Car designing using Solid Works

Solid Works

- File: new, part, and ok.
- Grid: the grid will be on the right tool bars
  - Grid/Snap: unclick snap to points
  - Annotations Display: click on Always display text at the same time.
- OK
- On the left hand side you will see the orientation, click on Plane 3
- On the top tool bars you will see blue boxes, click on normal to
- On the right tool bars click on sketch
- **Sketch Mode**
- Sketch the following using the tool bars on the right.
- Use lines, circle, spline, trim, extend and fillet.
- See page 4 of drawing for dimension and actual drawing.
On page 6 of drawing you will see dashed lines, so from page 6 highlight the dashed lines using ctrl on the keyboard and highlighting the lines. The ones that you highlighted will turn green. On the left screen click For construction. The sketch should look like that of page 6 of the drawing. On the right hand tool bar click add relation. Click 2 endpoints at a time and click on coincident until you have a close entity. Click apply and close.
• On the left hand tool bar click on Extruded Bose/Base
  • Type: Blind
  • Depth: 500mm
  • OK
  • Highlight line 1 shown on page 8 of the drawing and click Fillet on left hand toolbar
    • Radius: 1800mm
    • OK
  • Highlight line 2 shown on page 8 of the drawing and click Fillet on left hand toolbar
    • Radius: 2000mm
    • OK
  • Highlight line 3 shown on page 8 of the drawing and click Fillet on left hand toolbar
    • Radius: 50mm
    • OK
• Sketch 2 page 10 of the drawing
• Highlight the right side view of the part and click on sketch on the right hand tool bar
• Draw the following using circle and line.
• Add relation using tangent and concentric
• Extruded Cut
• Type: blind
• Depth 400mm
• OK
• Sketch 3 page 12 of the drawing
• Click the left hand side of the part and click on normal to
• Click on sketch and draw the following using line and spline
• Add relation on each end using coincident or make an extension of the line and trim it.
• Extruded Boss/Base
• Type: Blind
• Depth: 500mm
• OK
• Highlight line 1 from page 14 of the drawing
  • Fillet
  • Radius 250mm
  • OK
  • Highlight line 2 from page 14 of the drawing
  • Fillet
  • Radius 50mm
  • OK
  • Highlight line 3 from page 14 of the drawing
  • Fillet
  • Radius 250mm
  • OK
  • Highlight line 4 from page 14 of the drawing
  • Fillet
  • Radius 250mm
  • OK
• Click on the left side view of the part
• Click normal to
• Click sketch
• Draw the circles using the dimension shown on page 16 of the drawing
• Click on Extruded cut on the left hand side of the toolbar
• Type: Through All
• OK
• Using Mirror feature
• Click on mirror feature from the left hand side of the tool bar
• Mirror plane: Face<1> (Click on the left side view of the part from page 18 of the drawing, marked as dashed lines)
• Features to mirror:
• From the tree click on
• Base extrude
• Fillet1
• Fillet2
• Fillet3
• Cut-Extrude1
• Boss-Extrude1
• Fillet4
• Fillet5
• Fillet6
• Fillet7
• Cut-Extrude2
• Check the box on Geometry pattern
• OK
• Now you will have the 3d drawing of the car on page 19 of the drawing
• Tires
• File: new, part
• Click: Plane3
• Click: normal to
• Click: Sketch
• Click: Circle and draw a circle shown on page 21
• Click: Extrude-Base/Base
• Type: Blind
• Depth: 355.60mm
• OK
- Click the right side of the part
- Click: Sketch
- Click: Circle and dimension it with 457.20mm diameter
- Click: Add Relation
- Click: concentric, apply
- Click: Extruded-cut
- Type: Blind
- Depth: 254mm
  - OK
- Highlight the inner part of the tire on the right side view of the part
- Click: sketch
- Click: circle and dimension it with 127mm diameter
- Click: Add Relation, concentric, apply
- Click: Extruded Boss/Base
- Type: Blind
- Depth: 127mm
  - OK
- Highlight the face of the left side view of the part
- Click: normal to
- Click: sketch
- Click: circle and dimension it with 127mm diameter
- Click: Add Relation, concentric, apply
- Click: Extruded Boss/Base
- Type: Blind
- Depth: 508mm
  - OK
  - Save all the files
• Making an Assembly:
  • File: New, Assembly, OK
  • Window: Tile vertically
  • On the sub tree of each part
  • Click and drag part 1 to assembly 1
  • Click and drag part 2 (tire) to assembly 1 4 times shown on page 25 of the drawing

  • Click: Mate (on the left hand side of the tool bar of the Assembly screen)
  • Minimize part 1 and part 2
  • Click: the shaft of the tire
  • Click: the extruded cut on the body of the car
  • Click: concentric
  • OK
  • Do the same thing for the rest of the tires

  • Click: Mate (on the left hand side of the tool bar of the Assembly screen)
  • See page 26 of the drawing (highlighted with a dashed lines)
  • Click: the sidewall of the tire
  • Click: the side panel of the car
  • Click: Coincident
  • OK
  • Do the same thing for the rest of the tires

  • Then you’re done with the whole assembly