FEWeld

Demo Guide

Version 2000.2

Weaver Engineering
150 Nickerson Street, #102
Seattle WA 98109

Phone :   (206) 352-8027
Fax: (206) 352-8035
email   info@weavereng.com
web   http://www.weavereng.com

Trademark Information

Throughout this manual, and the software, you will see references to other applications and trademarks which are the property of various companies:

- Ansys is a registered trademark of ANSYS, Inc.

- Cosmos and Cosmos/m are registered trademarks of Structural Research and Analysis Corporation.

- Windows, Windows 95, Windows 98, and Windows NT are registered trademarks of Microsoft Corporation.

License Agreement

This package contains your FEWeld program. Please read this agreement before installing the software. If you do not agree with the terms of the agreement, return this package as is and Weaver Engineering Co. will, upon receipt, refund your purchase price. No refund will be made for packages returned with missing or damaged components.

1. License: Weaver Engineering Co. grants you a nonexclusive right to use the enclosed program. The enclosed program may be run on only one computer. This license may not be transferred without the written authorization of Weaver Engineering Co.

You may not: (a) distribute copies of the program or user manual to others; (b) provide, sell, or sublicense access to, or use of, the program to other third parties; (c) modify, use, copy, reverse assemble, reverse compile or transfer this program or user manual, or any copy thereof without prior written consent of Weaver Engineering Co.

You may: make copies of this program solely for backup purposes. Such "backup copies" must be so labeled.

2. Copyright: This program and user manual is protected by a copyright which is owned by Weaver Engineering Company. You may not copy the program or user manual except for the purpose of backup, as stated above, and to load the program into a computer for executing the program. All other copies of the program and user manual, except as stated herein are in violation of this agreement and Weaver Engineering Co.'s copyright.

3. Term: The term of this license is effective until terminated by your failure to comply with this agreement and Weaver Engineering Co.'s copyright. You may terminate the license by destroying the program, user manual, and all copies thereof, and returning any security devices along with written notice to Weaver Engineering Co. of such termination.

4. Restrictions: This program contains software and technical data that is subject to the export laws, orders and restrictions of the United States. You may not directly or indirectly import, export, re-export, transship, or use the program in any country or destination that requires an export license, other approval for export, or is in any way contrary to U.S. laws, without having obtained such permission.

Any license to the U.S. Government is subject to the restricted rights as stated in the following documents and any amendments and successors thereto: For the Department of Defense, DFARS 252.227-7013, Rights in Technical Data and Computer Software; DFARS 252.227-7037, Validation of Restrictive Markings on Technical Data. For the National Aeronautics and Space Administration, NFARSUP 18-52.227-19, Commercial Computer Software - Restricted Rights. For other civilian agencies FAR 52.227-14, Rights in Data - General and FAR 52.227-19 Commercial Computer Software - Restricted Rights. These clauses are incorporated into this agreement by reference, with the same force and effect as if they were given in full text. In no case however, will any interpretation or modification of these clauses grant you any rights in the enclosed software or data beyond those of using it under the terms of the remainder of this agreement.

5. Limited Warranty: (a) Weaver Engineering Co. warrants that the CD-ROM upon which the enclosed program is recorded is free from defects in materials and workmanship when used under normal conditions, and (b) Weaver Engineering Co. warrants that the program will perform substantially as described in the User Manual.

6. Disclaimer: Weaver Engineering Co., its agents, employees and distributors shall not be liable for technical, editorial, or other errors or omissions which may be contained in, or the negligent preparation of this material. Weaver Engineering Co. hereby disclaims any express or implied warranty that the enclosed software, documentation, or other materials are fit for any particular purpose.

Weaver Engineering Co., its agents, employees and distributors shall not be liable for compensatory, punitive, or other damages of any nature, arising from, or allegedly arising from any breach of the Limited Warranty above, nor shall they be responsible for claims for lost profits or revenues, lost data, or recreating lost data, substitute programs, nor any claims of third parties. This disclaimer shall apply regardless of any notice to Weaver Engineering Company of the possibility of such damages.

7. Exclusive Remedies: Upon breach of the Limited Warranty, stated above, Weaver Engineering Co. shall at its option: (1) provide a corrected program to you, or (2) provide a corrected User Manual, or (3) refund the fee paid for the program, without charge to you. In no case shall Weaver Engineering Company's liability exceed the amount of the license fee paid, or $100 (U.S.), whichever is greater. These are the exclusive remedies agreed in case of breach of the Limited Warranty.

8. Acknowledgement: You acknowledge that you have read this agreement and are bound by its terms, that this agreement is the complete and exclusive agreement of the parties hereto, and that it supersedes any and all communications, proposals, representations, and agreements, oral or written, previously or subsequently made between the parties. This agreement shall be enforced and interpreted in accordance with the laws of the State of Washington and applicable laws of the United States. If any provision of this agreement is determined to be invalid or unenforceable, the validity or enforceability of the other provisions shall not be affected.

9. Notices: All Notices required by this agreement, questions pertaining thereto or concerning the enclosed program and User Manual should be directed to:

> Weaver Engineering Company
> 150 Nickerson Street, Suite 102
> Seattle WA 98109
> U.S.A.

Install this software only if you agree to the terms of this License Agreement

Table of Contents

Chapter 1.   FEWeld Overview

1.1.   *What Is FEWeld*

FEWeld is a general mathematical tool for calculating weld parameters to meet performance criteria from the results of finite element analysis with shell elements[1]. It's primary use is for performing calculations on linear welds – fillet welds, groove welds, and with special consideration, seam welds. Spot and plug welds are currently not supported.

Chapter 3, Theoretical Overview, provides a full description of the mechanics behind FEWeld. In short, FEWeld calculates joint design parameters whose values determine the predicted performance of the weld with respect to design requirements. For example, the throat size of a fillet weld is a joint design parameter while throat shear stress may be the predicted performance and the throat shear allowable the performance criteria. At each node[2] of each weld joint, for every load case, FEWeld extracts the loads transmitted by the joint as well as the full stress state at both sides of the element in terms of the local weld joint coordinate system. This information, in addition to the part thickness, is fed into the "Weld Formulation" a standard or user defined set of mathematical expressions that characterizes the performance in terms of weld parameters and calculate parameter values that meet the performance criteria. The formulation can solve either explicit expressions for the design parameters from the load and/or stress data or there is a constrained goal seeking facility for more complex formulations.[3]

FEWeld is implemented as a database which enables rapid evaluation of configuration variations and organizes large models with numerous load cases effectively. Detail and summary output are available in report format or readable into spreadsheets for further analysis.

FEWeld is not a finite element analysis package. FEWeld currently works with ANSYS and COSMOS/M finite element analysis packages.

While FEWeld's primary use is for calculation of weldment performance with respect to code requirements, FEWeld is not a "Code in a Box" – sound engineering judgement and understanding of the problem at hand are required for successful use of this tool. To that end, the formulations that are provided with FEWeld have descriptions of what they are – i.e. "Single Sided Weld based on Throat Shear" with no direct reference to code applicability. It is up to the user to make the appropriate characterization and define the performance criteria for the situation

1.2. *Who is it For?*

Successful use of FEWeld requires an understanding of weld performance characterization and experience in finite element modeling. The user should have a strong background in mechanics of materials (a.k.a. strength of materials), familiarity with welding and weld joint design, and have full understanding of the performance requirements (i.e. code) against which the weldment is being designed. These abilities need not rest with one person. It is plausable for a fea analyst to

---

[1] Shell Elements: Sometimes called Plate Elements. They are three and four sided elements with no geometric thickness (like a surface) – the thickness is supplied as a property. They siimulate both in-plane and out-of plane loads and are good for representing structural shapes, plates, piping, etc. See Chapter 3.

[2] Nodes are located at the corners and possibly the midsides of each element in a finite element model

[3] Expressions of the following form can be calculated {Iterate the weld_size, calculate the weld stress, minimize the weld area such that the calculated stress is less than or equal to the stress allowable and the weld size is greater than or equal to the minimum weld size.}

build and run the model, while the weld joint person develops weld formulations, evaluates results, and makes recommendations.

1.3. Background, Current State, and Future

1.3.1. Background

FEWeld was originally developed at Weaver Engineering as an in-house tool for evaluation of weldments against the AWS structural welding codes – determination of throat shear and weld size requirements from the results of finite element analysis. This enabled us to gain the benefits of load path predictions of finite element analysis for complex structures and apply the classical weld load – throat shear formulations upon which the code allowables are based.

FEWeld originated as a command-line tool and the analysis was labor intensive for complex structures where the configuration was undergoing iterations. This original configuration had hard-coded formulations for single sided welds (fillet or partial penetration groove), double sided fillet and double sided partial-penetration groove welds. It could only handle straight welds (no pipe or formed sections). Circa 1995.

1.3.2. Current State

The current FEWeld handles general joint geometry (pipe connections, etc), provides a database driven GUI for entering the weldment configuration data, performing the analysis and evaluating the results.

There is a Weld Formulation editor that uses a simplified 'c' like mathematical language that allows for very general weld formulations based on loads, stresses, orientation and geometry. The formulation can specify user inputs for each weld (such as skew angle, prep size, restraint) that are then incorporated into the GUI for parametric weld definition, and outputs are designated as the results of the calculation for charting, summary reports, and export. Each weld in the database has one formulation associated with it.

Load cases are included in the database. In the simple case, a stress (or load) allowable is specified for each load case. For more complex situations where different parts of the structure have different allowables under the same loading, multiple 'allowables sets' can be defined, each with it's own value for each load case. Each weld is then associated with one allowables set.

Each FEWeld file can accommodate multiple configurations (associated with multiple fea models) for rapid comparison between design iterations.

FEWeld generates concise, professional reports of the configuration definitions, results summaries, and detail results formats. There is a chart browser which easily cuts through the data by load case, by weld, or by configuration.

The configuration definitions, results, and a copy of the formulations used for the calculations are stored in the FEWeld file. This is important for archiving and maintenance of definition integrity.

Chapter 2.   Installation

NORMAL INSTALLATION (Windows NT4, 95, & 98)

If you have FEWeld installed, delete the 'setup' directory that exists as a subdirectory of your installed FEWeld program directory before running the update file. The installation will not work if you don't. (Un-install and update did not work with previous FEWeld releases).

FROM CD-ROM

Execute Setup.bat on the CD or Setup.exe in the 'Program' directory of the CD.

FROM FTP UPDATE DOWNLOAD

The download, 'FEWeld_Update_DD_MMM_YYYY.exe' is a self extracting .zip file that unzips the install files and launches the installer upon completion. If you cancel the setup, run setup.bat from the directory where you unzipped the files.

Note for MS Access 2000 Users:

90% of the time, FEWeld will install correctly on computers that also have Microsoft Access 2000 (Part of MS Office 2000 Premium) installed.

For the other 10%, when FEWeld Starts, MS Access 2000 try to will open it, and you will get a message asking if you would like to convert the file, which you can't do. In this case, copy the FEWeld.mde file that is included in the 'If_It_Doesnt_Work_Out_of_The_Box_With_MSAccess2000' subdirectory of the 'Program' directory of this distribution to the   FEWeld program directory on your computer .

WINDOWS 2000 INSTALLATION:

FROM CD-ROM

Run the Setup.exe contained in the 'Program_for_w2000' instead of the 'Program' directory.

FROM FTP UPDATE DOWNLOAD

The download, 'FEWeld2000_Update_DD_MMM_YYYY.exe' is a self extracting .zip file that unzips the install files and launches the installer upon completion.

Note for MS Access 2000 Users:

For the windows 2000 installation, if you have MS Access 2000 installed but not the 'Graph' office tool installed, FEWeld charts will not work. You will need to install Graph9 from the installation disk that MS Access 2000 came on. (If MS Access is not installed, this is not an issue.)

Refer also to the *'Readme.txt'* on the cd-rom or download for update information.

2.1. System Requirements

- Pentium Processor
- Windows 95/98, NT 4.0 or 2000.
- 20 MB Free Hard Disk Space for Program
- 64 MB Ram Recommended

2.2. Getting Started

You are strongly encouraged to read the entire manual and work the example problems before using FEWeld for earnest engineering work.

2.2.1. Licensing

You must obtain a license key before you can run FEWeld outside of evaluation mode. To do this, you need to provide Weaver Engineering with the host id of your computer. This id is based on the network card you have installed.

FEWeld comes with a utility program for obtaining the host_id called *gethost.exe,* which is installed in the FEWeld application directory. Running this program will create a file called *host.txt* in the same directory. Email or fax the contents of this file to Weaver Engineering and we will send you your license string. If it is via email, it will be attached as *license.txt*. Copy this file to your FEWeld application directory. If it is faxed, create a file called *license.txt* in that directory using a text editor such as Notepad, and type in the license string exactly as given.

FEWeld should now run.

Note: If you have multiple network cards in your computer, and you want FEWeld licensed to a card other than the one provided by gethost.exe, call us.

If you do not have a network in your computer, let us know, we will be providing dongle licensing soon.

2.2.2. Evaluation Mode

FEWeld runs without a license in evaluation mode. The following restrictions apply:

o Weld Formulations cannot be created or edited.

o Files can only be created and edited. Existing files, except sample files as below, cannot be re-opened.

o The sample files provided in the distribution can be opened and edited once only.

2.2.3. Training

Weaver Engineering offers training periodically. Check our website at www.feweld.com or call us for schedules.

2.2.4. Technical Support

The purchase of FEWeld provides one year of technical support. After that, annual maintenance will continue technical support, which also provides continuous updates to the software.

Telephone: (206) 352-8027

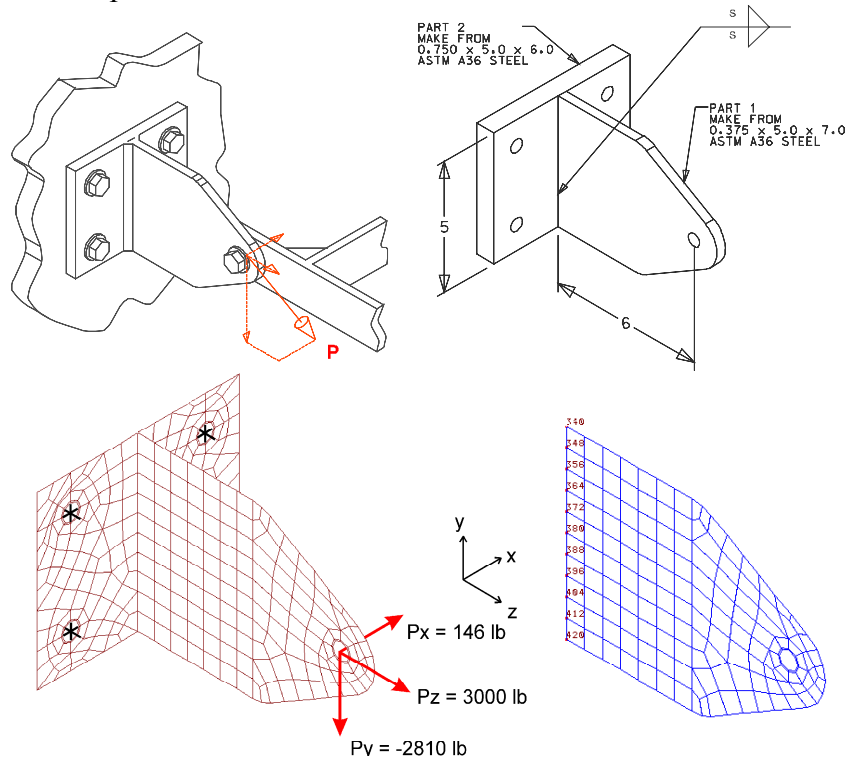Fax: (206) 352-8035

Email: techsupt@feweld.com

Chapter 3.   Theoretical Overview Excerpts from the Manual

The formulation used for a finite element shell model is that of full penetration welds at every joint. Although the loads carried through joints are calculated by FEA, they are not readily usable for weld calculations.

FEWeld extracts from the fea results load and stress information and presents it in a form that is readily usable for weld calculations. Additionally, FEWeld provides a robust mathematical environment for defining and performing calculations of weld requirements based on the load state and the performance criteria.

Figure 3-1   A Fillet Welded T-Bracket Illustrating the Terminated Part and Weld Joint Nodes
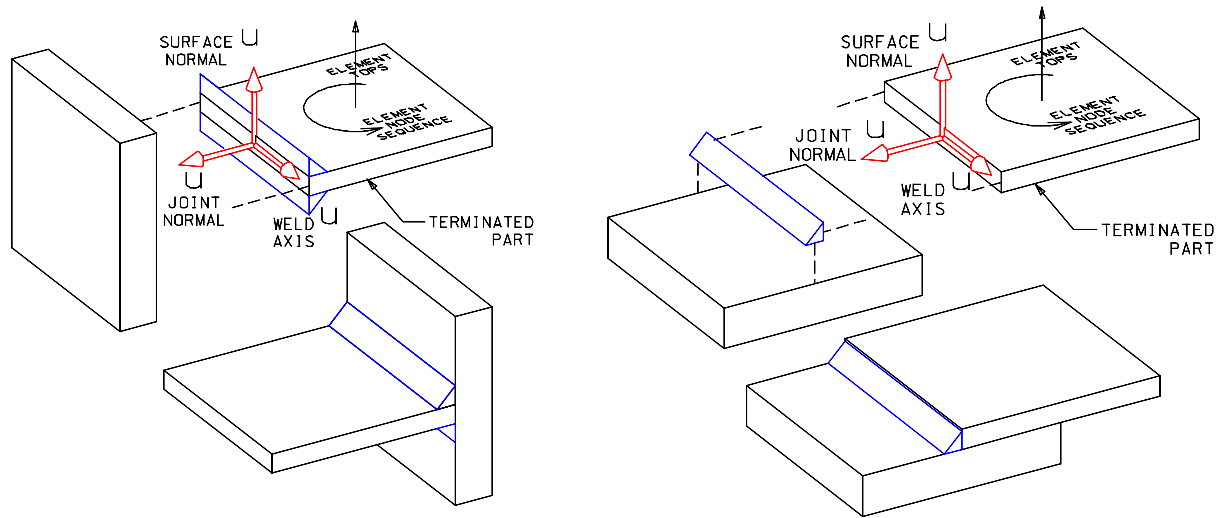
3.1.   The Terminated Part

Figure 3-1 above depicts a t-bracket constructed from a Part 1 Stem and Part 2 Base. By inspection, the Part 1 Stem is the only terminated part in this joint. The concept of a terminated part is central to modeling for FEWeld.

The lower right of the figure shows the elements of the stem and the nodes in the weld joint. The nodal stresses in these elements at the joint nodes represent the stress state in the stem at the weld. The z component of the stresses are directly the from the tractions (loads) being transmitted through the weld. Specifically, σ_{zz} is due to the combination of the Pz Load and the bending moments induced by Px and Py. σ_{zx} is due to Px transmitted through the weld, and σ_{yz} is due to Py transmitted through the weld.

From these stress traction components for the element top and bottom and the part thickness, the load components and bending moment about the weld axis per unit length can be calculated.

On the other hand, for the part 2 base, for elements on either side of the weld joint, the traction through the element edge associated with the weld joint is a combination of the loads from the weld and the loads from base plate elements on the other side of the weld.

For t-joints and lap joints, there is one terminated part for each weld. For butt and corner joints, both parts entering the weld are terminated. Many joint configurations, such as flare-v groove welds, have no terminated part on their own. Similar to modeling welds with solid elements, the weld connection must be modeled with shell elements to create a terminated part. Several examples of this technique are given an discussed in the FEA Modeling Guidelines, Chapter 5.

Figure 3-2 Weld Joint Coordinate System (wjc) of the Terminated Part

3.2.   Weld Joint Coordinate System

Figure 3-2 above shows depictions for a terminated part of a t-joint and a lap joint. The Weld Joint Coordinate System (wjc) is defined in terms of the local geometry of the elements with edges in the joint at each node of the joint. The three directions of the wjc are: The Weld Joint Normal, $\mathbf{u}_j$, the Weld Axis, $\mathbf{u}_w$, and the Surface Normal, $\mathbf{u}_s$. The order of the coordinates is j,w,s.

- The Weld Axis, $\mathbf{u}_w$, runs in the direction of the weld seam, and is oriented such that it points in the counter-clockwise direction around the element top, the same direction as the nodal sequence of the element definition.

- The Surface Normal, $\mathbf{u}_s$, is the surface normal of the element 'top' at the location where the weld joint coordinate system is being evaluated.

- The Weld Joint Normal, $\mathbf{u}_j$, points directly out of the element edge, in the element plane and perpendicular to the weld seam. It is perpendicular to both $\mathbf{u}_w$ and $\mathbf{u}_s$. It is defined by:

  $\mathbf{u}_w \equiv$ *weld axis unit vector*

  $\mathbf{u}_s \equiv$ *surface normal unit vector*

  $\mathbf{u}_j \equiv$ *weld joint normal unit vector*

  $\mathbf{u}_j = \mathbf{u}_w \times \mathbf{u}_s$.

The weld joint coordinate system is calculated in FEWeld locally at each weld joint node based on the elemental shape function for elements in the terminated part with and edge in the weld joint. The stress tensor in weld joint coordinates is represented as follows:

$$\begin{bmatrix} S_{jj} & S_{jw} & S_{sj} \\ S_{jw} & S_{ww} & S_{ws} \\ S_{sj} & S_{ws} & S_{ss} \end{bmatrix} \text{ where the 12, 23, 31 notation is used.}$$

3.3. *Classical Analysis of Fillet and Partial Joint Penetration Groove Welds*

The classical method of sizing fillet welds and partial joint penetration welds is to divide the load transmitted through the weld per unit length by the electrode shear allowable to come up with a minimum throat size. The joint is then designed around the throat requirement.

Calculating the load transmitted through is done by various means. The simplest, and most common method is to calculate the section properties of the weld treated as a line – weld throat area, and section modulus or moment of inertia, and polar moment of inertia. Loads are calculated at various points of the weld using beam bending formula for moments, dividing the direct axial and shear loads by the area, and using the Tr/J formula for torsion. This method ignores any stiffness effects of the connection.

A sample calculation is presented here using the weld section properties. The t-joint of Figure 3-1 is analyzed at the upper-left corner where the load is the greatest.
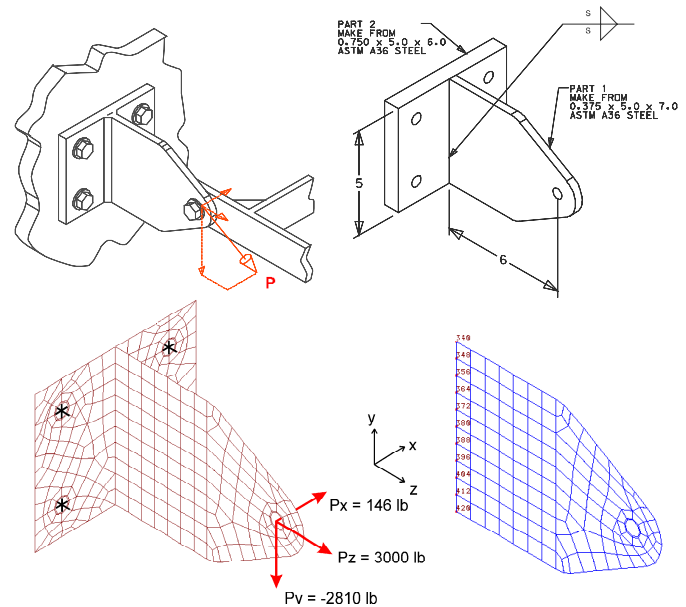
Figure 3-1 (Reproduced) T-Joint Configuration

Section Properties:

$$A_w = 2 \cdot d$$
$$= 2 \cdot (5\,in)$$
$$= 10\,in$$

$$S_{wx} = \frac{d^2}{3}$$
$$= \frac{(5\,in)^2}{3}$$
$$= 8.33\,in^2$$

$$S_{wy} = b \cdot d$$
$$= (.375\,in)(5\,in)$$
$$= 1.88\,in^2$$

Applied Loads:

Normal Load, P:

$$P = 3000\,lb$$

Shear Load, V:

$$V = \sqrt{(146\,lb)^2 + (-2810\,lb)^2}$$
$$= 2814\,lb$$

Bending Load About x, M_x:

$$M_x = (2810\,lb)(6\,in)$$
$$= 16860\,in - lb$$

Bending Load About y, M_y:

$$M_y = (146\,lb)(6\,in)$$
$$= 876\,in - lb$$

Figure 3-4 Classical Calculation

Weld Loads :

Normal Load, f_{normal}:

$$f_{normal} = \frac{P}{A_w}$$
$$= \frac{3000\,lb}{10\,in}$$
$$= 300\,lb / in$$

Shear Load, f_{shear}:

$$f_{shear} = \frac{V}{A_w}$$
$$= \frac{2814\,lb}{10\,in}$$
$$= 281\,lb / in$$

Bending Load About x, f_{bx}:

$$f_{bx} = \frac{M_x}{S_{wx}}$$
$$= \frac{16860\,in - lb}{8.33\,in^2}$$
$$= 2024\,lb / in$$

Bending Load About y, f_{by}:

$$f_{by} = \frac{M_y}{S_{wy}}$$
$$= \frac{876\,in - lb}{1.88\,in^2}$$
$$= 466\,lb / in$$

Total Weld Load, f_w:

$$f_w = \sqrt{(f_{normal} + f_{bx} + f_{by})^2 + (f_{shear})^2}$$
$$= \sqrt{(300\tfrac{lb}{in} + 2024\tfrac{lb}{in} + 466\tfrac{lb}{in})^2 + (281\tfrac{lb}{in})^2}$$
$$= 2804\,lb / in$$

Required Weld Throat Size, t_w:

$$t_w = \frac{f_w}{F_a}$$
$$= \frac{2804\,lb}{13200\,psi}$$
$$= 0.212\,in \leftarrow$$

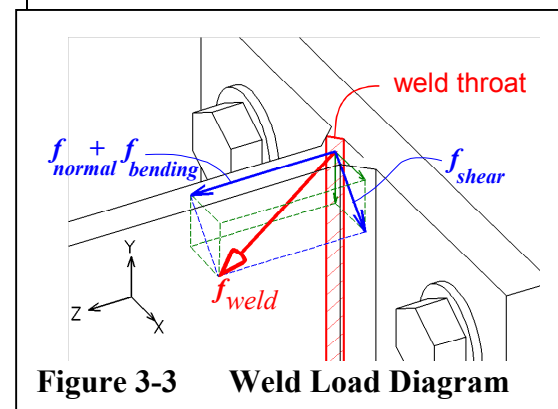**Figure 3-3**   **Weld Load Diagram**
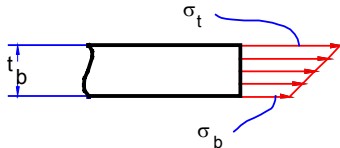
3.4. Method of Using Loads Calculated from FEA Shell Elements for Analysis of Fillet and Partial Joint Penetration Groove Welds

Node 340 of the finite element model depicted in Figure 3-1 corresponds to the top of the weld joint. FEWeld load calculation will be presented at this node and compared to the classical analysis. Refer to Figure 3-6 .

For this joint, the z axis corresponds to the negative of the 'joint normal' in the weld joint coordinates. Therefore, only the z stress components are based on the load transferred through the weld.

The equations used in classical analysis to determine part top and bottom stress due to bending, normal, and shear loads are easily reversed to determine bending, normal, and shear loads


```
          WELD JOINT STRESS TENSOR LISTS, LOAD, AND SIZE CALCULATIONS
A)STRESSES ON THE "TOPS" OF THE SHELL ELEMENTS OF PART 1 AT THE WELD JOINT:
* Selection List 1              Load case    51 Top Face     Layer 1    Cs = 3
  Node    SIG_X        SIG_Y         SIG_Z        TAU_XY        TAU_XZ       TAU_YZ
  340    3.753e-010  4.468e+003   1.956e+004   3.848e+002  -3.902e+002  -2.530e+003
  348    1.321e-009  3.808e+003   1.647e+004   3.493e+002   1.105e+002  -2.029e+003
  356    6.072e-009  3.715e+003   1.491e+004   1.948e+002   2.821e+002  -1.832e+003
   :        :          :           :            :            :           :

B)STRESSES ON THE "BOTTOMS" OF THE SHELL ELEMENTS OF PART 1 AT THE WELD JOINT:
* Selection List 1              Load case    51 Bottom Face  Layer 1    Cs = 3
  Node    SIG_X        SIG_Y         SIG_Z        TAU_XY        TAU_XZ       TAU_YZ
  340    3.753e-010  2.531e+003   7.884e+003   3.848e+002  -3.902e+002  -1.210e+003
  348    1.321e-009  1.132e+003   3.602e+003   3.493e+002   1.105e+002  -1.186e+003
  356    6.072e-009  3.181e+001   4.527e+002   1.948e+002   2.821e+002  -1.470e+003
   :        :          :           :            :            :           :

C)RESULTS OF WELD THROAT REQUIREMENT
CALCULATIONS:
tb_0750.xls
Tue Mar 18 14:11:28 1997
Joint Normal (X, Y, Z): (0,0,1)
Thickness: 0.375
Welded Both Sides, Fillet
Allowable Stress:  13200
Min Weld Throat, t: 0.224 at node 340
Min Fillet Size, S: 0.317 at node 340
Weld Load Output:
-----------------
El/Nd          Normal_load          Bending_Load          Shear_load      Min Throat
                  (lb/in)            (in-lb/in)             (lb/in)          (in)
340              5145.75             136.828               716.354          0.224
348              3763.5              150.797               604.235          0.175
356              2880.51             169.421               628.098          0.145
 :                 :                   :                     :               :
```
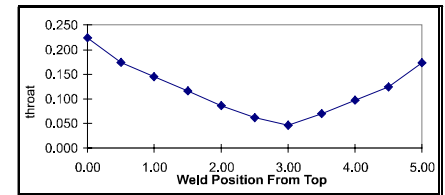
Figure 3-5   T-Bracket FEA Stresses and Loads at Joint

from the stresses. This is presented for node 340 in Figure 3-6 . Part C of Figure 3-5 presents the result of this calculation for every node in the weld joint.

3.5.   Weld Section Properties

The weld load calculations shown in Figure 3-6 give the local weld loads at each node as load per unit length. For calculating weld sizes from these results, the section properties per unit length of weld need to be calculated.

Resolution of Weld Loads, Node 340:

$t_b := \frac{3}{8} \cdot in$ Base Material Thickness

$\sigma_t := 19560 \; psi$ Normal Stress at Top of Joint

$\sigma_b := 7884 \; psi$ Normal Stress at Bottom of Joint

$\tau_{zx\_avg} := -390.2 \; psi$ Average Shear Stress in Joint

$\tau_{yz\_av} := \frac{-2530 \; psi \; -1210 \; psi}{2}$

$\tau_{avg} := \sqrt{\tau_{zx\_avg}^2 + \tau_{yz\_avg}^2}$ $\tau_{avg} = 1910 \; psi$

Joint Normal Load:

$P := \frac{\sigma_t + \sigma_b}{2} \cdot t_b$ $P = 5146 \; \frac{lbf}{in}$

Joint Bending Load:

$M := \left| \frac{\sigma_t - \sigma_b}{2} \right| \cdot \frac{t_b^2}{6}$ $M = 136.8 \; \frac{in \, lbf}{in}$

Joint Shear Load:

$V := \tau_{avg} t_b$ $V = 716.4 \; \frac{lbf}{in}$

Figure 3-6 Load Calculation for One Node

3.6. Weld Throat Stress Calculation

From the weld load components (Tj, Tws, Mw) and the weld section properties for a given weld size, the weld throat stress components can be determined as follows:

Stress due to normal load

$$f_{normal} = \frac{P}{A_w}$$

Stress due to bending:

$$f_{bending} = \frac{M}{S_w}$$

Stress due to shear:

$$f_{shear} = \frac{V}{A_w}$$

Total stress magnitude:

$$f_{weld} = \sqrt{\left(\left|f_{bending}\right| + \left|f_{normal}\right|\right)^2 + \left(f_{shear}\right)^2}$$

Double Sided Fillet Weld
Treated as a Line

Weld Throat Area: $A_w = 2 \cdot t_w$

Moment of Inertia: $I_w = \dfrac{t_w \cdot t_b^2}{2}$

Section Modulus: $S_w = t_w \cdot t_b$

Refer to Figure 3-3. Note in the total stress magnitude equation that the bending and normal stresses are combined so that their magnitudes are additive. This will always be the case on one side of the joint.

In order to compare this result to the classical result, a further calculation is required:

The loads calculated from FEA (P & V) must be divided by two since the weld is double sided and the classical calculation is per unit length of weld and not per unit length of weld joint which is the calculation for the FEA presented.

Figure 3-3 (Reproduced) Weld Load Diagram
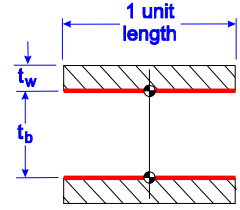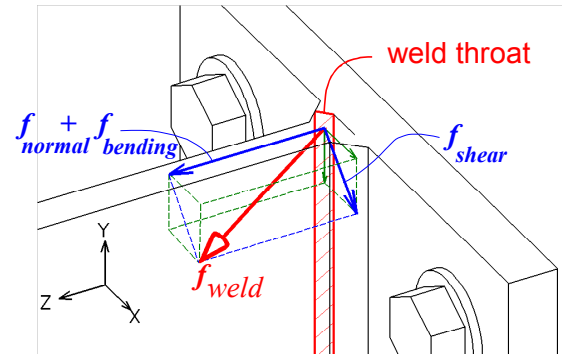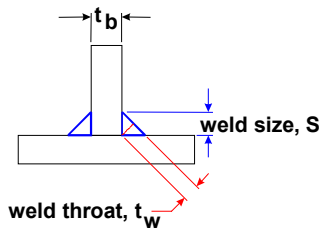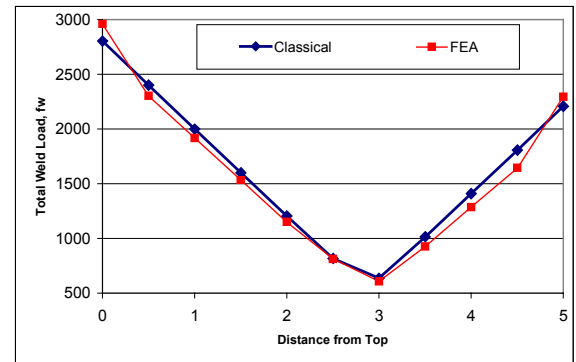
Bending Load About y, f_{by}:

$$f_{by} = \frac{M_{y-FEA}}{t_b}$$

$$= \frac{136.8 \frac{in-lb}{in}}{0.375\ in}$$

$$= 363.8\ lb/in$$

Total Weld Load, f_w:

$$f_w = \sqrt{\left(\frac{f_{normal}}{2} + f_{by}\right)^2 + \left(\frac{f_{shear}}{2}\right)^2}$$

$$= \sqrt{\left(\frac{5146 \frac{lb}{in}}{2} + 363.8 \frac{lb}{in}\right)^2 + \left(\frac{716.4 \frac{lb}{in}}{2}\right)^2}$$

$$= 2959\ lb/in$$

For evaluation of the weld size, the total stress magnitude is compared to the electrode shear allowable, F_a.

Figure 3-7 Comparison of FEA and Classical Load Calculations

3.7. Use of Elastic Stresses as Failure Resistance Predictors

Elastic Stresses are good predictors for high and medium cycle fatigue, loss of dimensional integrity, and fair proxies for ductile failure.

For resistance against ductile failure, structural plastic behavior comes into play, and can change the load proportionality significantly. This is particularly true with restrained single sided welds that are somewhat small and soft compared to the base metal. Often, in this situation, there will be local bending about the weld axis due to elastic rotation of the mated parts. If the base metal remains elastic while the weld is yielding, the strain on the weld due to the rotations will not increase nearly as fast as the strains due to the direct loads, and the joint failure will be aligned with what would be expected from the direct loads. However, if the base material becomes plastic, particularly with respect to rotations, the secondary bending loads should be considered. If this is an issue in your design, and you have non-linear FEA, a good method of obtaining an estimate of the plastic condition at the design ultimate load is to run your model with the predicted ultimate load of the structure, and perform a FEWeld analysis on that model (With non-linear analysis, FEWeld treats time steps the same as load cases).

The *Pipe_Lug* example problem in the next chapter is a good example of a geometry with secondary bending loads. Most common forms of classical analysis for ductile failure prediction (throat shear calculations) do not consider secondary bending of this sort.

The *'Direct_Shear'* formulation in FEWeld ignores the effects of local joint bending loads, which, when used for connections of compact sections, results in the same basic weld size calculation that is performed with classical analysis except that the direct loads are derived from the FEA load path instead of the nominal values used with classical analysis.

Note: The AWS D1.1 allowables for throat shear in fatigue (Category 'F') are also based on these nominal values, however, there is a major caveat: The geometries tested for the development of these allowables did not involve significant out-of-plane loading, and therefore were not subject to much local bending about the weld joint. For designs with out of plane loading and meaningful local elastic bending about the weld joint, use of effective notch stress concentration factors which are well described in [1][†] is recommended. FEWeld is well suited to perform nominal, geometric (Hot Spot), and effective notch stress calculations.

Final notes: With unrestrained geometries such as the T-Bracket of Figure 3-1, the bending load will remain proportional with the direct load, and the *'Direct_Shear'* formulation should not be used. If a single sided weld is to be used, the formulation should be one of the specific single sided formulations in the FEWeld library (F1_i, F1_o, P1, P1F1, etc).
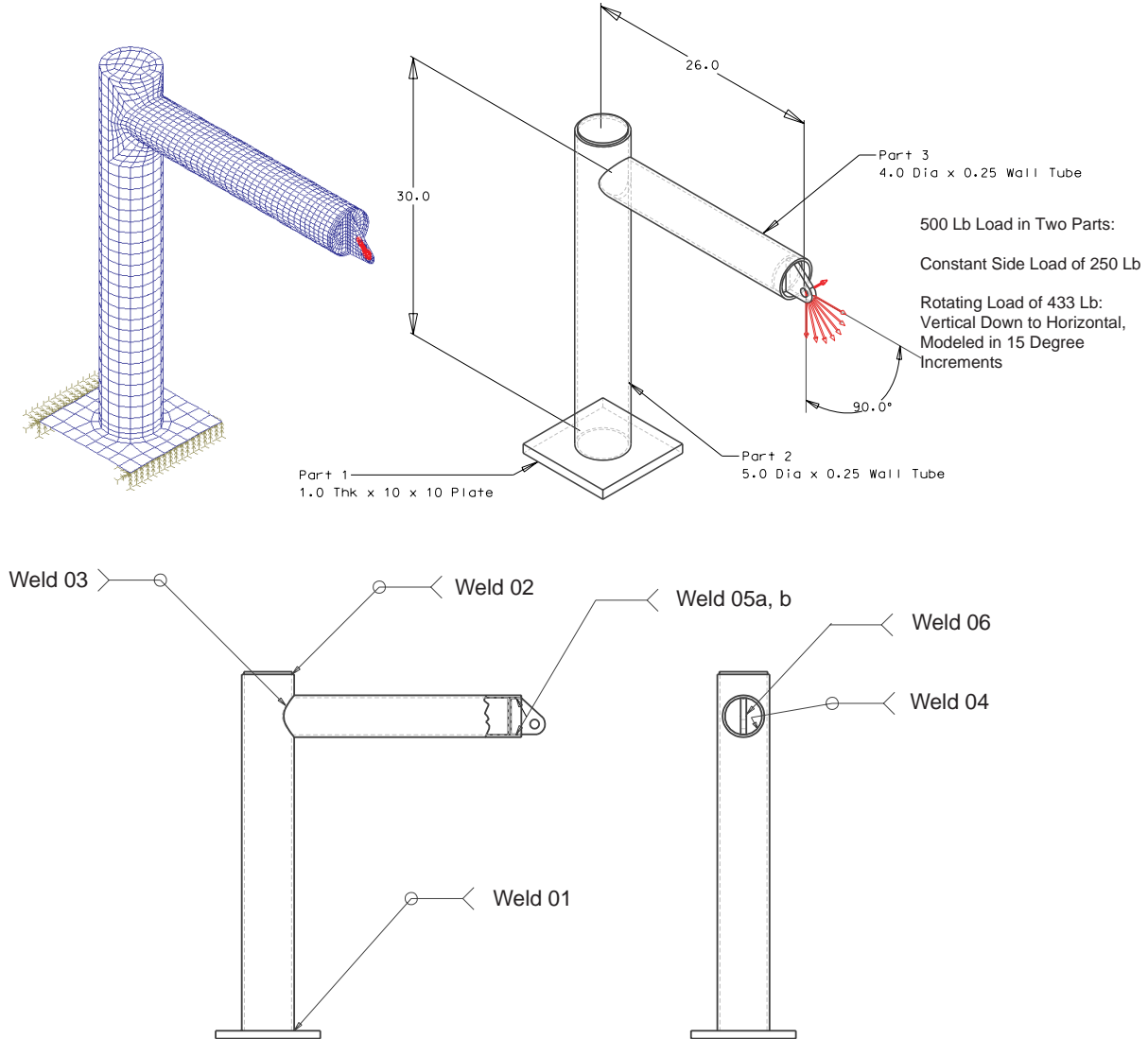

[†] [1] Hobbacher, A (Editor) 1996. *Fatigue Design of Welded Joints and Components, Recommendations of IIW Joint Working Group XII – XV.*   Woodhead Pub. Ltd.

Chapter 4.   Example Problems

4.1. Problem 1: Pipe_Lug

Estimated Time to Perform Exercise: 50 Minutes.

This exercise can be performed in the 'Demo' mode of FEWeld.

Figure 4-1 Basic Configuration of the Pipe_Lug problem

Figure 4-1 Depicts a lug that is supported by tubing that extends vertically up, then horizontally out. The lug experiences a load of varying direction from vertical down to horizontal, with a constant value of 433 pounds with an additional constant side load of 250 pounds for a total constant load magnitude of 500 pounds. The weldment is made of aluminum, with the tubing and lug details made from 6061 and welded with 4043 electrode. The base plate is made of 5083 and the weld of the vertical tube to the base plate is with 5356 electrode.

The throat shear stress design allowables against static failure for the electrodes are 5000 psi for the 4043 electrode and 7000 psi for the 5356 electrode. Welds 01 through 04 are welds of compact sections and will use the *'Direct_Shear'* formulation in FEWeld which ignores the local bending about the weld axis caused by the elastic rotation associated with the out-of-plane loading (Refer to Section 3.7). Welds 05 and 06 Will use the *'DF'* formulation (Balanced, Double Sided Fillet Weld).

Weld_No	Weld Type	CS	Description
01	Direct_Shear	1	Weld Between Base Plate and Post-Pipe
02	Direct_Shear	1	Weld Between Post-Pipe and Post-Pipe End Cap
03	Direct_Shear	3	Weld Between Post-Pipe and Stand-Off Pipe
04	Direct_Shear	3	Weld between Stand-Off Pipe and Lug-Plug
05a	F1_o	0	Weld between Stand-Off Pipe and Lug - Top
05b	F1_o	0	Weld between Stand-Off Pipe and Lug - Bottom
06	F1_o	0	Weld Between Lug-Plug and Lug

Load Case	Description
01	Lug-Load 0 Degrees (Vertical Down)
02	Lug-Load 15 Degrees
03	Lug-Load 30 Degrees
04	Lug-Load 45 Degrees
05	Lug-Load 60 Degrees
06	Lug-Load 75 Degrees
07	Lug-Load 90 Degrees (Horizontal)

Figure 4-2 Welds and Load Cases in the Pipe_Lug_01 Model

Exercise Objectives:

1)   Predict the fillet weld size requirements for the Pipe_Lug structure from the solved FEA model.

2)   Determine the effect on the weld sizes of changing the wall thickness of the Part 2 (vertical) tube from 0.25 inch to 0.50 inch.

Exercise Task Summary:

1)   Solve the Pipe_Lug_01 FEA Model that is provided on the CD or Download (5 Minutes)

2)   Create a new FEWeld file, Pipe_Lug.wld, and Build the FEWeld Weld Set for the Pipe_Lug_01 FEA Configuration, and run the FEWeld Analysis (20 Minutes)

3)   Review and Browse the Results (10 Minutes)

4)   Create a new FEA Model called Pipe_Lug_02 by copying the Pipe_Lug_01 model (or Reuse the Pipe_Lug_01 model), change the thickness of the vertical post to 0.50 and solve. (5 Minutes)

5)   Create a new Weld Set, Pipe_Lug_02, in the Pipe_Lug FEWeld file by copying the Pipe_Lug_01 Weld Set, and run the FEWeld Analysis. (1 Minute)

6)   Review and Browse the Results (10 Minutes).

4.1.1. **Task 1:** Solve the Pipe_Lug_01 FEA Model that is provided on the CD or Download. (5 Minutes)

The distribution (CD or Download) that contains the FEWeld Program also contains the Pipe_Lug Example problem in the *'FEWeld_Samples\Pipe_lug'* directory. This directory contains subdirectories for Ansys and Cosmos.

For Ansys, Create a new model called *'Pipe_Lug_01'* in the ansys directory. The *Pipe_Lug_01.db* is a built by issuing the *'CDREAD,COMB,Pipe_Lug_01,cdr'* command. Issue the *'LSSOLVE,1,7,1'*

For Cosmos, the directory contains *Pipe_Lug_01.gfm.* Create a new model in cosmos called *'Pipe_lug_01'* in the cosmos directory, and issue the following command from the Cosmos console: *'file,pipe_lug_01.gfm,0,1,0'.* This file will build and run the fea model.

Note: The model takes 3 minutes to solve in cosmos 2.5 using the sparse matrix direct solver on a 450 MHz Pentium II with 256 Meg of ram.

4.1.2. **Task 2:** Build and Run the FEWeld Model

Create a new FEWeld file, Pipe_Lug.wld, and Build the FEWeld Weld Set for the Pipe_Lug_01 FEA Configuration.

Step 1: Set the Default FEA System for FEWeld

Start FEWeld. Each FEWeld file has a FEA System setting. Now is a good time to set the default FEA system for new files.

Figure 4-3 FEA Default

Select the **File >> Options…** menu command. The Options dialog will appear as in Figure 4-3. Select your preferred FEA System. You can set the FEA system for individual files also, with the **Analysis >> Select FEA System** menu command when a FEWeld file is open. In addition to affecting the command file format, the fea system setting affects the input for defining the weld nodes and elements in the weld definition window.
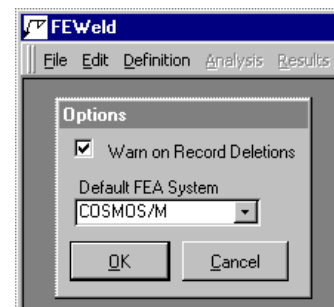
Step 2: Create a new FEWeld File

Select the **File >> New** menu command. Navigate to the *Sample_Problems\Pipe_Lug* Directory. Type in file name such as *Pipe_Lug* and hit OK. The new FEWeld file will be created and opened. The screen should look like Figure 4-4
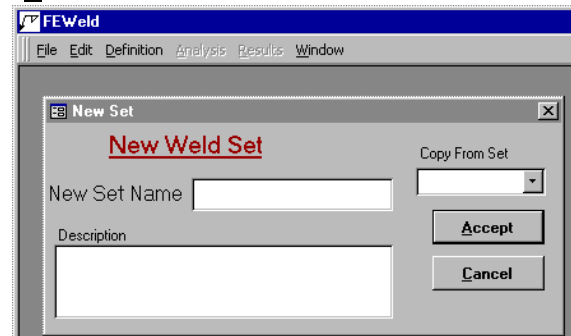
Figure 4-4 New FEWeld File

Step 3: Create a new Blank Weld Set

In the New Set Dialog of Figure 4-4, Type in *Pipe_Lug_01* for the Set Name and *"0.25 Thick Wall Post, Quadratic Displacement Formulation FEA Model"* for the description and click *Accept.*

The New Set Dialog will close, and the Weld_Set main window will appear as shown in Figure 4-5.

Step 4: Populate the Weld Set with Weld, Load Case, and Criteria Definitions

The Weld_Set main window is free-form in that there is no required order for putting in welds, load cases, and Stress Allowable Sets.

Figure 4-5 Empty Weld Set,
The 'Main Window'

Entering The Welds:

Since there are 7 welds (counting welds 05a and 05b separately), you may want to enlarge the welds section of the form. Do this by dragging the divider bar at the bottom of the weld list downward while holding down the mouse button (Item 1, Figure 4-5). Also experiment with resizing the window width and height.

Refer to Figure 4-2 and, using *'Direct_Shear'* formulation (for static analysis of connections of compact sections), type the Weld Number, Weld Type, and Description into Weld Grid section (Figure 4-5, Item 2) for the 7 welds[1].
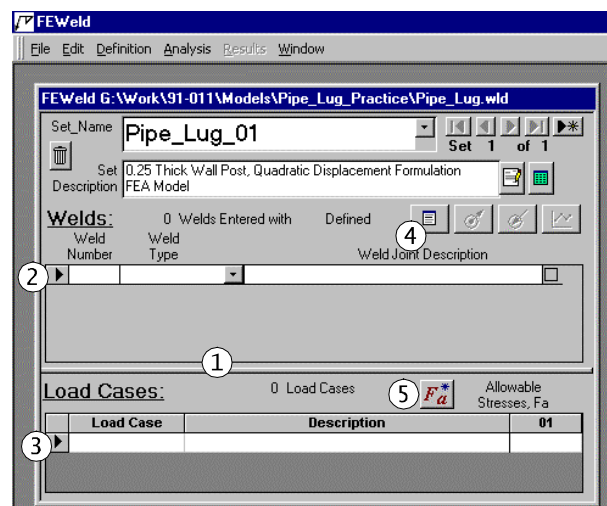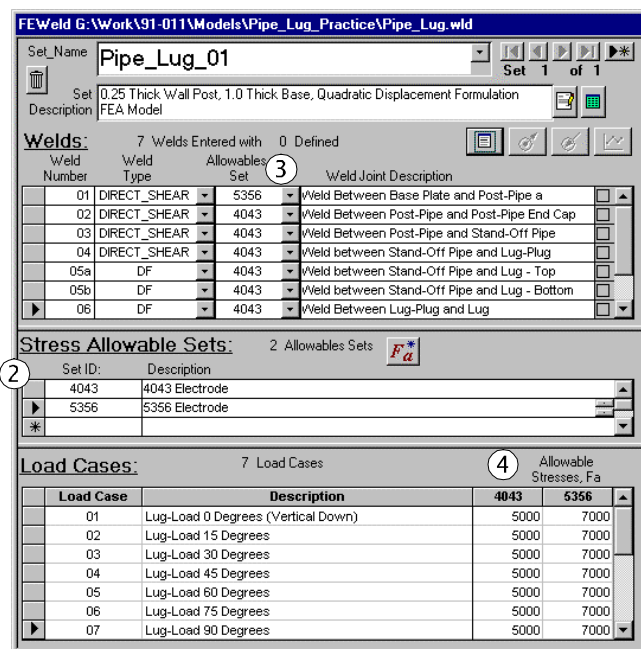
---

[1] Tip: After the first weld is entered, in each field of subsequent records, depressing <ctrl-"> (ditto) fills the current field with the value of the same field of the previous record.

Entering the Load Cases:

There are 7 Load Cases and two electrode materials with different stress allowables.   Since most problems involve a single electrode material, we will start with the allowables for 4043, and add the 5356 later. Refer to Figure 4-2. Into the Load Case Grid (Figure 4-5, Item 3) try entering the load case by columns instead of rows: Enter the load case numbers into the form, Load Cases 01 through 07, followed by the descriptions, then the stress allowable for 4043 (5000 psi). Depressing the return key moves to the next load case while hitting the tab key moves to the next field of the current load case. (Load case numbers must be entered as they are represented in the FEA system, however leading zero's are o.k., and advised since FEWeld sorts entries in alphabetic order).
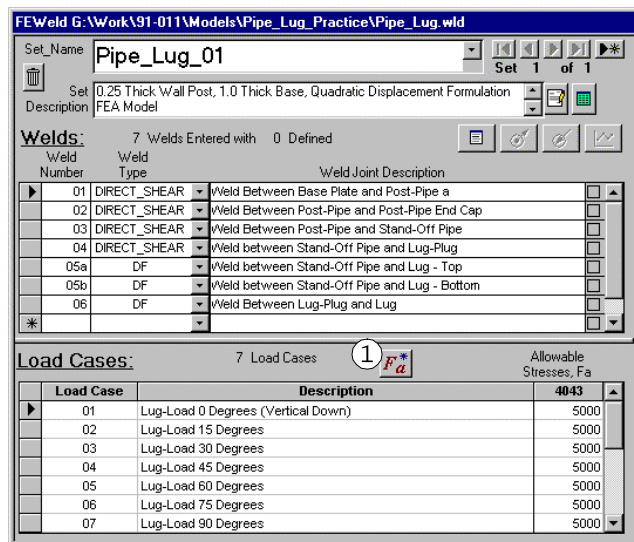
Part A: Before Stress Allowable Definition   **Part B:** After Stress Allowable Definition

Figure 4-6   Defining Stress Allowable Sets

Defining Multiple Stress Allowables for the Same Load Case:

Because this weldment will be fabricated with both 4043 and 5356 electrodes, different welds will have different allowable throat stresses for the same load cases. In order to accommodate this, 'Stress Allowable Sets' must be defined.

Click the F_a^* button in the main window (Figure 4-6, Part A Item 1) or select the **Definition >> New Allowables Set** menu command. The appearance of the main window will change to show the area for stress allowable sets as shown in Figure 4-6, Part B, Items 2, and 3. Note that when the Stress Allowables Grid is Opened, there is a stress allowable set called 01 already there. Rename that to 4043 and observe what happens in the allowables set part of the weld grid (Item 3), and in the Allowable Stresses part of the Load Case Grid (Item 4). Next, add a 5356 Allowables Set in the Stress Allowable Sets Grid (Item 2). Change the allowables set for weld 01 in the weld grid to 5356 (Item 3), and add the allowable stress values (7000 psi) in the 5356 column of the Load Cases grid (Item 4).

Step 5: Provide Weld Detail Data for Each Weld: Terminated Part Element Selection and Weld Joint Node Selection And Weld Parameters
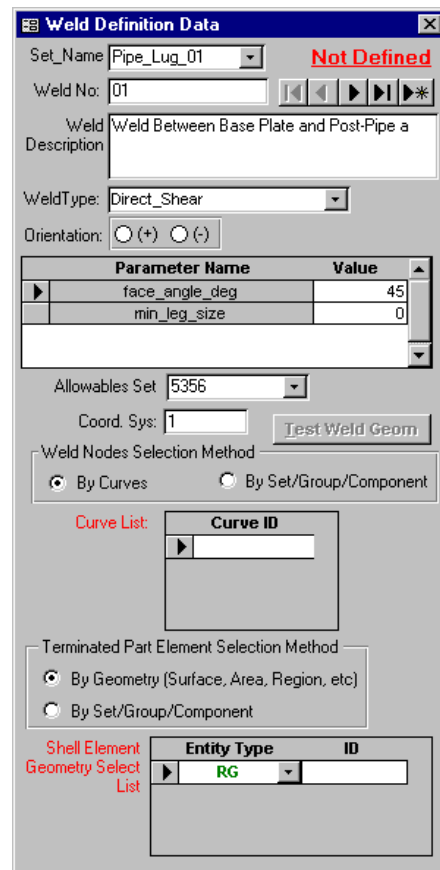
Open the weld definition window by clicking the 'weld definition' button, ⬛, in the weld set main window (Figure 4-5, Item 4), by selecting the **Definition>>Weld Detail** menu command, or by double clicking on a weld number or description in the weld list, Items 2 and 3. (Double clicking the weld type brings up the formulation editor for that weld type.)

Figure 4-7 Shows the weld definition data window. If it is not set to Weld No 01, either press the ⏮ button or click anywhere on the Weld 01 row in the Welds grid of the Weld Set main window. Notice that this weld is not 'defined'. A weld becomes defined when there is sufficient information provided to put out a valid FEA command file for that weld into the appropriate FEA system. For Weld 01, the Weld Node Selection Curve List and the Shell Element Geometry List are highlighted in red. This information is needed to complete the definition for weld 01.

Note, rather than defining the welds in the main window as we did, they can also be defined in this form.

We will go through the items below the weld type one by one.

Orientation: Many weld types require the element orientation with respect to where the weld will be laid – some balanced formulations, such as equal double sided fillet welds *'DF'*, and the *'Direct Shear'* don't care about the orientation, and the orientation is not required for those welds.

Figure 4-7 Weld Definition

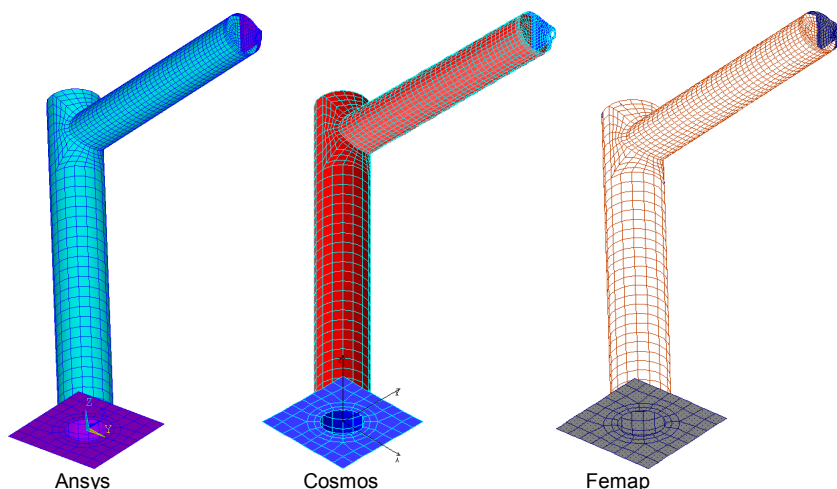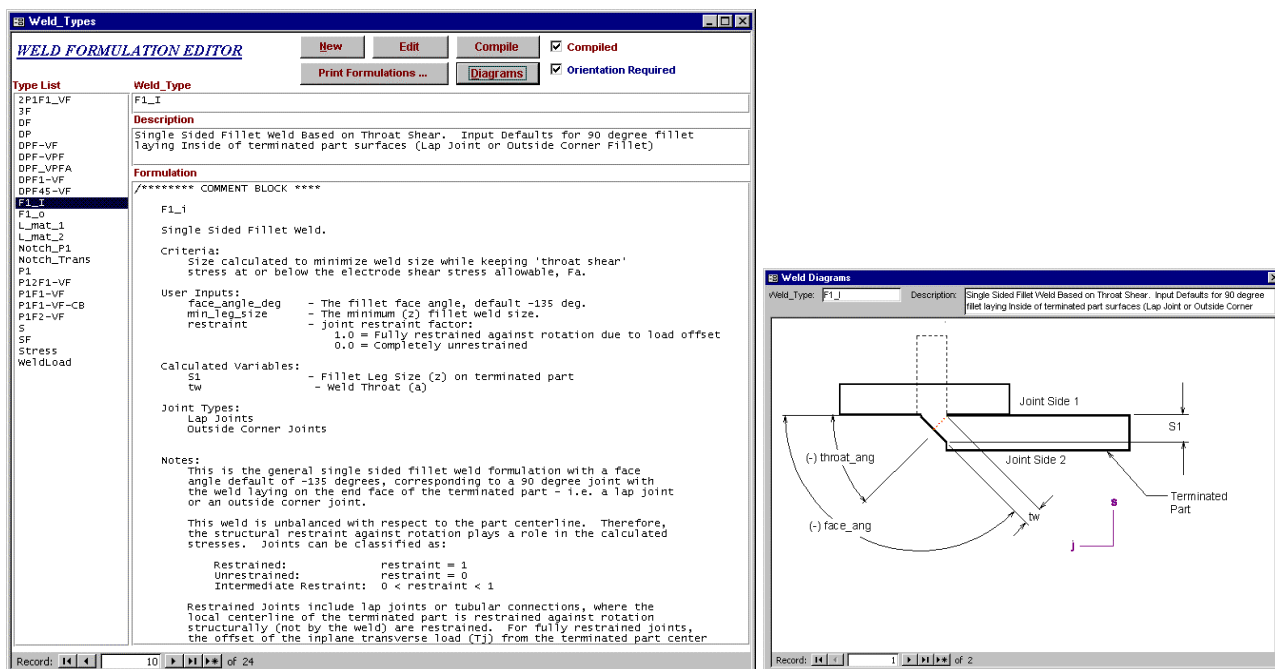

Ansys   Cosmos   Femap

Figure 4-8 Ansys and Cosmos Plots Colored to Show Element Orientation.

The Pipe_Lug_01 model was meshed with the positive element normals pointing outward and up. Figure 4-8 shows filled, colored element plots from Ansys and Cosmos with the element orientation identified by color. (If you are looking at this in black and white, the Cosmos plot is difficult to distinguish, the positive faces are colored red and the negative faces are colored blue.)

When you are curious about a formulation, double click on the weld_type field in the main window or in the Weld Definition Data window, or alternatively, issue the **Definition>>Library Formulations** menu command and select the formulation from the list on the left.

Figure 4-9 The Weld Formulation Editor and Diagram Windows showing the F1_i Formulation.

Weld Parameters:

The next item in the weld definition data form is the undefined weld parameters. The default values for the *Direct_Shear* and *DF* formulations are a 45 degree fillet and no minimum size (0.).

Coordinate System: (Optional):

The coordinate system is an optional parameter. It is used for weld joint nodal position listings only. If none is specified, then the global Cartesian system is used (0). For the pipe lug problem, Welds 01 and 02 should use Csys 1 (the global cylindrical system), Welds 03 and 04 should use Csys 3, a cylindrical system defined in alignment with the stand-0ff pipe, and welds 05 and 06 should use Csys 0.

Weld Joint Definition – Nodes and Elements:

There are several approached for specifying the weld joint nodes and terminated part elements. In Ansys, it may be expedient to build components or groups and put the nodes and elements in them for each weld and just specify the groups. Or, the curves (lines) for the weld joint nodes, and the surfaces, areas, etc, for the terminated part elements could be listed a priori and filled into the form. The method presented here will be to fill in the form interactively, querying the FEA entities and putting them into FEWeld one weld at a time.

Open the Pipe_Lug_01 FEA model in your FEA application. Switch to the FEWeld application after the Pipe_Lug_01 fea model is opened. The Weld Definition Data window can float outside the FEWeld application window. Make the FEWeld application window small as shown in Figure 4-10. (Size it small, do not minimize it, minimizing the FEWeld application window will remove the weld definition data window.)
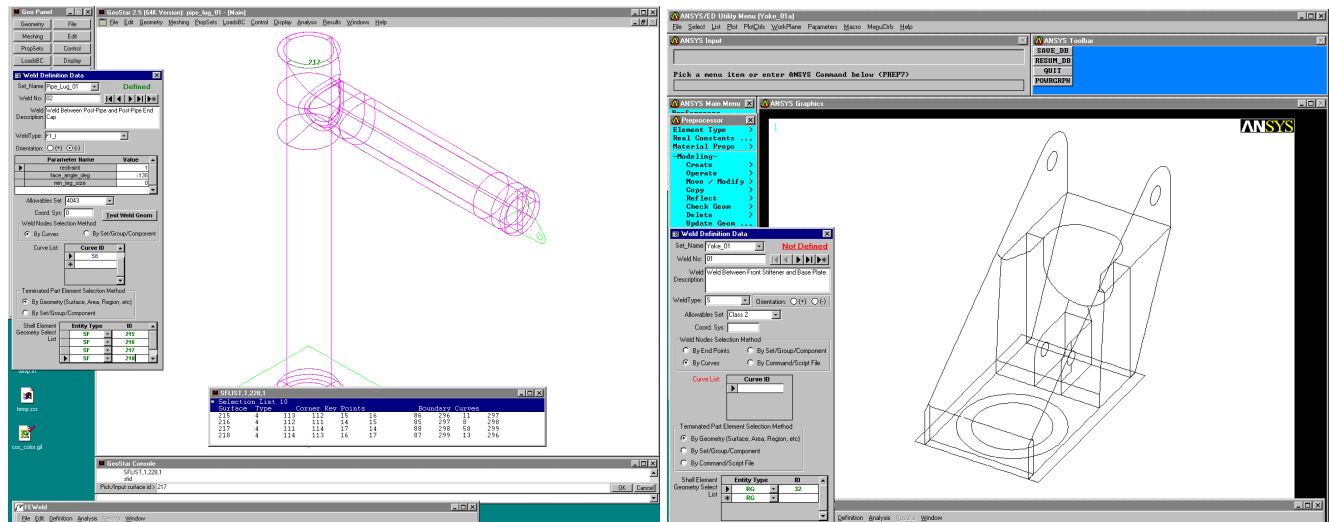
Figure 4-10 Interactive Weld Definition

With the screen setup similar to Figure 4-10, interactively entering the weld definitions is fairly simple. Table 4-1 and Table 4-2 give useful commands for Ansys, Cosmos, respectively, for querying the FEA model for Point, Curve, Surface, Area, etc. ID's.

Entity	Command Line	Menu	Description
Points	KLIST,P…	List>Picked Entities>Keypoints…	Pick Points and List
	KSEL,S,P…	Select>Entities…->Keypoints	Select Points and List
	KLIST,ALL	List>Keypoints…	
Lines	LLIST,P…	List>Picked Entities>Lines…	Pick Lines and List
	LSEL,S,P…	Select>Entities…->Lines	Select Lines and List
	LLIST,ALL	List>Lines…	
Areas	ALIST,P…	List>Picked Entities>Areas…	Pick Areas and List
	ASEL,S,P…	Select>Entities…->Areas	Select Areas and List
	ALIST,ALL	List>Areas…	

Table 4-1 Ansys Useful Query Commands

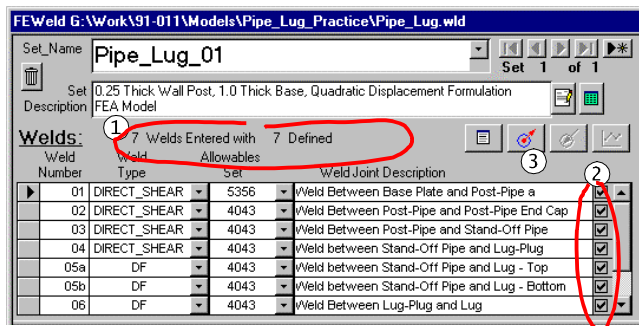| Entity	Command Line	Menu	Description
Points	PTID…	Geometry>Points>Editing>Identy	Query Point
	INITSEL,PT;	Control>Select>Initialize->Points	Select Points and List
	SELPIC,PT…	Control>Select>By Picking …	
	PTLIST;	Geometry>Points>Editing>List	
Curves	CRID…	Geometry>Curves>Editing>Identy	Query Curve
	INITSEL,CR;	Control>Select>Initialize->Curves	Select Curves and List
	SELPIC,CR…	Control>Select>By Picking …	
	CRLIST;	Geometry>Curves>Editing>List	
Suraces	SFID…	Geometry>Surfaces>Editing>Identy	Query Surface
	INITSEL,SF;	Control>Select>Initialize->Surfaces	Select Surfaces and List
	SELPIC,SF…	Control>Select>By Picking …	
	SFLIST;	Geometry>Surfaces>Editing>List	
Regions	RGID…	Geometry>Regions>Editing>Identy	Query Region
	INITSEL,RG;	Control>Select>Initialize->Regions	Select Regions and List
	SELPIC,RG…	Control>Select>By Picking …	
	RGLIST;	Geometry>Regions>Editing>List	

Table 4-2 Cosmos Useful Query Commands

The set of entities for the weld definitions is listed in Table 4-3. Move between welds in the Weld Definition window with the right and left arrow buttons at the top right, ◄ ►.

When entering data in the curve list or surface list, pressing **<ctrl-tab>** will move you out of the list without the mouse.

As the Welds become fully defined, their status is updated in the main window. See Figure 4-11, Items 1 and 2.

Figure 4-11 Weld Definition Status in the Main Window

Step 6: Generate the FEWeld-FEA Command/Macro File

When the FEWeld definition for the Pipe_Lug_01 weld set is complete, FEWeld is ready to write out the FEA Command file. Note that the '*Write Command File*' button in the main window has become enabled (Figure 4-11, Item 3).

The write_fea_cmd_File command is invoked either by it's button, item 3 in Figure 4-11, or with the **Analysis>>Write FEA Cmd File** menu command. A save-file dialog comes up and the default name for the command file is *set_name_feweld.cmd* where *.cmd* is *.mac* for Ansys and *.geo* for Cosmos, and *set_name* is the name of the current weld set. For example, the command file name for Cosmos output for Pipe_Lug_01 is *Pipe_Lug_01_FEWeld.geo.*

It is recommended to put the command file in the directory that contains the FEA model because that is where FEWeld looks for the FEWeld results file that is generated by the command file. The relative path from the FEWeld file to the location where the command file is written is stored with each weld set, so that FEWeld will look in the correct location if the job directory is moved.

Write the FEA Command File now.

Weld	Ansys		Cosmos	
No	Lines	Areas	Curves	Surfaces
01	9	2	70	SF 1
	10	3	75	SF 2
	11	4	79	SF 3
	12	5	82	SF 4
02	46	20	8	SF 215
	47		11	SF 216
	48		13	SF 217
	49		58	SF 218
03	33	11	36	SF 87
	34	13	43	SF 89
	36	15	151	SF 124
	60	17	153	SF 125
	61		197	SF 147
	71		200	SF 148
	72		244	SF 170
	73		247	SF 171
04	37	18	45	SF 17
	38	21	46	SF 33
	39		141	SF 118
	40		144	SF 119
			182	SF 141
			184	SF 164
			230	SF 165
			231	SF 201
05a	92	19	15	SF 25
05b	94	19	16	SF 26
06	55	19	25	SF 25
			26	SF 26
			27	SF 27
			31	SF 28

Table 4-3 Pipe_Lug_01 Weld Geometric Entities

Step 7: Execute the Command/Macro File in the FEA Environment

If your FEA program is not open in the Pipe_Lug_01 model, open it now. Executing the command file is very simple. See Table 4-4.

FEA Program	Command Line	Menu Command
Ansys	/INPUT,FNAME,EXT,DIR	File>Read Input From…
Cosmos	FILE[1]	File>Load…

Table 4-4   Commands to execute FEA command/macro/script files

Execute the *Yoke_01_FEWeld.xxx* FEA Command file now. Depending on the speed of your computer, the script should take 1 to 5 minutes to run.
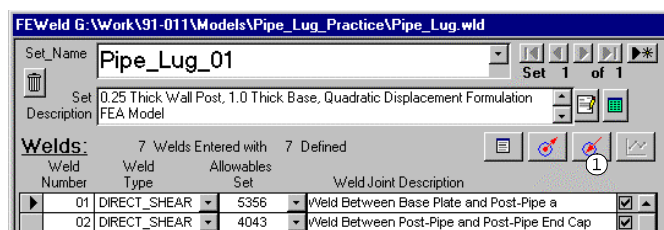
The FEA Command File created a file called *Yoke_01.fewinp* in the FEA problem directory. This will be read into FEWeld.

Step 8 Read the Weld Results into FEWeld

Read the Weld Results into the FEWeld file by clicking the Read Results Button, Item 1 in Figure 4-12 or with the **Analysis>>Read FEA Results** menu command.

Figure 4-12

A standard windows file read dialog will appear as shown in Figure 4-13. When this file is opened, FEWeld will begin calculating the weld loads and weld formulations. A progress dialog will display the calculation status. For a large model, this can take a little while, although generally, it is much faster than the original FEA
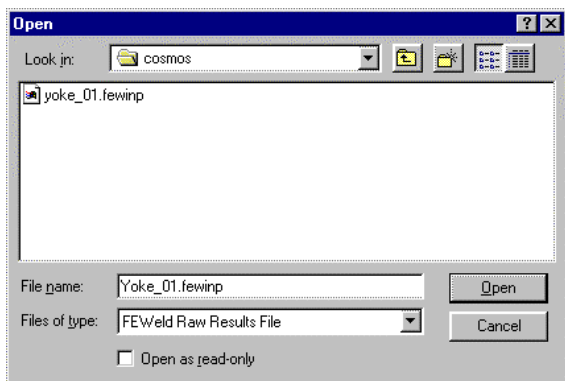
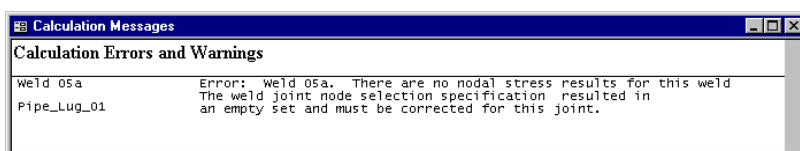 **Figure 4-13**
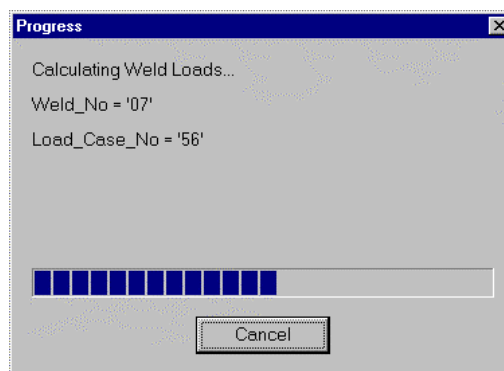
Figure 4-14

solution.

Troubleshooting: If there was a gross error in the weld definition such that there is no association between the weld nodes and elements, the calculation will stop with an error message as shown Figure 4-15. FEWeld attempts to catch errors and give descriptions. Some descriptions are more helpful than others.

For example, say that you entered curve no 16 instead of 15 for Weld No. 05a.

Figure 4-15

---

[1] For Cosmos, just typing the file command and pressing return will bring up the standard windows file read dialog box to browse and pick the file. Or, type in the filename: FILE, Pipe_Lug_01_FEWeld.geo;

To identify the error, open the weld definition data window for weld 05a. Either manually query the curves and areas/regions/surfaces or write out a test weld FEA command file, by clicking the 'Test Weld Geom' button, Item 1 in Figure 4-16.

The 'Test Weld Geom' command will write out a file titled *Pipe_Lug_01_W05a_FEWeld.cmd* where .*cmd* is .*mac* for Ansys and .*geo* for Cosmos. Executing this file in the FEA program should select and display the Geometry (Curves/Lines, Areas, Surfaces, Regions, etc) selected for the weld and their associated Terminated Part Elements and Weld Joint Nodes. Repair the definition and return to Step 6: Generate the FEWeld-FEA Command/Macro File.
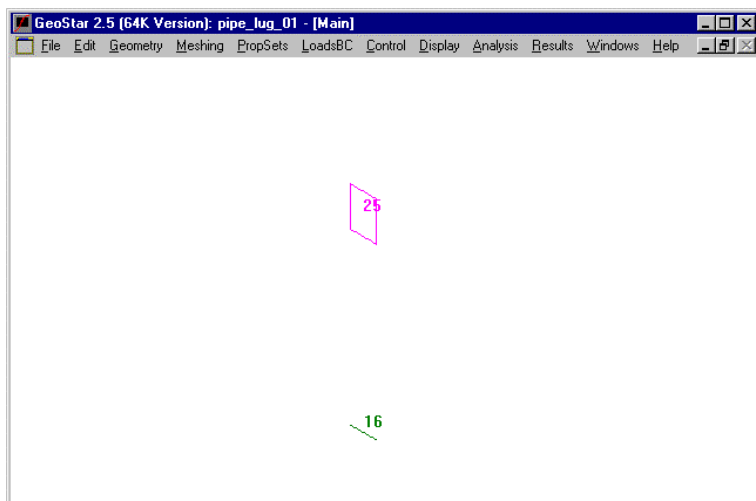
Figure 4-16

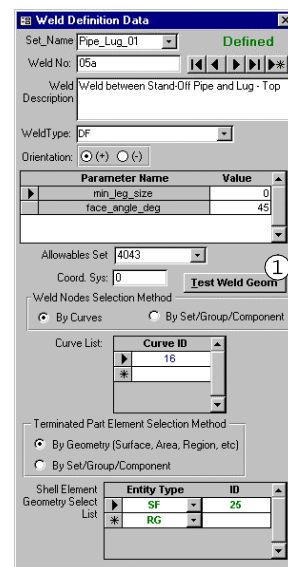**Figure 4-18** Result of running *Pipe_Lug_01_W05a_FEWeld.geo* in Cosmos

Step 9 Review the Results for Joint Definition Correctness.

Figure 4-17

Open the Detail Results Charting Window by clicking the chart button in the Weld Set main Window, Figure 4-17, Item 1, that becomes enabled after results for the Weld Set are successfully read in. or by selecting the **Results>>Detail Results** menu command.

Configure the chart control panel so that it sequences through the welds first, as arranged in Figure 4-19. Do this by clicking the load case toggle, Figure 4-20, Item 1, and moving the load case position as shown below.
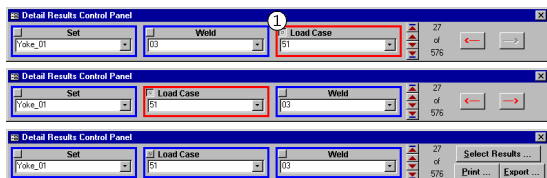

Figure 4-20 Rearranging the Chart Sequence

Now, scroll through the welds looking at the length and the coordinates to see if they make sense. Figure 4-19 shows the plot for Weld No 01, except that curve number 81 was used in the definition instead of curve 82. This weld definition would need correction.
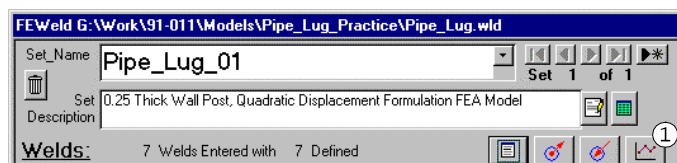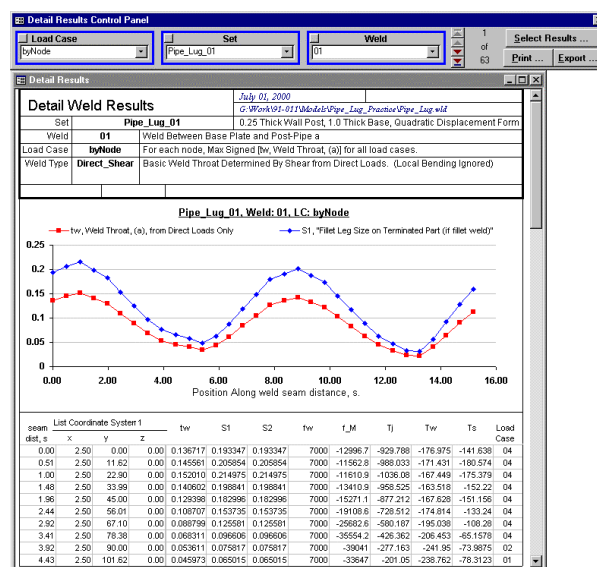

Figure 4-19

4.1.3. **Task 3:** Review and Browse the Results (10 Minutes)

There are 2 basic levels of results review: Summary reports and detail charts.

Summary Reports

There two summary reports available in FEWeld: ***Results Summary by Weld*** and ***Results Summary by Load Case.*** They are both accessed under the <u>**Results**</u> menu.

Results Summary by Weld for Set: Pipe_Lug_01

File: G:\Work\91-011\Models\Pipe_Lug_Practice\Pipe_Lug.wld	July 01, 2000	Page 1 of 2

Set Description
0.25 Thick Wall Post, 1.0 Thick Base, Quadratic Displacement Formulation FEA Model

Load Case Data: Load Case Count 7

LC	Description
01	Lug-Load 0 Degrees (Vertical Down)
02	Lug-Load 15 Degrees
03	Lug-Load 30 Degrees
04	Lug-Load 45 Degrees
05	Lug-Load 60 Degrees
06	Lug-Load 75 Degrees
07	Lug-Load 90 Degrees

Allowables Sets: Allowables Sets Count 2

ID	Description
4043	4043 Electrode
5356	5356 Electrode

Load Case - Allowables Crosstab:

Load Case	4043	5356
01	5000	7000
02	5000	7000
03	5000	7000
04	5000	7000
05	5000	7000
06	5000	7000
07	5000	7000

Weld Data: Weld_Count 7 entered with 7 Calculated

		Allowable Stress	Node	Load Case	Criteria	Value
Weld 01	Weld Between Base Plate and Post-Pipe a					
DIRECT_SHEAR	Basic Weld Throat Determined By Shear from Direct Loads. (Local Bending Ignored)				Sa_Set_No: 5356	
	face_angle_deg 45					
	min_leg_size 0					
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	7000	5633	04	Max Signed	.215
tw	Weld Throat, (a), from Direct Loads Only	7000	5633	04	Max Signed	.152
Weld 02	Weld Between Post-Pipe and Post-Pipe End Cap					
DIRECT_SHEAR	Basic Weld Throat Determined By Shear from Direct Loads. (Local Bending Ignored)				Sa_Set_No: 4043	
	face_angle_deg 45					
	min_leg_size 0					
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7149	01	Max Signed	.066
tw	Weld Throat, (a), from Direct Loads Only	5000	7149	01	Max Signed	.046
Weld 03	Weld Between Post-Pipe and Stand-Off Pipe					
DIRECT_SHEAR	Basic Weld Throat Determined By Shear from Direct Loads. (Local Bending Ignored)				Sa_Set_No: 4043	
	face_angle_deg 45					
	min_leg_size 0					
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	8785	01	Max Signed	.337
tw	Weld Throat, (a), from Direct Loads Only	5000	8785	01	Max Signed	.239
Weld 04	Weld between Stand-Off Pipe and Lug-Plug					
DIRECT_SHEAR	Basic Weld Throat Determined By Shear from Direct Loads. (Local Bending Ignored)				Sa_Set_No: 4043	
	face_angle_deg 45					
	min_leg_size 0					
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	11328	07	Max Signed	.109
tw	Weld Throat, (a), from Direct Loads Only	5000	11328	07	Max Signed	.077

Results Summary by LC for Set: Pipe_Lug_01

File: G:\Work\91-011\Models\Pipe_Lug_Practice\Pipe_Lug.wld	July 01, 2000	Page 2 of 4

Weld Data: Weld_Count 7 entered with 7 Calculated

|		Allowable Stress	Node	Criteria	Value
Weld 02					
Weld Between Post-Pipe and Post-Pipe End Cap					
DIRECT_SHEAR	Basic Weld Throat Determined By Shear from Direct Loads. (Local Bending Ignored)			Sa_Set_No: 4043	
	face_angle_deg 45				
	min_leg_size 0				
Load Case: 01	Lug-Load 0 Degrees (Vertical Down)				
→ S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7149	Max Signed	.066
→ tw	Weld Throat, (a), from Direct Loads Only	5000	7149	Max Signed	.046
Load Case: 02	Lug-Load 15 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7149	Max Signed	.064
tw	Weld Throat, (a), from Direct Loads Only	5000	7149	Max Signed	.045
Load Case: 03	Lug-Load 30 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7149	Max Signed	.06
tw	Weld Throat, (a), from Direct Loads Only	5000	7149	Max Signed	.042
Load Case: 04	Lug-Load 45 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7149	Max Signed	.053
tw	Weld Throat, (a), from Direct Loads Only	5000	7149	Max Signed	.038
Load Case: 05	Lug-Load 60 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7149	Max Signed	.045
tw	Weld Throat, (a), from Direct Loads Only	5000	7149	Max Signed	.032
Load Case: 06	Lug-Load 75 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7148	Max Signed	.037
tw	Weld Throat, (a), from Direct Loads Only	5000	7148	Max Signed	.026
Load Case: 07	Lug-Load 90 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	7147	Max Signed	.033
tw	Weld Throat, (a), from Direct Loads Only	5000	7147	Max Signed	.024
Weld 03					
Weld Between Post-Pipe and Stand-Off Pipe					
DIRECT_SHEAR	Basic Weld Throat Determined By Shear from Direct Loads. (Local Bending Ignored)			Sa_Set_No: 4043	
	face_angle_deg 45				
	min_leg_size 0				
Load Case: 01	Lug-Load 0 Degrees (Vertical Down)				
→ S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	8785	Max Signed	.337
→ tw	Weld Throat, (a), from Direct Loads Only	5000	8785	Max Signed	.239
Load Case: 02	Lug-Load 15 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	8785	Max Signed	.326
tw	Weld Throat, (a), from Direct Loads Only	5000	8785	Max Signed	.231
Load Case: 03	Lug-Load 30 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	8785	Max Signed	.308
tw	Weld Throat, (a), from Direct Loads Only	5000	8785	Max Signed	.218
Load Case: 04	Lug-Load 45 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	8785	Max Signed	.284
tw	Weld Throat, (a), from Direct Loads Only	5000	8785	Max Signed	.201
Load Case: 05	Lug-Load 60 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	10123	Max Signed	.268
tw	Weld Throat, (a), from Direct Loads Only	5000	10123	Max Signed	.19
Load Case: 06	Lug-Load 75 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	10122	Max Signed	.269
tw	Weld Throat, (a), from Direct Loads Only	5000	10122	Max Signed	.19
Load Case: 07	Lug-Load 90 Degrees				
S1	"Fillet Leg Size on Terminated Part (if fillet weld)"	5000	10122	Max Signed	.276
tw	Weld Throat, (a), from Direct Loads Only	5000	10122	Max Signed	.195

Figure 4-21 *Results by Weld* and *Results By Load Case* Reports

In Figure 4-21, the report on the left is the *Results By Weld* report. It contains the most distilled summary of the analysis. It presents only the worst case single value of each summarized variable for each weld, along with the node, load case, and criteria (allowable stress).

The report on the right is the *Results By Load Case* report. This report shows for each load case the worst case single value of each summarized variable along with the node, and criteria.

Detail Charts

The detail chart will plot stored variables from the weld formulation along the weld joint.
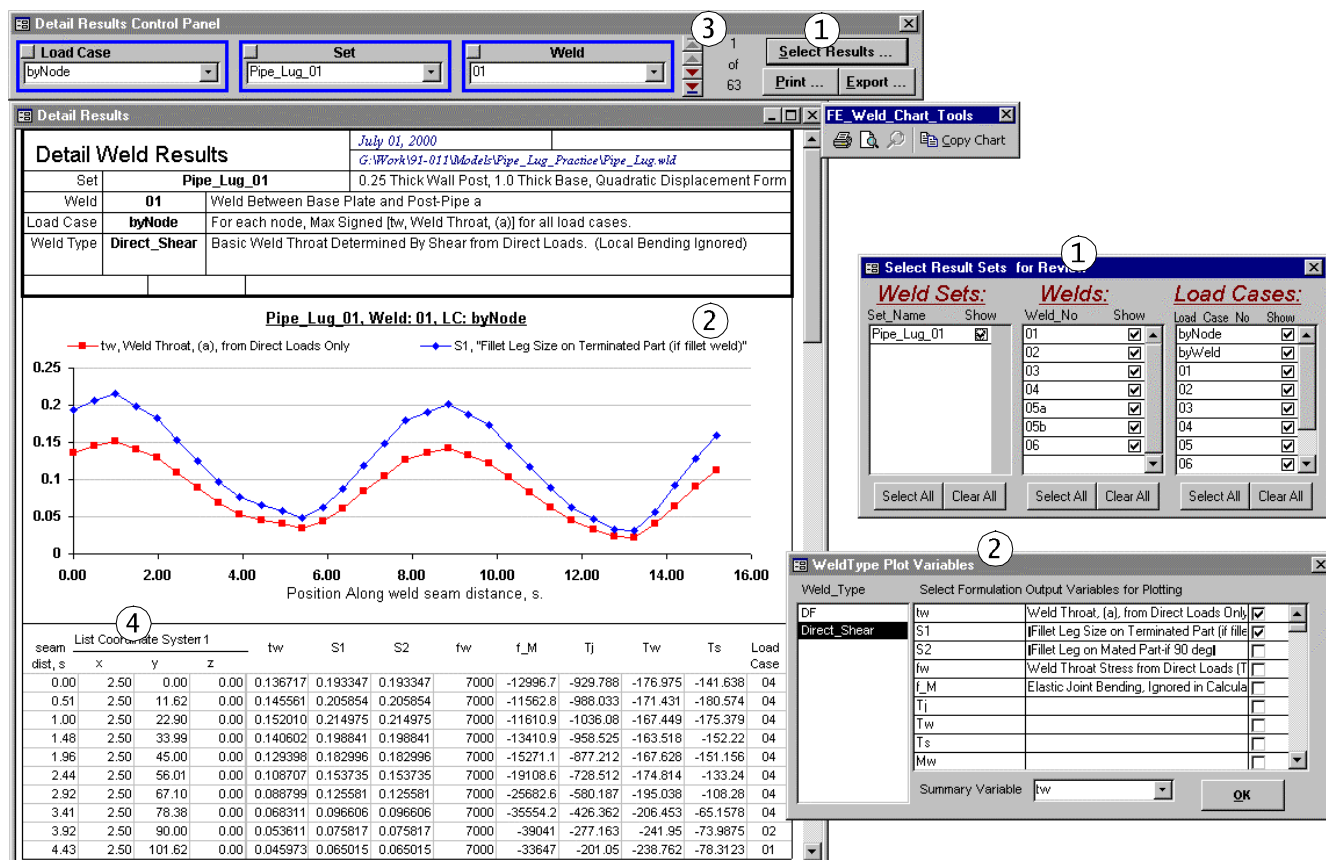

Figure 4-22 The Detail Weld Results Tools

Some items about the detail chart to briefly note:

Item 1: The Select Results button brings up a dialog to select Weld Sets, Welds, and Load Cases the the *Detail Results Control Panel* will Browse through. Note two special entries in the load case column: ByNode and ByWeld. ByNode plots at each node the value for the worst case load case of the summarized variable (Note the 'Summary Variable' in item 2). ByWeld plots the results for the single load case that resulted in the worst single value of the summarized variable. Note also in the *Control Panel*, the buttons for Print and Export. These are for batch printing charts, and exporting results for specific charts.

Item 2: The chart. Clicking on the chart area will open the *Weldtype Plot Variables* window for selecting which variables get plotted and which summary variable is used for the byNode and byWeld plots.

Item 3: The scrolling buttons in the Control Panel. Browse through the welds and load cases.

Item 4: Click the mouse on s, x, y, or z, and that will become the independent variable in the chart.
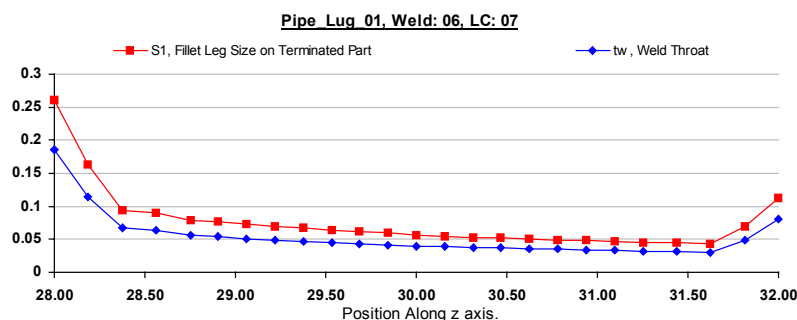
Note Weld No 06. The weld size is a little small, let's try a single sided weld.

Switch over to the Main Window (Using the Window menu or closing the chart window if necessary). Change the weld type for Weld No 06 to F1_o. Notice that if the chart was open, it is immediately closed, and the defined status of Weld No 06 is now 'Undefined' (The weld is un-checked).

Open the Weld Definition Data Window, ▣, (refer back to Task 2, Step 5), and set the restraint for Weld No 06 to 1 (In the upper and lower ½ inch, the joint will have a fair degree of restraint (against rotation) due to the adjacent, perpendicular Welds No 05).

Now, Open the detail chart window, ⌇. This will cause recalculation dialog to be opened. Unselecting any items in the calculation list will skip calculation for that item and that weld will remain in the recalculation list and be unavailable for post processing until it is recalculated in the future. Leave Weld 06 selected and click the *Go* button. Weld No 06 will be recalculated with a restraint factor of 1, and the chart will be displayed.

Weld_Recalc_List

The Following Welds are not calculated for their current Formulations, Parameters, and/or Stress Allowables:

Selected Welds will be recalculated and un-selected welds will not have formulation results available.

Select All	Clear All	Go

Set_Name	Weld_No	Select
Pipe_Lug_01	06	☑

Pipe_Lug_01, Weld: 06, LC: byNode

── tw , Weld Throat, (a)   ──◆── S1, "Fillet Leg Size on Terminated Part"

Position Along weld seam distance, s.

Figure 4-23

The resulting byNode chart for Weld 06 is shown in Figure 4-23. The required single sided weld size is quite large. Without closing the chart, change the weld type back to *'DF'* in the Weld Set main window or the weld detail window. Weld 06 will be immediately recalculated.

4.1.4. **Task 4:** Run the Pipe_Lug FEA Model with the Vertical Post Wall Thickness Changed to 0.50 (5 Minutes)

Either copy the Pipe_Lug_01 FEA model to Pipe_Lug_02 and revise property set 2 to 0.50 thickness or just revise the existing model and resolve the problem.

4.1.5. **Task 5:** Create a new FEWeld Weld Set in the Pipe_Lug file and Run the Analysis
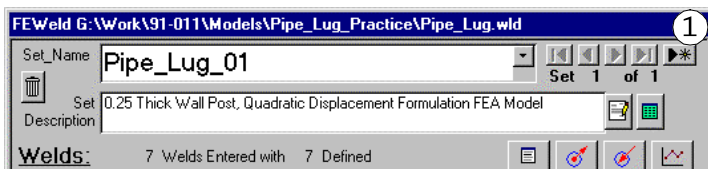
Figure 4-24

Close the Result Windows. In the Weld Set Main Window, Click the *New Weld Set Button*, ▶⁕, Figure 4-24, Item 1, or Select the **Definition>>New Set Menu** Command. The *New Set* window will appear as shown below in Figure 4-25.
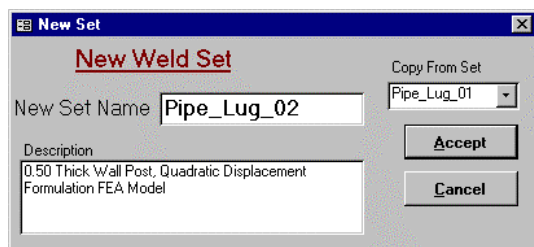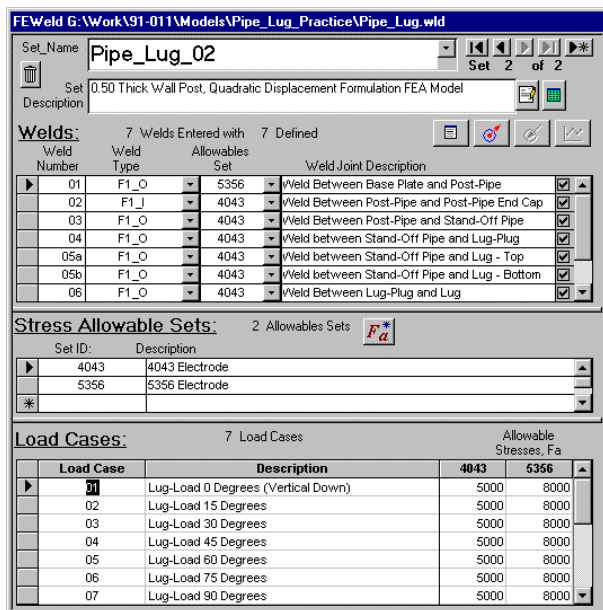
Figure 4-25

Make the new set name *'Pipe_Lug_02'*, and select Pipe_Lug_01 from the *Copy From Set* drop down list. Revise the description to note the new material thickness, and click accept.

The new weld set, Pipe_Lug_02 is created that is an exact copy of Pipe_Lug_01, except there are no results. No revision is necessary since the weld and load case definitions are remaining the same. Repeat task 2, Steps 6 through 8 to write out the FEA Command file, execute it, and load the results back into FEWeld.

The welds that are interesting with this wall thickness change are Welds No 01 through 03.

Weld_No	First Weld Description	Weld_Type for First Set	Var Name	Pipe_Lug_01	Pipe_Lug_02
01	Weld Between Base Plate and Post-Pipe a	DIRECT_SHEAR	S1	.215	.216
02	Weld Between Post-Pipe and Post-Pipe End Cap	DIRECT_SHEAR	S1	.066	.047
03	Weld Between Post-Pipe and Stand-Off Pipe	DIRECT_SHEAR	S1	.337	.234
04	Weld between Stand-Off Pipe and Lug-Plug	DIRECT_SHEAR	S1	.109	.109
05a	Weld between Stand-Off Pipe and Lug - Top	DF	S1	.171	.171
05b	Weld between Stand-Off Pipe and Lug - Bottom	DF	S1	.376	.376
06	Weld Between Lug-Plug and Lug	DF	S1	.262	.262

Table 4-5 Summary of Results ByWeld Across the Weld Sets

Table 4-5 Shows a summary of the results. This was generated with the **Results>>Results Sum ByWeld / Set Crosstab** menu command.
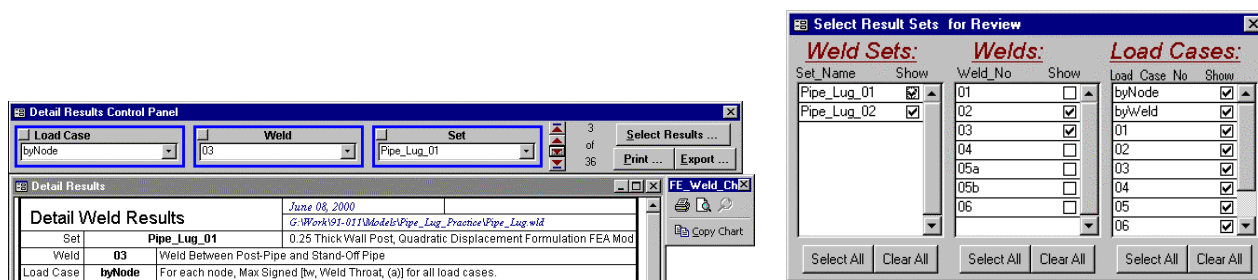
Figure 4-26 Selecting Specific Welds for Review

Open the Detail Weld Results (Chart) Window, and click the *Select Results...* button to bring up the Results Selection Window as shown in Figure 4-26. Clear all of the weld selections then select welds 02 and 03. Next, reorder the positions in the *'Detail Results Control Panel'* so that load cases are leftmost, welds in the middle, and sets at the right (See Figure 4-20, Step 9, Task 2).

Now the control panel will sequence across the weld sets first (corresponding to the FEA models), followed by the welds, and last, the load cases.
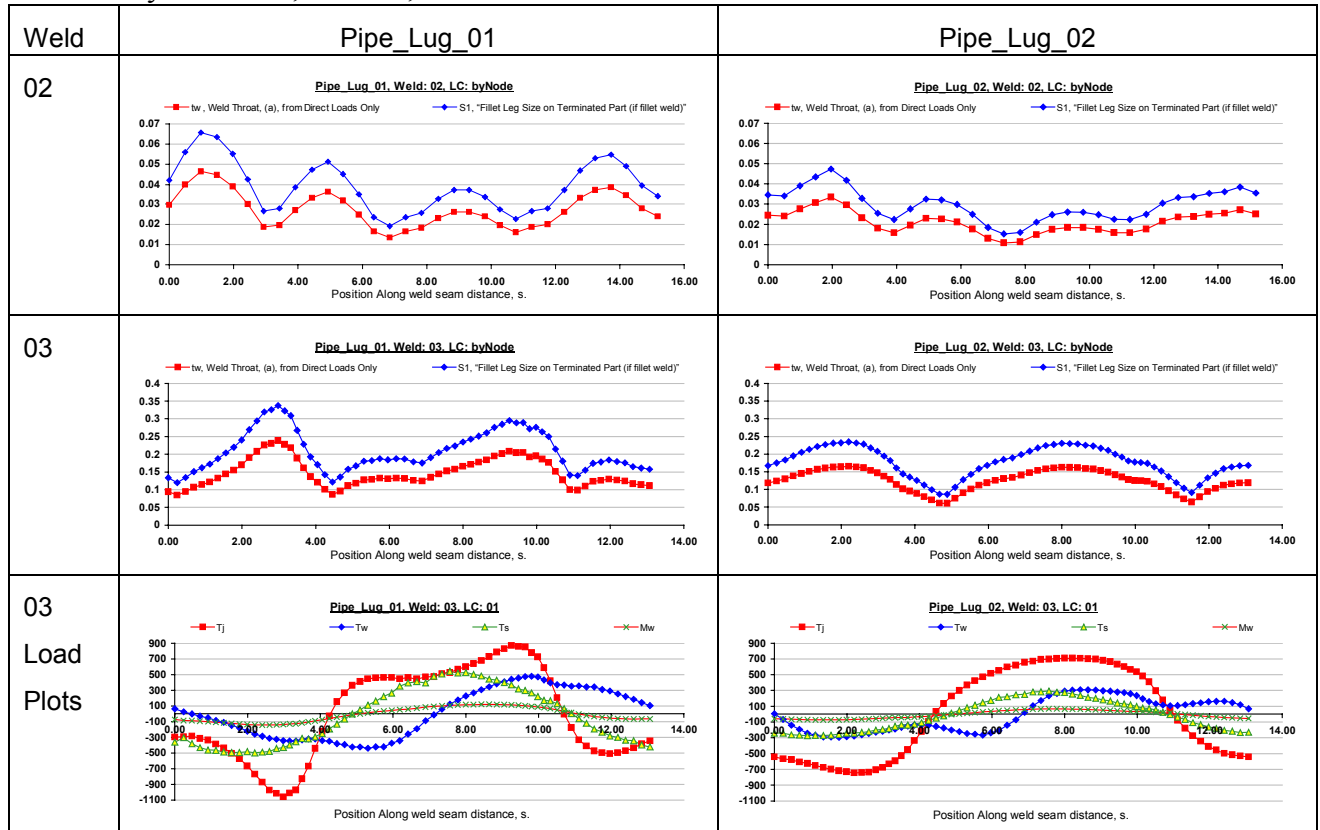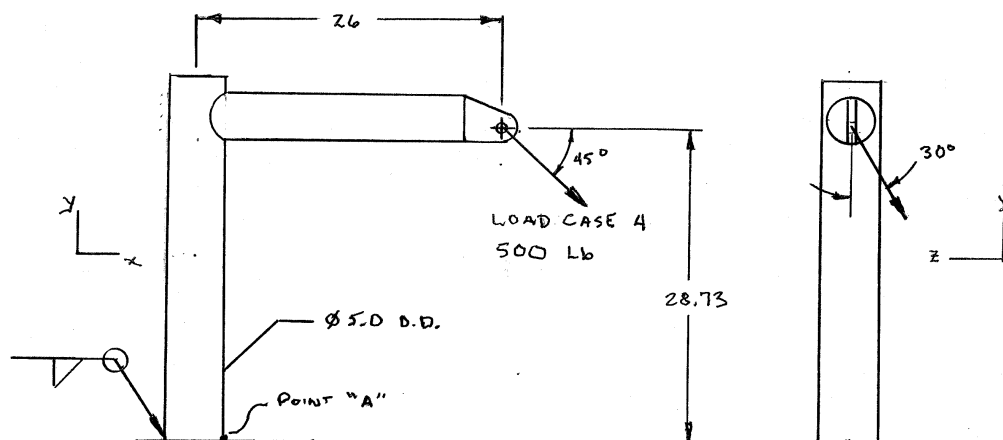
Figure 4-27 Result Plots for Welds 01 through 03

4.1.6. Concluding Notes for the Pipe Lug Example.

Fatigue:

If you perform fatigue analysis, then the analysis just presented is sufficient for determining the fillet sizes for adherence to the published 'throat shear' allowables for fatigue, however, for evaluation against base material allowables based on the 'hot spot' method (also known as the structural stress method or geometric stress method), further evaluation is necessary. One of the Weld Formulations (Weld Types) in FEWeld is called *'Stress'* . This formulation simply outputs the base metal stress tensor on each side of the terminated part in weld joint coordinates. The transverse stresses can be compared against category E or X2 (AWS D1.1) or other base metal fatigue allowables. There is also the *'notch trans'* formulation to apply effective toe and root notch stress concentration factors for membrane and bending loads to output the worst case effective notch stress due to transverse loads (you supply the effective notch stress concentration factors as weld parameters). For fatigue, additional welds would need to be defined for the Pipe_Lug example to look at the base metal stress at all material load paths passing through or past the weld joint.

4.1.7. Classical Calculation for Weld 01, Load Case 04:

WELD 01, LOAD CASE 4:

LOADS

$F_x = (500 \text{ lb})(\cos 30°)(\cos 45°) = 306.2 \text{ lb}$

$F_y = (500 \text{ lb})(\cos 30°)(\sin 45°) = 306.2 \text{ lb}$

$F_z = (500 \text{ lb})(-\sin 30°) = -250 \text{ lb}$

$M_x = F_z \cdot 28.73 \text{ in} = 7180 \text{ in-lb}$

$M_y = F_z \cdot 26 \text{ in} = 6500 \text{ in-lb}$

$M_z = F_x \cdot 28.73 \text{ in} + F_y \cdot 26 \text{ in} = 16760 \text{ in-lb}$

WELD SECTION PROPERTIES:

$A = \pi D = \pi \cdot 5 \text{ in} = 15.7 \text{ in}$

$S_x = S_z = \dfrac{\pi D^2}{4} = \dfrac{\pi \cdot (5 \text{ in})^2}{4} = 19.63 \text{ in}^2$

$J_y = \dfrac{\pi D^3}{4} = \dfrac{\pi (5 \text{ in})^3}{4} = 98.17 \text{ in}^3$
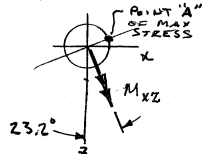
WELD STRESS PER UNIT THROAT:

IN THE Y DIRECTION

COMBINED Mx & Mz MOMENT

$M_{xz} = [(7180 \text{ in-lb})^2 + (16760 \text{ in-lb})^2] = 18230 \text{ in-lb}$

$\theta = \text{ATAN}\left(\dfrac{7180}{16760}\right) = 23.2°$

$f_y = \dfrac{F_y}{A_w} + \dfrac{M_{xz}}{S_w} = \dfrac{306.2 \text{ lb}}{15.7 \text{ in}} + \dfrac{18230 \text{ in-lb}}{19.63 \text{ in}^2}$

$f_y = 948 \text{ lb/in}$

WELD SHEAR DUE TO My

$f_{M_y} = \dfrac{M_y \, r}{J} = \dfrac{(6500 \text{ in-lb})(2.5 \text{ in})}{98.17 \text{ in}^3}$

$f_{M_y} = 165 \text{ lb/in}$

X DIRECTION WELD STRESS

$f_x = \dfrac{F_z}{A_w} - f_{M_y} \sin 23.2°$

$f_x = \dfrac{306.2 \text{ lb}}{15.7 \text{ in}} - 165 \dfrac{\text{lb}}{\text{in}} \sin 23.2° = 84 \dfrac{\text{lb}}{\text{in}}$

Z DIRECTION STRESS

$f_z = \dfrac{F_z}{A_w} + f_{M_y} \cos 23.2°$

$f_z = \dfrac{250 \text{ lb}}{15.7 \text{ in}} + 165 \dfrac{\text{lb}}{\text{in}} \cos 23.2° = 168 \dfrac{\text{lb}}{\text{in}}$

TOTAL MAGNITUDE OF WELD STRESS PER UNIT THROAT:

$f_w = [f_x^2 + f_y^2 + f_z^2]^{1/2} = 966 \text{ lb/in}$

REQUIRED WELD SIZE:

THROAT:

$t_w = \dfrac{f_a}{F_A} = \dfrac{966 \text{ lb/in}}{7000 \text{ psi}} = 0.138 \text{ in}$ ← THROAT

FILLET SIZE:

$S = \sqrt{2} \, t_w = 0.195 \text{ in}$ ← FILLET SIZE

The fillet size calculated in FEWeld was .215 inch, a difference of 10%. Figure 4-28 shows the weld loads calculated from the FEWeld analysis. Note, however, if the base plate is changed to 0.50 inch, FEWeld predicts a maximum weld size of .265 inch. This is a real effect due to the additional softness of the base plate toward the center that is not accounted for with classical analysis.
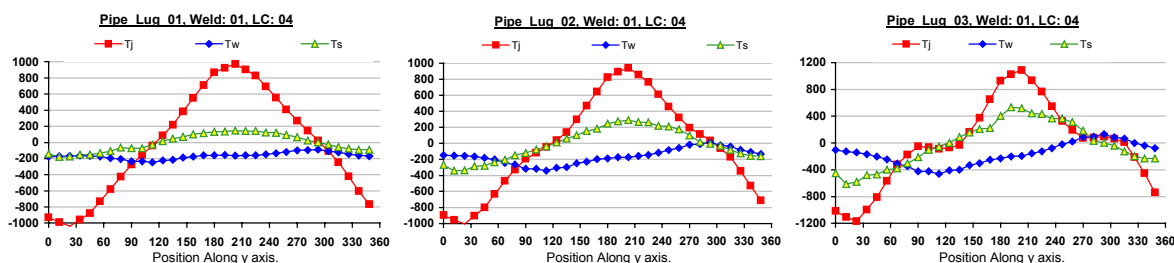
Figure 4-28 Load Plots From FEWeld for Weld 01, Load Case 04, .25 and .50 thick vertical post with 1.0 inch Base, .25 vertical post with 0.50 inch Base.

4.2. Example Conclusion:

This example exercised many of the features of using FEWeld for analysis of welds. Even so, the analysis is fast, the more so once you learn to move around the program. The weld loads for the problem presented are readily calculated with classical analysis. Performing classical analysis on this weldment would take some time, however. Several of the welds may have to be calculated at more than one load condition to determine the worst case. Often, with complex weldments, calculating the load distribution along the weld joint using classical analysis is difficult.

Future Enhancements:

- FEWeld will have facilities for general load case interactions that are intended to be able to handle user specifications from simple Goodman diagrams, to nonlinear interaction formulae, to cycle counting methods. This facility will be general, just like the weld formulation language is general.

- Simple Load Path Accounting for applying effective notch stress concentration factors multiple load path joints such as to cruciform, T, and Lap Joints.

Other:

The FEWeld user manual covers the material introduced here in much greater depth. It is ~180 pages in length.