LLRGeomN( double * dims,
                  double  M,
                  char   * buf,
                  long    bufsize )
```

dims

A pointer to an array of doubles representing the geometry dimensions.

M

The ratio of the weld proof or yield strength to the base material weld proof or yield strength.

buf

A pointer to a character array. Used for passing back error strings to the calling program.

bufsize

The size, in characters, of the array pointed to by buf

The return value is a double containing the mis-match limit load as a function of the specified inputs. If the user geometry being programmed is not mis-matched, this function should simply return 1.0.

Geometry Input Verification Function

The program allows verification as a function of the dimensional inputs. The general form of a custom verification function is as follows:

```
BOOL VerifyGeom1( double * dims,
                  char   * buf,
                  long    bufsize )
```

dims

A pointer to an array of doubles representing the geometry dimensions.

buf

A pointer to a character array. Used for passing back error strings to the calling program.

bufsize

The size, in characters, of the array pointed to by buf

The return value is TRUE or FALSE depending on whether or not the specified dimensions are valid for the user geometry. In it's simplest form, this function just returns TRUE without checking any inputs.

12.6 Installing the completed user geometry extension

Once the user geometry extension has been completed, it must be installed.

Under standard installation, this is achieved by copying the compiled SINTAPU.DLL file to the Windows system directory.

Where the **SINTAP** software exists on a central server, the SINTAPU.DLL file must be copied to the server directory where the software is installed.

In both cases, the existing SINTAPU.DLL in the destination directory can be replaced.