Part A – Short Answer Questions – 30 points
Answer FOUR of the following seven questions

1. How do we modify more than one dimension at the same time in the sketcher? (pg 2-22)
   Ans: 1) Select Edit  2) Select all  3) Click Modify Dimensions  4) turn off regenerate  5) Enter new values  6) turn on regenerate  7) accept changes (click the check mark)

2. What are the differences between creating a CUT feature and creating a HOLE feature in pro-engineer? (pg 2-18, 2-24)
   Ans: For a cut feature, you would need to add a new extrude feature, which is of the shape of the cut, and then remove that shape. For a hole you simply need to select the hole tool and identify its placement and references.

3. What is the history-based part modification approach, and how do you use it (list steps). (pg 3-23)
   Ans: It’s a tool that allows us make changes to features in the model history tree in such a way that these features remain connected (or are re-linked) to the rest of the other features. The steps required for this approach are: 1) right click on the name of the feature to be modified (listed on the model tree window) 2) select edit definition, or edit references or edit 3) make the desired changes.

4. List five geometric constraints and explain how they are applied in Pro-E (pg 4-13)
   Ans: The geometric constraints are: 1) vertical/line up vertical, 2) horizontal/line up horizontal, 3) perpendicular, 4) tangent, 5) midpoint, 6) collinear/point on entity/same points, 7)Symmetric, 8) equal radii/equal length, 9) parallel

   The are applied by 1) selecting the ‘Constraints’ icon in the sketcher toolbar 2) selecting the desired constraint from the menu and 3) line(s), arc(s), circle(s) or points(s) to which the constraints are to be applied.

5. What determines how a feature reacts when other features in the model change? How do you suppress an existing feature?
   Ans: a) parent child relationships.  b) you suppress and existing feature using the ff steps: 1) right click on the name of the feature to be suppressed (listed on the model tree window) 2) select suppress

6. Describe the procedure to create an angled datum plane. (pg 6-9,10)
   Ans: 1) Select a datum axis from the model tree window, 2) select the ‘Datum plane tool’ in the datum toolbar (window opens up that lists the pre-selected axis, 3) hold down the control key and select additional references (this will be added to the references listed in the window, 4) enter in the appropriate angle properties, 5) click ok to accept the creation.

7. What are parametric relations, and how are they applied? (pg 4-25, 7-13)
   Ans: Parametric relations are designer defined relationships between dimensions on a sketch that ensure that the position or sizes of certain dependent lines, arcs, circles, or other components remain constant regardless of changes in the dimensions of independent components. They are applied by: 1) left-clicking on tools, 2) selecting relations, 3) entering the required relations or dependencies
Part B – Modeling

1. 20 points – All dimensions are in inches. Note the color!

2. 20 points – Dimensions don’t matter – only relations do.