Spline

Hull design
SolidWorks Sketch Commands

Drawing tools
- Spline
- Offset
- Mirror
- Rectangular and Polar (circular) array

Modifying tools
- Trim and Extend
- Project entities
- Convert Entities
- Offset Entities
- Linear Sketch Pattern
- Move Entities

Dimensioning tool
- Smart Dimension
- Trim Entities
- Copy Entities
- Rotate Entities
- Scale Entities
- Stretch Entities

Checks sketch for problems
- Display/Delete Relations
- Repair Sketch
A smooth curve is fitted through selected points.
Spline on a curved surface

Select the surface to draw

Select the 3D Sketch option

Select the Spline on Surface option

Ken Yousefi

Introduction to Engineering, E10.
Draw the curve

Exit the sketch

Select **Extrude Cut** option, choose the spline curve and the direction for extrusion.
Select body to keep
SolidWorks Sketch Commands

Use **Spline** to trace the curve

Importing an image into the sketch environment
Projecting Entities onto a Sketch plane

Projected entities
SolidWorks Sketch Commands

Locates the intersection of two planes and turns it into a sketch curve

Allows you to extract a curve for other applications, **not active in sketch mode.**
Constraints

Select Add Relations

Select line

Select Vertical
Constraints

Select two entities (lines)

Select Collinear

Tangent constraint
Constraints

Two curved entities

Two straight entities

Curved & straight entities
Repair Sketch Option