Engineering Fundamentals
ENG1100

UGNX – Drafting
Session Objectives

- Edge Operations
- Create Engineering Drawings in UGNX
- Student Presentations
Sketch Based Modeling

1. Sketch plane is chosen
2. 2-D sketch is created
   • Sketches can be the base feature of a solid body
   • Sketches can be added to an existing body
3. 3-D feature is created from the sketch (e.g. extrude, revolve … add, subtract, intersect)

Feature can be edited by editing the *constraints* of the sketch
Edge Operations

Can be added to model in modeling application

**Edge blend**

- Creates cylindrical faces in place of an edge

**Chamfer**

- Bevels the edges
Edge Operations

Edge Blend
Select icon or Insert > Detail Feature > Edge Blend
Select edge(s)
Key in radius

Chamfer
Select Insert > Detail Feature > Chamfer
Select edge(s)
Single offset
Key in offset
UGNX: Basic Drafting

Note: This is NOT a course in drafting, however, engineers need to be “graphically literate”; i.e., you need to be able to read and write engineering drawings.

Today’s Objective:

- Learn basic process of converting a UG solid model into an Engineering Drawing
Software Structure

UGNX is organized into:

**Applications:** Gateway, Modeling, Drafting
(and others)

When a part file is opened or created, the **Gateway** application is entered.

To create or edit a solid body the **Modeling** application is entered

To create an engineering drawing the **Drafting** application is entered
UGNX: Basic Drafting

- The Drafting application lets you create drawings with views of the part, dimensions, and necessary drafting annotations.

- This application supports the drafting of engineering models in accordance with ANSI and ISO standards.
Short Demo …

- Drafting Application
- Add Base View
  - Change view, style, orientation, scale
- Add Dimensions
- Add Annotation (text)
To Create a Drawing

1. Create part in modeling application
2. Switch to drafting application
3. Insert Sheet form select:
   - Paper size
   - Scale
   - Units
To Create a Drawing

4. Add Base View
   - Insert > View > Base View
   - MB3 on drawing border > Add Base View
   - Icon on sidebar
   - Part Nav MB3 on drawing sheet > Add Base View
To Create a Drawing

5. Choose View Options
   - Style (hidden lines)
   - View (front/iso)
   - Scale

6. Place Base View

7. Add Projected Views

8. To get an isometric, Add Base View
Adding Dimensions

- For many dimensions, just use the Inferred dimension tool
- To be more specific, pick the tool that best selects your part geometry
- Dimensions are fully associated to solid, i.e. if you change the model (in modeling) dimensions (in drafting) automatically update
Adding Notes (Annotation)

- Select Annotation Editor
  - Insert > Annotation
- Icon
- Enter Text in Edit Window
- Place Text