ASCENT - Center for Technical Knowledge®
Creo Simulate 3.0
Structural and Thermal Analysis
1st Edition

Prepared and produced by:
ASCENT Center for Technical Knowledge
630 Peter Jefferson Parkway, Suite 175
Charlottesville, VA 22911
866-527-2368
www.ASCENTed.com

Lead Contributor: Scott Hendren

ASCENT - Center for Technical Knowledge is a division of Rand Worldwide, Inc., providing custom developed knowledge products and services for leading engineering software applications. ASCENT is focused on specializing in the creation of education programs that incorporate the best of classroom learning and technology-based training offerings.

We welcome any comments you may have regarding this training guide, or any of our products. To contact us please email: feedback@ASCENTed.com.

© ASCENT - Center for Technical Knowledge, 2017

All rights reserved. No part of this guide may be reproduced in any form by any photographic, electronic, mechanical or other means or used in any information storage and retrieval system without the written permission of ASCENT, a division of Rand Worldwide, Inc.

Creo Parametric 3.0 is a registered trademark of Parametric Technology Corporation.

All other brand names, product names, or trademarks belong to their respective holders.

General Disclaimer:
Notwithstanding any language to the contrary, nothing contained herein constitutes nor is intended to constitute an offer, inducement, promise, or contract of any kind. The data contained herein is for informational purposes only and is not represented to be error free. ASCENT, its agents and employees, expressly disclaim any liability for any damages, losses or other expenses arising in connection with the use of its materials or in connection with any failure of performance, error, omission even if ASCENT, or its representatives, are advised of the possibility of such damages, losses or other expenses. No consequential damages can be sought against ASCENT or Rand Worldwide, Inc. for the use of these materials by any third parties or for any direct or indirect result of that use.

The information contained herein is intended to be of general interest to you and is provided "as is", and it does not address the circumstances of any particular individual or entity. Nothing herein constitutes professional advice, nor does it constitute a comprehensive or complete statement of the issues discussed thereto. ASCENT does not warrant that the document or information will be error free or will meet any particular criteria of performance or quality. In particular (but without limitation) information may be rendered inaccurate by changes made to the subject of the materials (i.e. applicable software). Rand Worldwide, Inc. specifically disclaims any warranty, either expressed or implied, including the warranty of fitness for a particular purpose.
Contents

Preface ............................................................................................................................ vii

In this Guide .................................................................................................................... ix

Practice Files .................................................................................................................. xi

Chapter 1: Introduction to Creo Simulate ...................................................................... 1-1
  1.1 Finite Element Analysis (FEA) ............................................................................. 1-2
  1.2 FEA Solution Refinement .................................................................................... 1-5
  1.3 P-Elements .......................................................................................................... 1-7
  1.4 Convergence Methods ....................................................................................... 1-8
  1.5 Types of Analysis in Creo Simulate ..................................................................... 1-11
  1.6 FEA Process ........................................................................................................ 1-12
  1.7 CAD Model Preparation ..................................................................................... 1-13
  1.8 Creo Simulate Modes of Operation .................................................................... 1-14

Chapter 2: Basic Creo Simulate Modeling .................................................................... 2-1
  2.1 Launching Creo Simulate .................................................................................... 2-2
    Structure Ribbon ........................................................................................................ 2-3
  2.2 Modeling Steps ................................................................................................... 2-4
    Defining Model Type .............................................................................................. 2-4
    Creating Idealizations ............................................................................................ 2-4
    Applying Material .................................................................................................. 2-5
    Applying Constraints ............................................................................................ 2-6
    Applying Loads ...................................................................................................... 2-7
    Meshing the Model ............................................................................................... 2-8
  2.3 Analysis ................................................................................................................. 2-9
    Analysis Types ........................................................................................................ 2-9
    Convergence Methods ........................................................................................... 2-10
    Design Studies ....................................................................................................... 2-11
# Creo Simulate 3.0: Structural and Thermal Analysis

## 2.4 Results ................................................................. 2-12
- Files and Directories .............................................. 2-12
- Result Verification ............................................... 2-13
- Result Visualization ............................................ 2-14

## Practice 2a Static Stress Analysis of a Bracket .............. 2-15
## Practice 2b Static Stress Analysis of a Bike Crank .......... 2-50
## Practice 2c Static Stress Analysis of a Cast Part ............ 2-83

## Chapter 3: Loads and Constraints .................................. 3-1

### 3.1 Constraints .......................................................... 3-2
- Rigid Body Motions .............................................. 3-2
- Constraint Sets .................................................. 3-3
- Displacement Constraint ...................................... 3-4
- Planar, Pin, and Ball Constraints ......................... 3-5
- Symmetry Constraints ......................................... 3-7

### 3.2 Loads ................................................................. 3-10
- Load Sets .......................................................... 3-10
- Force/Moment Load .......................................... 3-11
- Pressure Load ................................................... 3-13
- Bearing Load .................................................... 3-14
- Gravity Load ...................................................... 3-15
- Centrifugal Load ................................................. 3-16
- Temperature Load ............................................. 3-17
- Preload ............................................................. 3-18

### 3.3 Surface Regions ................................................... 3-20

### 3.4 Singularities ....................................................... 3-21
- Types of Singularities ......................................... 3-21
- Excluded Elements .............................................. 3-22
- Guidelines ....................................................... 3-24

## Practice 3a Stress Analysis of a Crank ......................... 3-25
## Practice 3b Cyclic Symmetry Constraints ..................... 3-55
## Practice 3c Surface Regions ....................................... 3-69
## Practice 3d Excluded Elements ................................. 3-87
## Practice 3e Static Stress Analysis of a Shaft .................. 3-95

## Chapter 4: Shell Idealizations ....................................... 4-1

### 4.1 Shell Idealizations ................................................. 4-2

### 4.2 Midsurface Shells ................................................ 4-4
- Shell Pair Creation ............................................. 4-5
- Manual Shell Pairs ............................................. 4-6
- Automatic Shell Pairs ......................................... 4-7
- Shell Pair Compression ....................................... 4-8
## Contents

### 4.3 Standard Shells
- Standard Shell Creation .......................................................... 4-9

### 4.4 Applying Loads and Constraints to Shell Models .................. 4-11
- Practice 4a Automatic Shell Creation ........................................ 4-12
- Practice 4b Manual Shell Creation .............................................. 4-37
- Practice 4c Shells from Surfaces ................................................. 4-64
- Practice 4d Shell and Solid Combination .................................... 4-81
- Practice 4e Analysis of a Sheet Metal Bracket ......................... 4-98

### Chapter 5: Beams and Frames .................................................. 5-1
- 5.1 Beam Elements ..................................................................... 5-2
  - Beam Definitions .................................................................... 5-3
  - Beam Sections ........................................................................ 5-5
- 5.2 Beam Coordinate Systems ..................................................... 5-6
- 5.3 Beam Action Coordinate System (BACS) ............................... 5-7
- 5.4 Beam Shape Coordinate System (BSCS) ............................... 5-10
- Practice 5a Beam Analysis ....................................................... 5-12
- Practice 5b 2D Frame Analysis .................................................. 5-34
- Practice 5c 3D Frame Analysis .................................................. 5-52
- Practice 5d Analysis of a Grain Hopper ..................................... 5-65

### Chapter 6: Sensitivity and Optimization Design Studies ............ 6-1
- 6.1 Design Considerations .......................................................... 6-2
  - Objectives .............................................................................. 6-3
  - Measures ............................................................................... 6-3
  - Design Variables .................................................................... 6-3
- 6.2 Types of Design Studies ........................................................ 6-4
  - Standard Studies ................................................................... 6-4
  - Sensitivity Studies ................................................................ 6-4
  - Optimization Studies ............................................................. 6-4
  - Setting Up Design Studies ...................................................... 6-5
- 6.3 Design Variables ................................................................... 6-6
- Practice 6a Design Study of a Hinge Plate ................................. 6-7
- Practice 6b Design Study of a Hanger ........................................ 6-31

### Chapter 7: Assembly Interfaces ............................................... 7-1
- 7.1 Types of Interfaces ............................................................... 7-2
- 7.2 Bonded Interface ................................................................. 7-3
- 7.3 Free Interface ....................................................................... 7-4
7.4 Contact Interface ......................................................... 7-5  
    Automatic Contact Detection ............................................ 7-8  
7.5 Reviewing Interfaces .................................................... 7-9  
7.6 Setting Up Contact Analysis .......................................... 7-10  
7.7 Mesh Refinement ......................................................... 7-13  
Practice 7a Door Handle Assembly ..................................... 7-14  
Practice 7b Pin-Jointed Assembly ....................................... 7-31  
Practice 7c Press-Fit Analysis ............................................. 7-53  
Practice 7d Contact Analysis of a Skid Shoe ....................... 7-68  

Chapter 8: Thermal Analysis ............................................... 8-1  
8.1 Modes of Heat Transfer ................................................ 8-2  
    Conduction ........................................................................ 8-2  
    Convection ......................................................................... 8-3  
    Radiation ........................................................................... 8-3  
8.2 Creo Simulate Thermal .................................................. 8-4  
    Thermal Ribbon ................................................................. 8-5  
8.3 Modeling Steps ............................................................. 8-6  
    Interfaces ......................................................................... 8-6  
    Idealizations ..................................................................... 8-6  
    Applying Boundary Conditions ........................................... 8-7  
    Applying Heat Loads ......................................................... 8-10  
8.4 Analysis ......................................................................... 8-12  
    Analysis Types ................................................................. 8-12  
8.5 Results .......................................................................... 8-13  
    Result Visualization .......................................................... 8-13  
8.6 Thermal Load Transfer .................................................... 8-14  
Practice 8a Thermal Steady State Analysis ......................... 8-16  
Practice 8b Thermal Deformation Analysis of an Assembly ...... 8-32  

Chapter 9: Modal Analysis ................................................. 9-1  
9.1 Natural Frequency ......................................................... 9-2  
9.2 Natural Modes .............................................................. 9-3  
9.3 Defining a Modal Analysis ............................................. 9-4  
Practice 9a Modal Analysis of a Bracket ............................. 9-6  
Practice 9b Unconstrained Modal Analysis ......................... 9-17
## Chapter 10: Welds, Springs, and Masses

10.1 Weld Connections
- Spot Welds
- End Welds
- Perimeter Welds
- Weld Features

10.2 Springs
- Stiffness
- Orientation

10.3 Masses
- Practice 10a Spot Welds
- Practice 10b Perimeter Welds
- Practice 10c Springs and Masses Analysis
- Practice 10d Bracket Supported by Rubber Bushings

## Chapter 11: Fasteners and Rigid Links

11.1 Rigid Links
- Creating Rigid Links

11.2 Fasteners
- Prerequisites
- Part Separation
- Fastener Preload
- Creating Fasteners
- Fastener Measures
- Practice 11a Modal Analysis of a PCB Assembly
- Modeling Tasks
- Analysis Tasks
- Results Tasks
- Practice 11b Static Analysis of a Mixed Solid/Shell/Beam Model
- Practice 11c Fasteners with Preload
- Practice 11d Handling Surface Separation Issues When Using Fasteners
- Practice 11e Fasteners in Shell Models

## Chapter 12: Buckling Analysis

12.1 Theory of Buckling

12.2 Creo Simulate Buckling Analysis
- Practice 12a Buckling Analysis of a Pole
- Practice 12b Buckling Analysis of a Thin Shell
Appendix A: Basics of Structural Analysis

A.1 Quantities and Units
   Fundamental Quantities
   Derived Quantities
   Units

A.2 Newton’s Laws
   Equilibrium & Free-Body Diagrams
   Stresses
   3D Stresses
   Failure Theories

Appendix B: Poisson’s Ratio Project

B.1 Poisson’s Ratio
   Modeling Tasks
   Analysis Tasks
   Results Tasks

Appendix C: Verification and Practice Examples

C.1 Structural Analysis
   Case 1 - Cantilevered I-Beam
   Case 1 - Plate with a Hole
   Case 1 - Stepped Circular Bar
   Case 1 - Loaded Plate with Hole
   Practice C1 Structural Analysis Examples - Set 2
   Case 1 - Straight Beams
   Case 2 - Rectangular Plate

Appendix D: Conversion Factors
Preface

This student guide covers the fundamentals of Creo Simulate: Structural and Thermal Analysis. It provides you with the knowledge to effectively use Creo Simulate for finite element analysis, thereby reducing design time. Many concepts apply to both Structure and Thermal analysis; a portion of this guide is specifically dedicated to Thermal analysis. This is an extensive hands-on training guide, in which you have the opportunity to apply your knowledge through real-world scenarios and examples.

Topics Covered

• FEA Fundamentals: P-elements and analysis convergence methods
• Basic Modeling and Analysis
• Types of Loads and Constraints
• Idealizations: Shells and Beams
• Sensitivity and Optimization Studies
• Assembly Interfaces and Contact Analysis
• Thermal Analysis
• Modal Analysis
• Welds, Springs, and Masses
• Fasteners and Rigid Links
• Buckling Analysis

Note on Software Setup

This student guide assumes a standard installation of the software using the default preferences during installation. Lectures and practices use the standard software templates and default options for the Content Libraries.
In this Guide

The following images highlight some of the features that can be found in this Student Guide.

Practice Files
The Practice Files page tells you how to download and install the practice files that are provided with this student guide.

Chapters
Each chapter begins with a brief introduction and a list of the chapter’s Learning Objectives.
### Instructional Content

Each chapter is split into a series of sections of instructional content on specific topics. These lectures include the descriptions, step-by-step procedures, figures, hints, and information you need to achieve the chapter’s Learning Objectives.

### Practice Objectives

Practices enable you to use the software to perform a hands-on review of a topic. Some practices require you to use prepared practice files, which can be downloaded from the link found on the Practice Files page.
Chapter 1

Introduction to Creo Simulate

Creo Simulate is a powerful software tool that enables you to simulate structural and thermal behavior of your design to understand and improve the design’s performance.

Learning Objectives in this Chapter

• Understand the concept of FEA.
• Understand the concepts of H-refinement and P-refinement.
• Understand the advantages of P-elements.
• Understand solution convergence methods in Creo Simulate.
• Understand the analysis abilities of Creo Simulate.
• Understand the steps involved in a Creo Simulate analysis.
• Understand the recommendations for CAD model preparation.
• Understand the Creo Simulate modes of operation.
1.1 Finite Element Analysis (FEA)

Finite Element Analysis is a numerical mathematical method based on the following process:

- Discretize (i.e., divide) the model into smaller and more simplified volumes (tetrahedra, bricks, wedges, etc.) called finite elements. The collection of finite elements approximates the shape of the model, and is called finite element mesh, or just mesh. An example of a meshed model is shown in Figure 1–1.

In 2D models, finite elements are triangles or quadrilaterals. In 3D models, finite elements are 4-node tetra, 6 node wedge or 8-node bricks.

- Approximate the variation of the principal quantity of interest (such as displacement, stress, etc.), within each finite element with polynomials. These polynomials are typically called local approximation functions or shape functions.

- Connect the finite elements across the inter-element boundaries, thus effectively sewing elemental polynomials together. The sewn local polynomials now approximate a variation of the quantity of interest over the entire model, and therefore comprise the global approximation function in the form of a piece-wise polynomial.
• Solve the governing equations and boundary conditions for the global approximation function, and find the best fitting solution. In structural mechanics, the principle of minimum total potential energy is typically used to find the best fitting solution, which results in solving a large number (sometimes hundreds of thousands), of simultaneous linear equations.

• Present the results for this approximate solution.

Therefore, the key FEA concept is the use of piece-wise polynomials to approximate the sought field quantity in the model, which effectively replaces a continuum problem with an infinite number of degrees of freedom (DOF) by a discrete problem with a finite number of DOF (i.e., finite elements and discretization).

For example, consider how the FEA method works when applied to calculate deflections in a simple beam as shown in Figure 1–2. The beam is clamped at the left end, has a couple of supports in the middle, and is loaded by a couple of transversal forces and a moment. The bottom graph shown in Figure 1–2 represents the unknown true deflection of the beam, which you are trying to determine using the FEA method.
The first step in the process (shown in the example in Figure 1–3) is to mesh the beam by breaking it into a collection of shorter pieces (i.e., finite elements) connected at their ends (i.e., the nodes).

Next, the deflection $Y$ within each finite element is approximated by a polynomial. In this example, you use linear polynomial $Y = a_0 + a_1X$, which means that deflection within each element is approximated by essentially a straight line.

Next, the local linear polynomials are sewn together at the nodes, creating a global approximation function in the form of a piece-wise linear polynomial, which is a polyline.

Finally, the global approximation function is best-fit to satisfy both the bending differential equations and beam boundary conditions (loads and constraints). The resulting function (the dashed line shown in Figure 1–3) now represents the FEA solution for the true deflection (the solid line shown in Figure 1–3) in the beam.

It is important to note that your FEA result contains a certain amount of error, which is the deviation between the true deflection (the solid line shown in Figure 1–3) and the FEA solution (the dashed line shown in Figure 1–3), and which is called a discretization error.

Any FEA solution is just an approximation, which means it always contains a discretization error. Therefore, in the FEA process, it is critical to know how to estimate, how to control, and how to reduce this unavoidable approximation error to acceptable levels.
1.2 FEA Solution Refinement

The process of bringing the FEA approximation error to acceptable levels is typically called *solution refinement*. There are two alternative ways in which an FEA solution can be refined.

The first option involves making the finite elements in the mesh progressively smaller while maintaining the order of polynomials within each element.

For example, consider the beam shown in Figure 1–3. If you make the finite elements smaller without changing anything else, the approximation error becomes smaller as well, as shown in Figure 1–4.

![Figure 1–4](image)

This approach is called *h-refinement* because the letter *h* in FEA literature typically refers to the size of the finite elements in the mesh. It is also worth noting that h-refinement requires re-meshing the model every time you need a more accurate solution.

The h-refinement approach is used by most FEA software systems that are commercially available today. However, this is not the only available option.

An alternative strategy involves increasing the order of polynomials within the finite elements, without changing the elements’ sizes.
Again, consider the example of the beam shown in Figure 1–3. If you use second-degree polynomials $Y = a_0 + a_1X + a_2X^2$ to approximate the deflection within each element, this results in a more accurate solution, without needing to make the finite elements smaller, as shown in Figure 1–5.

![Figure 1–5](image)

This approach is called **p-refinement**, because the letter $p$ in FEA literature typically refers to the order of polynomials, and the process of progressively increasing the polynomial order is called **polynomial escalation**.

The p-refinement mathematical apparatus has been historically developed much later than the h-refinement. Today, the p-refinement approach is only used by a few commercial FEA software systems, one of which is Creo Simulate.
1.3 P-Elements

Creo Simulate exclusively uses p-elements and p-refinement technology to ensure the accuracy of the solution. The maximum polynomial order in Creo Simulate can be as high as 9. (The maximum polynomial order in the h-version of FEA is typically 2.)

The advantages of p-technology over h-technology are as follows:

- Solution accuracy can be improved without having to re-mesh the model.
- P-elements use hierarchical polynomials, which permits the use of different polynomial orders in different areas of the model for better efficiency.
- The rate of convergence to the true solution is greater than that of the h-technology.
- High stress gradients, such as in stress concentrators, are simulated extremely well.
- The restrictions on the shape of elements (aspect ratio, skewness, etc.) are less stringent. Therefore a p-mesh always contains fewer elements that an h-mesh.
- P-elements have curvilinear boundaries and tend to approximate CAD geometry very well.

The rate of convergence refers to how quickly the refinement process converges to the true solution.
1.4 Convergence Methods

Convergence in FEA (also called adaptivity) is a process of automatic solution refinement to achieve the required accuracy. In other words, the FEA software automatically adapts the solution parameters to better fit the true solution.

One of the key advantages of p-refinement over h-refinement is that p-elements permit an adaptive solution improvement without re-meshing the model. Instead, the maximum orders of polynomials used to approximate the solution are increased as needed. The solution process can then be repeated on the same mesh, with the new increased polynomial orders. Such an adaptive step (called pass in Creo Simulate) can be repeated until the required accuracy is achieved.

In p-elements, the polynomial orders (called P-levels in Creo Simulate) can be assigned independently to each edge, face, or solid in the mesh. Using the convergence algorithm, Creo Simulate can pick P-levels independently for each mesh edge in the model, the goal being to select just the correct P-levels to achieve the required solution accuracy at minimum computational expense.

Multi-Pass Adaptivity (MPA) is the most commonly used convergence method in Creo Simulate. To identify the edges that warrant a P-level increase, the MPA algorithm compares displacements and element strain energies on the current solution pass with the corresponding values on the previous pass. Where the difference is larger than the user-specified percentage (i.e., convergence percentage), the P-level is increased and otherwise left unchanged. This process is repeated until the user-specified convergence percentage for the solution is met. The convergence criteria might include percentages on the default local and global quantities, such as displacement, strain energy, and RMS stress, but could also involve user-defined solution parameters.

The MPA convergence graphs can be visualized once the analysis has finished, as shown in Figure 1–6.
Since convergence percentage(s) are selected by the user, the MPA algorithm provides the user with maximum control, and is best used if the accuracy of the solution is critical.

The second convergence algorithm in Creo Simulate, called Single-Pass Adaptivity (SPA), uses a different theoretical foundation to reach an accurate solution.

The SPA algorithm is based on the fact that, although displacements in an FEA solution are continuous between elements, stresses are not, and the magnitude of stress jump at the discontinuity is a good indicator of the solution accuracy (i.e., the greater the stress jump, the less accurate the solution).

In the SPA algorithm, Creo Simulate first calculates the solution for P-level 3, assigned uniformly to all edges, and average stress discontinuities around each element (i.e. element error indicators) are computed. The P-levels of edges belonging to elements with large stress jumps are increased. Edges of elements with larger errors receive a higher P-level increase than edges of elements with lower errors. The solution is then repeated, and the result obtained at this point is taken as the final answer. The element error indicators are recomputed to indicate the overall stress accuracy.

Since only two convergence passes are performed in SPA, the computation time is typically much shorter than in MPA.
No convergence graphs are available in the SPA algorithm. Instead, the RMS stress error estimate is printed out to the Creo Simulate report file, as shown in Figure 1–7.

**RMS Stress Error Estimates:**

<table>
<thead>
<tr>
<th>Load Set</th>
<th>Stress Error</th>
<th>% of Max Prin Str</th>
</tr>
</thead>
<tbody>
<tr>
<td>LoadSet1</td>
<td>1.26e+01</td>
<td>10.3% of 1.22e+02</td>
</tr>
</tbody>
</table>

**Figure 1–7**

The SPA algorithm has been optimized by PTC with the goal of obtaining as good or better result as using the MPA convergence with the default (10%) convergence percentage.

The SPA convergence method provides limited control over the accuracy of the solution. Therefore, it should be reserved for quick design-analysis iterations when solution accuracy is not critical.

The third convergence method in Creo Simulate is called *Quick Check*, and does not perform a convergence process. The model is only run once, with all of the P-levels fixed at 3. The results of a Quick Check should not be trusted. The intention of a Quick Check analysis is to quickly run the model through the solver to detect any potential modeling errors (such as in constraints), before committing to a more lengthy analysis run (such as when using MPA).
1.5 Types of Analysis in Creo Simulate

Creo Simulate analysis capabilities straddle two physics domains:

- **Structural**: Determines deformations, stresses, and strains in solid bodies caused by external forces, moments, and other types of loading.
- **Thermal**: Determines temperatures and heat fluxes in solid bodies due to heat sources and/or sinks.

The Structural part of Creo Simulate can perform the following types of analysis:

- Static (including nonlinear material models, large displacements, and contact)
- Pre-stress Static
- Buckling
- Fatigue
- Modal (Natural Vibrations)
- Pre-stress Modal
- Dynamic Time Response
- Dynamic Frequency Response
- Dynamic Random Response
- Dynamic Shock Response

The two options for the Thermal analysis are as follows:

- Steady State Thermal analysis
- Transient Thermal analysis

The models in Creo Simulate can be analyzed in 3D formulations (purely solids, or combinations of solids, shells, and beams) or 2D formulations (plane stress, plane strain, or axisymmetric).
1.6 FEA Process

A typical FEA analysis process in Creo Simulate consists of three principle steps, as shown in Figure 1–8:

- **Pre-processing**: All input data for the analysis is prepared, such as material properties, loads, and constraints.

- **Solution**: The convergence type and criteria are specified and the analysis computation is performed.

- **Post-processing**: The analysis results are reviewed and verified. A report is prepared.

Figure 1–8

The CAD model simplification step is optional. It might not be needed, depending on the complexity of the model.
1.7 CAD Model Preparation

A CAD model is developed to provide detailed information for manufacturing. All of the required information related to fillets, rounds, holes, and threads must be included. Processing steps and surface finishes are indicated and dimensions are fully specified.

An FEA model is developed to determine model behavior under a specific set of loading and boundary conditions. To analyze a model effectively, an FEA model is often different from a model developed for manufacturing. The symmetry of a model can often be used. Minor features, such as rounds, fillets, chamfers, and holes, can often be ignored unless they have a large effect on the result. Therefore, the general recommendation is to use the simplest model possible that is going to yield reliable results at the lowest computational time and cost.

In the example shown in Figure 1–9, the area of interest is the stress in the weld between two pipes due to high pressure. The FEA model within the component is shown on the right. In this case, the symmetry of the component (1/2 of the component) is used for the FEA model. The minor rounds, fillets, chamfers, and holes are ignored. The CAD model prepared for FEA would be different if the area of interest was the stress at the intersection of lips and pipes.

![Figure 1–9](image)
1.8 Creo Simulate Modes of Operation

Creo Simulate can operate in the following modes:

- **Integrated**: The simulation is fully integrated within the Creo Parametric design process (i.e., the models can be analyzed without ever leaving the Creo Parametric user interface). This is the most common way to use Creo Simulate.

- **Standalone**: Enables the loading of CAD models directly into Creo Simulate, without first loading Creo Parametric. This is useful if models originating from different software than Creo Parametric need to be analyzed.

- **FEM**: Provides pre- and post-processing capabilities only. The model has to be exported to a neutral file format and then solved by a 3rd party FEA solver (ANSYS, NASTRAN, etc.). H-elements meshing is used.

- **Simulate Lite**: Limited model size of up to 200 surfaces and also has a limited user interface. Does not require a Simulate license.

This training guide focuses on the Integrated mode, which provides the most streamlined approach to part or assembly simulation and optimization within the Creo environment.