

Tutorial on How to Do FEA in ProE

Xi Chen
November 2015

- [1. Introduction](#)
- [2. Basics of FEA](#)
- [3. Instructions on How to Do FEA in ProE](#)
 - [3.1 FEA Procedures in ProE](#)
 - [3.2 How to Build a Structural Model in ProE](#)
 - [3.2.1 Defining Load in FEA](#)
 - [3.2.2 Defining Constraints in FEA](#)
 - [3.2.3 Assigning Materials to Parts in FEA](#)
 - [3.2.4 FEA Analysis and Study](#)
- [4. Summary](#)

1. Introduction

Both ProE and SolidWorks are popular CAD softwares for mechanical design and analysis including FEA (finite element analysis). In OPTI 521 class, how to build FEA modules and run analysis were demonstrated with SolidWorks. However, a lot of companies use ProE instead of SolidWorks since ProE could handle more complicated parts and features especially for comprehensive curved boundaries or shape. In this tutorial paper, I will describe how to build FEA module and run it in ProE. It is very difficult to find good tutorial materials including papers or videos on internet teaching people how to use ProE to do FEA. I hope this paper could provide instruction and guidance on how to do FEA in ProE to mechanical engineers.

2. Basics of FEA

FEA (finite element analysis) cuts a structure into a lot of elements (pieces of the structure). Then these elements are re-connected at “nodes” as if nodes were pins or drops of glue that hold elements together. This process results in a set of simultaneous algebraic equations. These equations could be expressed as matrix equation:

$$[K] \{u\} = \{F\} \quad \text{- Eq. (1)}$$

For elastic problem, K is a part's stiffness, u is displacement or deformation and F is force. For thermal problem, K is material's thermal conductivity, u is temperature and F is heat flux.

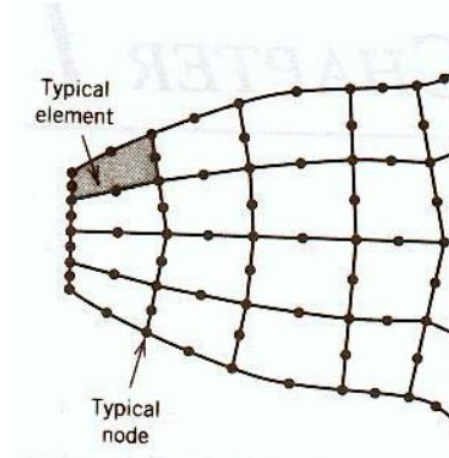


Figure 1 FEA Basics: parts are decomposed of elements and nodes

Displacement at each element could be solved as:

$$\{u^E\} = [K^E]^{-1}\{F^E\} \quad - \text{Eq. (2)}$$

Adjacent elements share the degrees of freedom at connecting nodes. By connecting elements together, the field quantity becomes interpolated over the entire structure in piecewise polynomial fashion. A set of simultaneous algebraic equations at nodes are solved. This set of equation is shown in Equ. (2). $[K]$ is the stiffness matrix. All stiffness matrix is symmetric which means $K_{ij} = K_{ji}$. This is the result of Maxwell's Reciprocity Theorem. In other words, the displacement at point i due to an **unit** load at another point j is equal to the displacement at j due to an **unit** load at i , provided that the displacements and forces "correspond," i.e., that they are measured in the same direction at each point. When the determinant of matrix $[K]$ ($\det[K] = 0$) is zero, then K could not be inverted. This leads to non-unique displacement $\{u\}$ solutions. This happens for underconstrained systems.

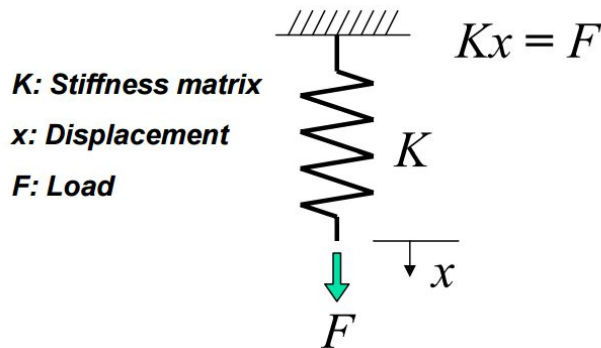


Figure 2 Stiffness matrix, displacement and force relationship

By solving the stiffness matrix equation, FEA could handle solid mechanics, dynamics, heat problems and fluid problems. Meanwhile, FEA provides "approximate" solutions. It has "inherent" errors and mistakes by users could be fatal. First, the errors could be made by simplified geometry. Secondly, field quantity is assumed to be a polynomial over an element which is not true.

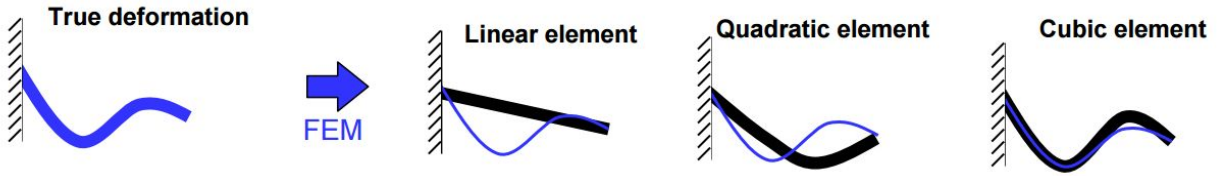
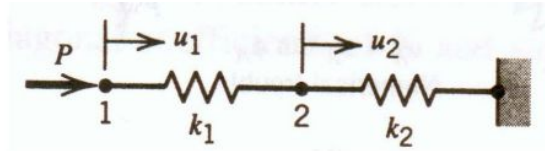


Figure 3 Illustrations of different types of elements approximation

The FEA algorithm uses very simple integration algorithm to do integration which causes errors. Also when there is very large stiffness difference in a system, the algorithm is unstable. Careful inspection and intuition of the FEA results is very important.



$$k_1 \gg k_2, \quad k_2 \approx 0$$

$$[(k_1 + k_2) - k_2]u_2 = P \Rightarrow u_2 = \frac{P}{k_2} \approx \frac{P}{0}$$

Figure 4 Ill-conditioned system (too large difference in stiffness k_1 and k_2)

3. Instructions on How to Do FEA in ProE

How to build a FEA module for structural stresses is explained in this Section. In Section 3.1, the FEA procedures in ProE are described. In Section 3.2, I will introduce how to do FEA in ProE step by step with a beam example.

3.1 FEA Procedures in ProE

The FEA procedures in ProE are shown in Fig. 5. The entire procedures include three main processes: (1) Preprocess to build a FE model. This includes identify geometry, loads and constraints. The user need divide the geometry into simpler parts and create a mesh. (2) Performing and running analysis and reviewing results. (3) Varying parameters and optimizing the design.

The model must first be simplified. Once the simplification is complete, the model is broken down into smaller elements which are connected at nodes like the edges of puzzle pieces. The actual size and shape of the elements depend on the model, and each element has a finite number of degrees of freedom (DOF). The simultaneous equations and boundary conditions of these elements will help determine how the model behaves in the analysis. Adjacent elements, or elements that share nodes, also share their DOFs at these nodes. Values at individual points are found using piecewise polynomial interpolation or a series of differential equations,

depending on the situation. This leads to an analysis of the part overall. Once this is complete the results are reviewed and, if necessary, the model is optimized.

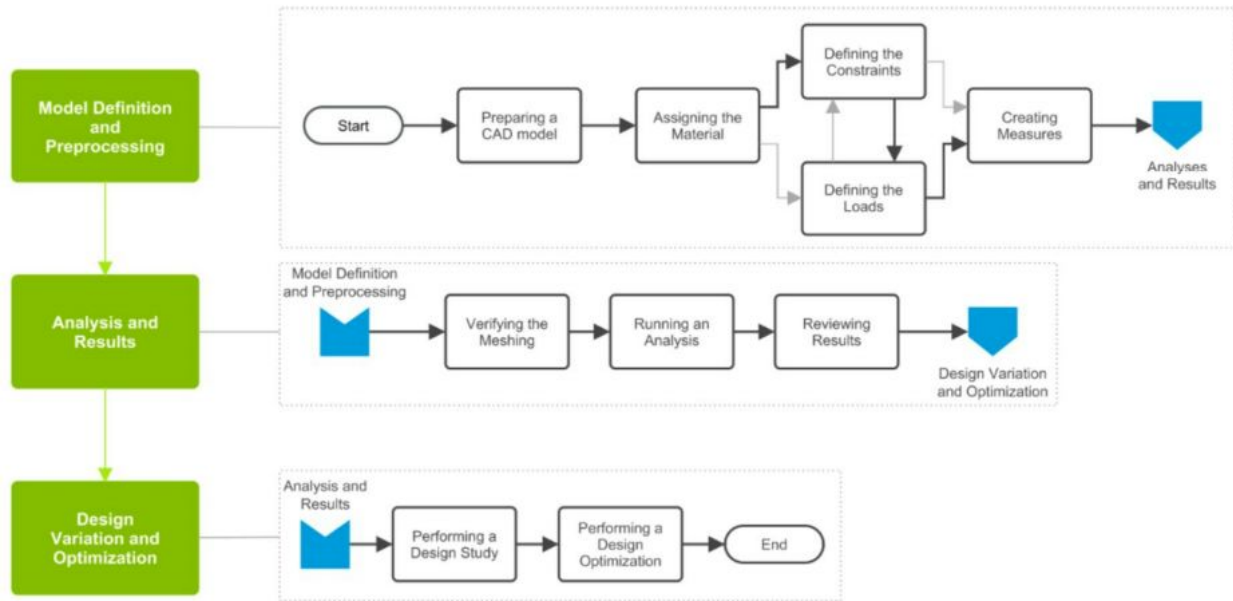


Figure 5 ProE FEA Procedure

For preprocessing step, one need select analysis type such as structure static analysis, modal analysis, transient dynamic analysis, buckling analysis, contact, steady-state thermal analysis or transient thermal analysis. Then one need select element type such as 2-D or 3-D, linear or quadratic, truss, beam, shell, plate or solid. One also need assign material properties to parts. ProE makes nodes, builds elements by assigning connectivity at nodes and apply boundary conditions and loads.

| | | |
|--------------|---|---|
| | | |
| create nodes | build elements by assigning connectivity at nodes | apply boundary conditions (constraints) and loads |

Table 1 FEA Preprocess steps

3.2 How to Build a Structural Model in ProE

In this section, the procedures on how to build a structural model for FEA analysis are described. A fixed end beam with 10N force applied on the other end is used to illustrate how to build FEA model in ProE. Alternatively, static analysis is done on the same beam with thermal gradient load instead of force load to illustrate how to calculate thermal stress in ProE.

To run FEA in ProE, one need first build the model for the part or assembly. Then one could access to FEA tool which is provoked by clicking “Tool>Simulate” in ProE. The dialog box of ProE to run FEA is shown in Fig. 6 below.

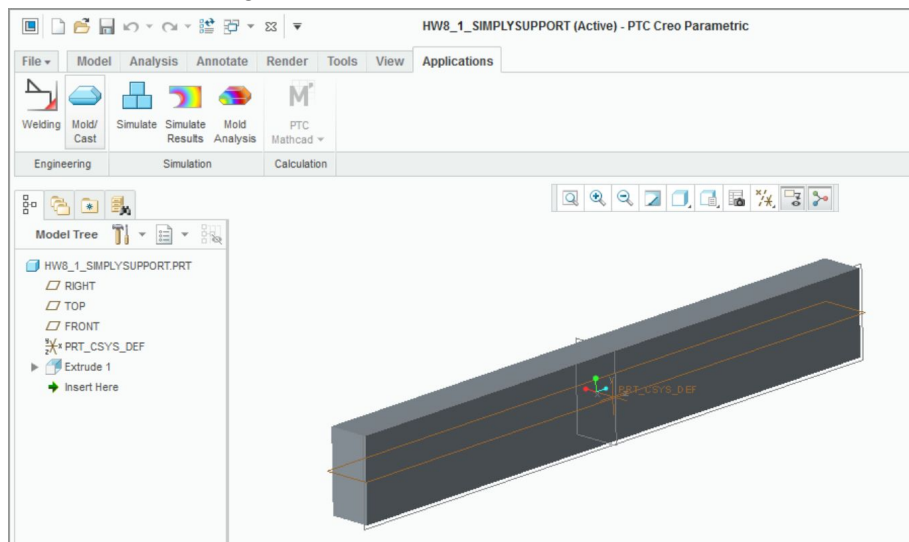


Figure 6 ProE FEA Tool “Simulate” (Click “Simulate” for structural and thermal FEA from ProE)

There are two FEA modes that could be selected: “Structure Mode” or “Thermal Mode” in ProE. In this report, we will deal with structural mode. It is important to simplify the system. **Chamfers and rounds are usually ignored to simplify system.** If one is uncertain about what parts should be kept and what parts are necessary in order to create an accurate analysis, one can use the **Inheritance** and **Remove** features. The Inheritance feature keeps the simulation model independent from the CAD geometry of the original part and will update accordingly. The Remove feature allows the user to simplify imported CAD geometry. These features take advantage of nonessential cuts and material and will save time and money while easing uncertainty. Also, note that in order to save meshing and analysis time, different types of symmetry can be used. These include mirror symmetry constraints and cyclic symmetry constraints. Mirror symmetry constraints cut a model through a plane about which it is symmetrical.

To do FEA analysis, one need first build the structural model. **The model need include defining constraints, loads and assigning materials.** Once the model definition is done, one could run “analysis and studies” for static, modal, buckling, fatigue and dynamic analyses. The ProE FEA modeling for a fixed-end beam is shown in Fig. 7

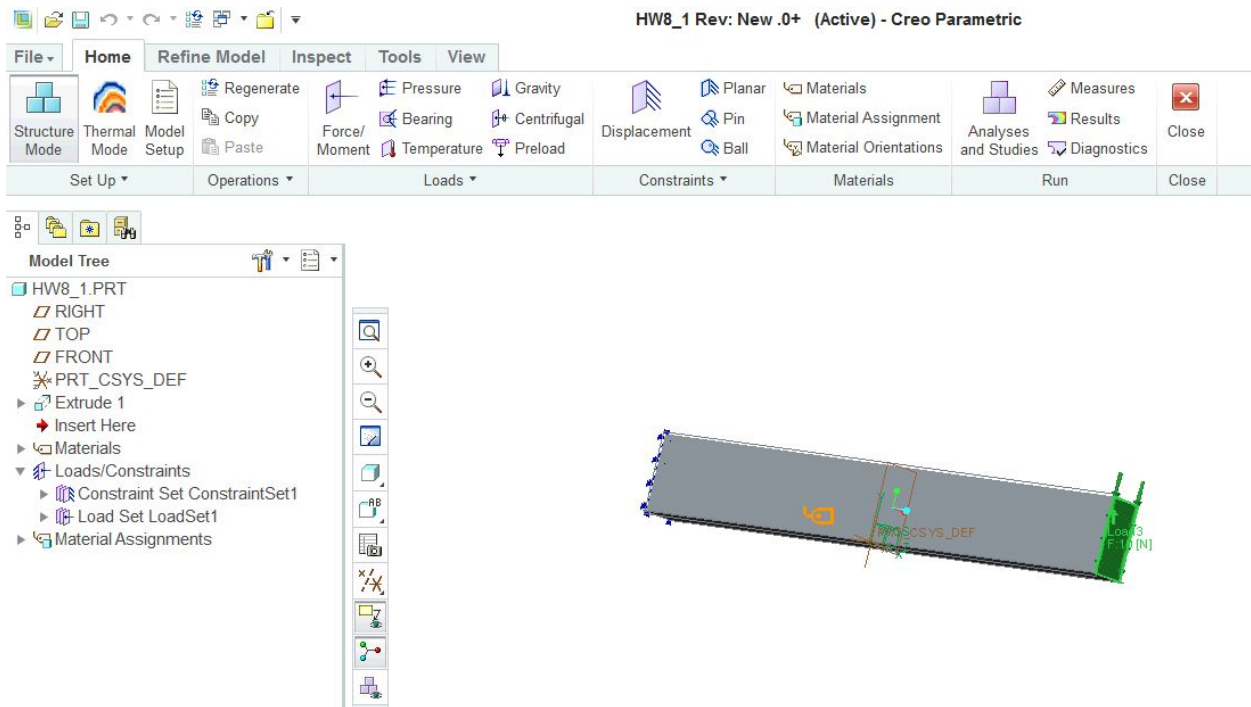


Figure 7. ProE FEA Model User Interface

3.2.1 Defining Load in FEA

To build the model, let us first define the **load** for this beam. There are several kinds of loads including “Force/ Moment”, “Pressure”, “Gravity”, “Bearing”, “Centrifugal Load”, “Temperature Load” and “Preload” in ProE. The most common load is “Force/ Moment”. For example, I applied -10N along the y-axis (green arrow direction) on the end surface highlighted in green in Fig. 7. The definition of this force in ProE is shown in Fig. 8.

There are also other kinds of loads-definitions in ProE. For instance, one could select a bearing load which is a force applied to half of a circular surface or edge. This represents a sinusoidal load distribution, a more accurate representation of the real-life effects of a bearing. This bearing force example is shown in Fig.9. If there is a rotational axis or angular velocity that needs to be applied to the part you can select a centrifugal load. For PTC Creo Simulate users, the software can calculate the radial force or torque needed to satisfy the defined requirements of the part. If gravity is a strong factor, one can include that in the analysis as well by defining “gravity” load. Loads can be applied to the part in any direction or within any degree of freedom. Loads can also be imported using PTC Creo Parametric Mechanism Design Extension. Loads on the part can be linear or function-driven, allowing the load to be interpolated over entities or coordinates. If the part experiences preload such as lenses mounted in barrel does, then preload could be defined in the model. If the part experiences considerable strain due to temperature conditions, a uniform temperature, temperature spatial variation defined by a function or imported temperature field can be used to properly model the deformation and

thermal expansion that can occur. The example of a gradient temperature load along the beam's y-axis is shown in Fig.10. One must pay attention to the default units used for the model. For the beam example given in this report, English units are applied. Temperature unit is in F. The temperature gradient function must be defined in the unit of F instead of C or K. In ProE, the x-axis is always shown as red arrow, the y-axis is in green arrow and the z-axis is in blue arrow. Once loads are defined, the loads are shown on the part.

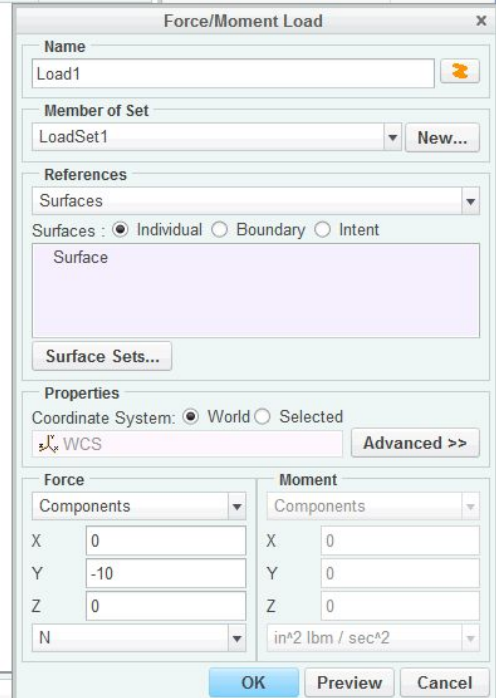


Figure 8 Defining a force/ moment in ProE

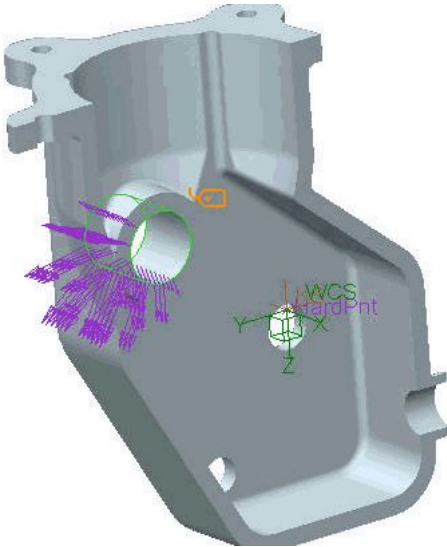


Figure 9 Bearing Load Example

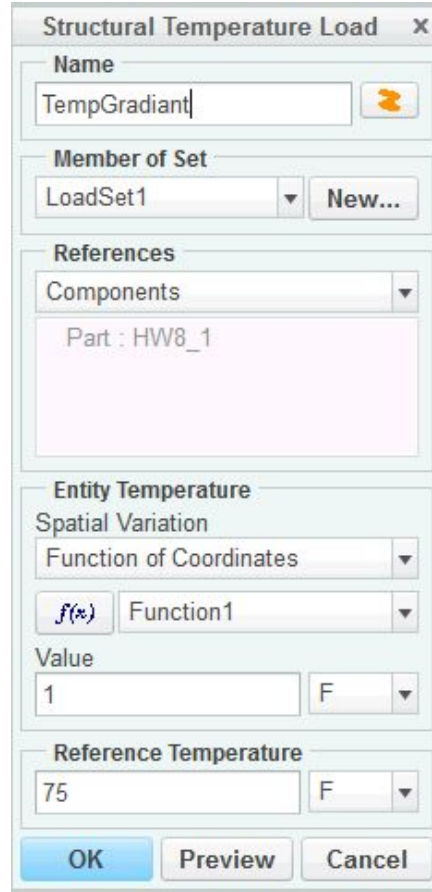


Figure 10 Temperature Load Definition. (temperature spatial variation is defined by a function: $f(x)=75+y/10$; Value of 1 is set since function $f(x)$ fully represents the temperature difference between the beam surfaces along the y-axis and no scaling factor is needed; Reference temperature is set as 75F since the zero thermal stress is at 75F.)

3.2.2 Defining Constraints in FEA

Next, one need define **constraints** for this beam. The definition of the fixed-end constraint on the beam is shown in Fig.11. The surface where the beam is fixed need be chosen. The x, y and z translations need be chosen as “fixed” instead of “free” or “prescribed”. Once the constraint is defined, one could see constraint symbol on the part where constraints are applied.

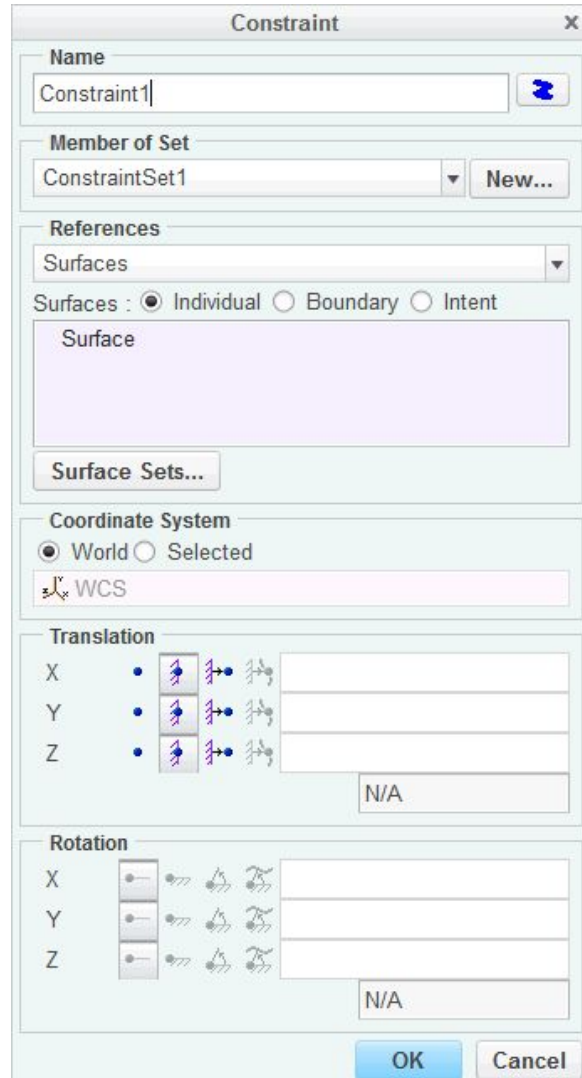



Figure 11 Displacement Constraint Definition

There are also other constraints such as **Pin**  constraint. As shown in Fig. 12 below, three holes of the part are selected which are highlighted in green once they are selected. Then in the Pin constraint dialog box, one could choose either fixed or free angular constraint and fixed or free axial constraint. The definitions of these two parameters for Pin Constraint are shown in Table 2. The Pin constraint simulates the constraints for parts interfaced through pins or screws/ bolts.

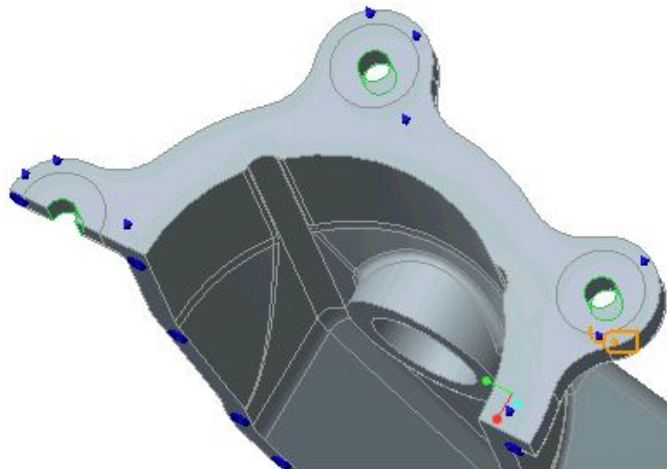


Figure 12 Example of Part with Pin Constraints


| Angular Constraint | Axial Constraint | Pin Constraint |
|--------------------|------------------|--|
| | | |
| | | Allow rotational and axial displacement but constrain radial displacement. |
| | | Allow angular displacement but constrain radial and axial displacement. |
| | | Allow axial displacement but constrain radial and angular displacement. |
| | | Constrain radial, angular, and axial displacement. |

Table 2 Pin Constraints

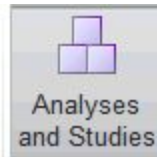
A ball constraint Ball represents a ball joint in which translation is fixed while rotation is free.

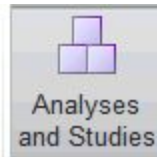
One can select only spherical surfaces for this type of constraint. A Planar constraint enables one to create a constraint that allows full planar movement, but constrains off-plane displacement. Only planar surfaces can be selected for this type of constraint. In addition to these four kinds of constraints, there is also Mirror Symmetry Constraints which enables one to analyze only a segment of a model and project the results to the entire model. Using mirror symmetry allows one to take advantage of the model's symmetry to reduce meshing and analysis time. Mirror symmetry constraints are not available in FEM mode. To use mirror symmetry successfully to analyze the model, the part or assembly must exhibit reflective symmetry through a plane. One need be cautious about the modal analysis on a model with mirror symmetry constraints or a representative segment of that model. The results will not include any nonsymmetric modes. To determine whether there are nonsymmetric modes, one should run the modal analysis with the entire model without mirror symmetry constraints.

3.2.3 Assigning Materials to Parts in FEA

To run FEA analyses, one need **assign material** to parts. This could be done by assigning material from the  **Material Assignment** dialog box.

3.2.4 FEA Analysis and Study



Finally one could run FEA analysis by clicking  in the ribbon. The **analyses and studies** dialog box then show up. The snapshot of the analyses and studies dialog box is shown in Fig. 13. From “File” menu, one could choose analysis among “new static”, “new modal”, “new buckling”, “new fatigue”, “new prestress”, “new dynamic” and sensitivity and optimization studies.

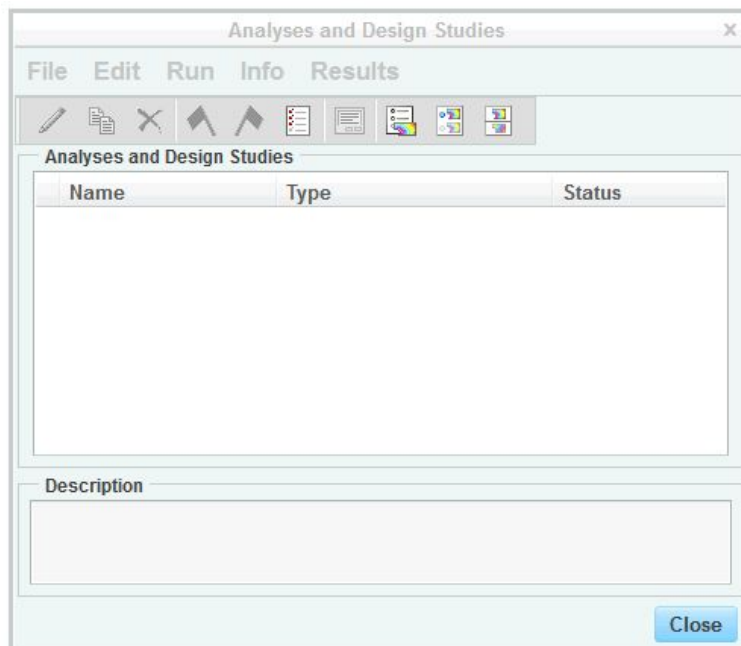


Figure 13 FEA Analyses and Studies Dialog Box

New static analysis allows one to analyze part's deflection and stress distribution. New modal analysis provides the resonant frequencies and deflection and stress distribution for selected resonant frequency. New buckling analysis provides the buckling safety factor for the part. New dynamic allows one to calculate the safety factor for specified acceleration that the part experiences. The static analysis on the fixed-end beam deflection distribution under (a) a 10N force as shown in Fig. 8 and (b) a lateral temperature gradient of 0.1F/inch as shown in Fig. 10

are shown in Fig. 14 and Fig. 15, respectively. Both FEA results agree with theoretical calculations.

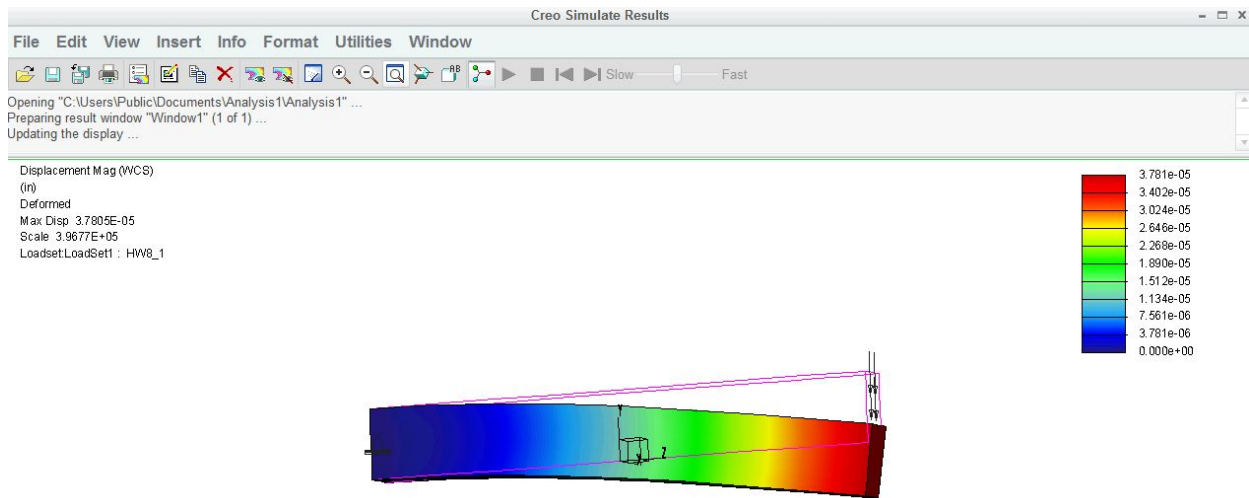


Figure 14 FEA deflection distribution result of a fixed-end beam under 10N force load

From Fig. 14, one could see the beam maximum deflection along the applied force direction (the y-axis) is 3.78×10^{-5} inch. The corresponding theoretical calculation is

$$\Delta y = \frac{FL_z^3}{3*E*I} = \frac{10*150^3}{3*69*10^9/39.37^2*10*20^3} = 3.79 * 10^{-5} (inch),$$

where L=150" (length along the z-axis), b=10" (length along the x-axis) and h=20" (length along the y-axis).

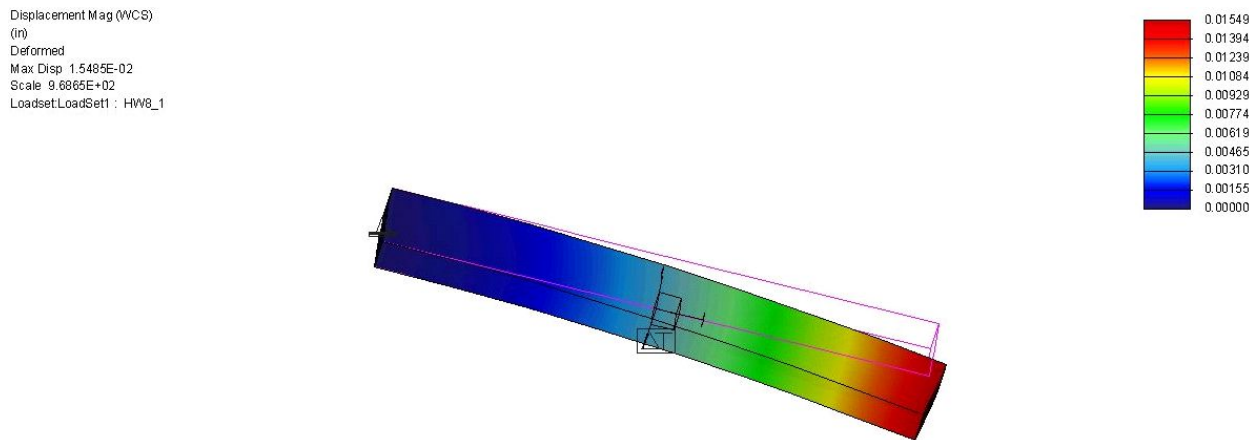


Figure 15 FEA deflection distribution result for the fixed-end beam with 0.1F/inch thermal gradient along the y-axis

From Fig.15, one could see the maximum deflection due to thermal gradient is 0.0155inch. The theoretical calculation is

$$\delta y = \frac{\alpha L^2 dT}{2} = \frac{23.6*10^{-6}*5/9*150^2*0.1}{2} = 0.0148 (inch).$$

4. Summary

How to use ProE for FEA structural analysis is described in this report. Different kinds of constraints and loads are introduced in details. The procedures for FEA modeling in ProE are described in steps. A fixed-end beam with force load and lateral thermal gradient load is given as an example.