Rebar Addon for FreeCAD

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Contents

Chapter 1

Rebar Addon for FreeCAD

Started as a Google Summer of Code (GSoC 2017) project.

Documentation

This project is aimed at easing up the process of rebaring in FreeCAD. In this project, list of rebars will be provided to user under Rebar tools in the form of dropdown. This project covers six different rebar shapes as given below:

- **Straight Rebar**: wiki
- **UShape Rebar**: wiki
- **LShape Rebar**: wiki
- **BentShpae Rebar**: wiki
- **Stirrup Rebar**: wiki
- **Helical Rebar**: wiki

Video Tutorial

Installation

Pre-requisites

- FreeCAD (version >= 0.17): Installation guide

Steps to install Rebar Addon in FreeCAD

1. Open the FreeCAD Addon Manager (`Tool -> Addon manager`).
2. When an addon manager will open, select `Reinforcement` from a list of workbenches shown by an addon manager.
3. After selecting, click on `Install/Update` button.
4. Restart FreeCAD.
5. Now you will see different rebars in a drop-down list of rebar tools (`Arch -> Rebar tools -> Different rebars`).

How it works

Each rebar tool has two files, one is `Python` file and second is there respective name `UI` file like `StraightRebar.py` and `StraightRebar.ui` file). Let's take a straight rebar tool. In `StraightRebar.py` file, there are two functions. One is `makeStraightRebar()` function. This function creates straight rebar and adds new properties to the default `Rebar` object. Second function is `editStraightRebar`. This function is used when we want to change a new properties(which is created by `makeStraightRebar` function) of the rebar object and it will take `Rebar` object as input which is created by `makeStraightRebar` function. In `StraightRebar.py`, `_StraightRebarTaskPanel` class is present. This class loads UI(present in `StriaghtRebar.ui` file) in the task panel of FreeCAD. First time when a user clicks on `Apply` or `Ok` button, then `makeStraightRebar` function is executed and after that when user want to change the properties of Straight rebar then `editStraightRebar` function is excuted.

Extras

- `FreeCAD forum thread`
- `GSoC proposal`
- `Development logs`

Chapter 2

Namespace Index

2.1 Namespace List

Here is a list of all namespaces with brief descriptions:

Chapter 3

Hierarchical Index

3.1 Class Hierarchy

This inheritance list is sorted roughly, but not completely, alphabetically:

Chapter 4

Class Index

4.1 Class List

Here are the classes, structs, unions and interfaces with brief descriptions:

Chapter 5

File Index

5.1 File List

Here is a list of all files with brief descriptions:

Chapter 6

Namespace Documentation

6.1 BentShapeRebar Namespace Reference

Classes

- class _BentShapeRebarTaskPanel

Functions

- def getpointsOfBentShapeRebar (FacePRM, l_cover, r_cover, b_cover, t_cover, bentLength, bentAngle, orientation)
- def makeBentShapeRebar (f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation="Bottom Left", structure=None, facename=None)
- def editBentShapeRebar (Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def editDialog (vobj)
- def CommandBentShapeRebar ()

Variables

- string __title__ = "BentShapeRebar"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

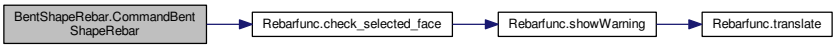
6.1.1 Function Documentation

6.1.1.1 def BentShapeRebar.CommandBentShapeRebar ( )

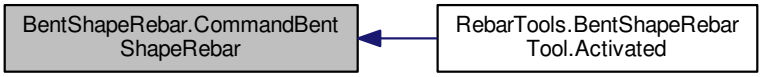
Definition at line 359 of file BentShapeRebar.py.

```
00359 def CommandBentShapeRebar():
00360     selected_obj = check_selected_face()
00361     if selected_obj:
00362         FreeCADGui.Control.showDialog(_BentShapeRebarTaskPanel())
00363
```

Here is the call graph for this function:



Here is the caller graph for this function:



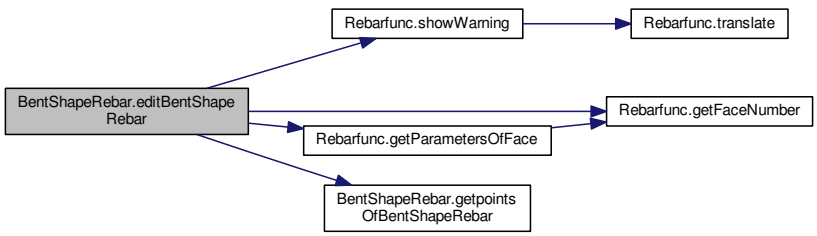
6.1.1.2 def BentShapeRebar.editBentShapeRebar (*Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation, structure =* `None`, *facename =* `None`)

Definition at line 270 of file BentShapeRebar.py.

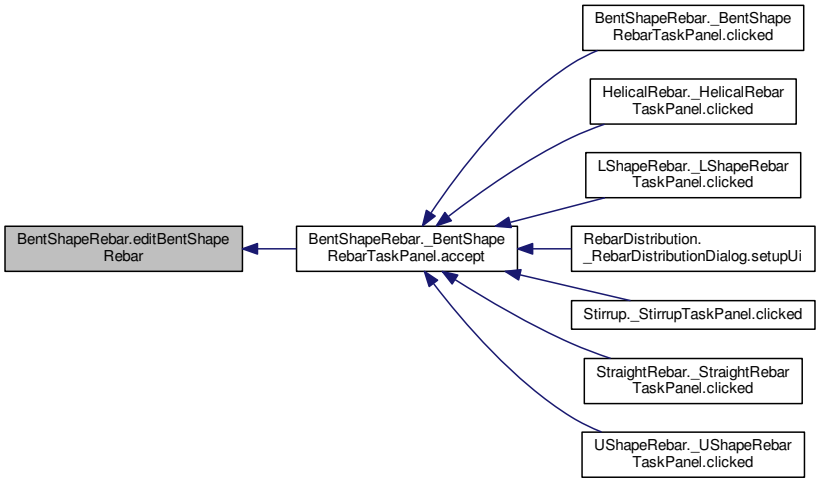
```
00270 def editBentShapeRebar(Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover,
      bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation, structure = None,
      facename = None):
00271     sketch = Rebar.Base
00272     if structure and facename:
00273         sketch.Support = [(structure, facename)]
00274     # Check if sketch support is empty.
00275     if not sketch.Support:
00276         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00277         return
00278     # Assigned values
00279     facename = sketch.Support[0][1][0]
00280     structure = sketch.Support[0][0]
00281     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00282     #StructurePRM = getTrueParametersOfStructure(structure)
00283     # Get parameters of the face where sketch of rebar is drawn
00284     FacePRM = getParametersOfFace(structure, facename)
00285     # Get points of L-Shape rebar
00286     points = getpointsOfBentShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover
      , bentLength, bentAngle, orientation)
00287     sketch.movePoint(0, 1, points[0], 0)
00288     FreeCAD.ActiveDocument.recompute()
00289     sketch.movePoint(0, 2, points[1], 0)
00290     FreeCAD.ActiveDocument.recompute()
00291     sketch.movePoint(1, 1, points[1], 0)
00292     FreeCAD.ActiveDocument.recompute()
00293     sketch.movePoint(1, 2, points[2], 0)
00294     FreeCAD.ActiveDocument.recompute()
00295
00296     sketch.movePoint(2, 1, points[2], 0)
00297     FreeCAD.ActiveDocument.recompute()
00298     sketch.movePoint(2, 2, points[3], 0)
00299     FreeCAD.ActiveDocument.recompute()
```

```
00300     sketch.movePoint(3, 1, points[3], 0)
00301     FreeCAD.ActiveDocument.recompute()
00302     sketch.movePoint(3, 2, points[4], 0)
00303     FreeCAD.ActiveDocument.recompute()
00304
00305     sketch.movePoint(4, 1, points[4], 0)
00306     FreeCAD.ActiveDocument.recompute()
00307     sketch.movePoint(4, 2, points[5], 0)
00308     FreeCAD.ActiveDocument.recompute()
00309
00310     Rebar.OffsetStart = f_cover
00311     Rebar.OffsetEnd = f_cover
00312     if amount_spacing_check:
00313         Rebar.Amount = amount_spacing_value
00314         FreeCAD.ActiveDocument.recompute()
00315         Rebar.AmountCheck = True
00316     else:
00317         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00318         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00319         FreeCAD.ActiveDocument.recompute()
00320         Rebar.AmountCheck = False
00321     Rebar.Diameter = diameter
00322     Rebar.FrontCover = f_cover
00323     Rebar.LeftCover = l_cover
00324     Rebar.RightCover = r_cover
00325     Rebar.BottomCover = b_cover
00326     Rebar.TopCover = t_cover
00327     Rebar.BentLength = bentLength
00328     Rebar.BentAngle = bentAngle
00329     Rebar.Rounding = rounding
00330     Rebar.TrueSpacing = amount_spacing_value
00331     Rebar.Orientation = orientation
00332     FreeCAD.ActiveDocument.recompute()
00333     return Rebar
00334
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.1.1.3 def BentShapeRebar.editDialog (*vobj*)

Definition at line 335 of file BentShapeRebar.py.

```
00335 def editDialog(vobj):
00336     FreeCADGui.Control.closeDialog()
00337     obj = _BentShapeRebarTaskPanel(vobj.Object)
00338     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00339     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00340     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00341     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00342     obj.form.diameter.setText(str(vobj.Object.Diameter))
00343     obj.form.topCover.setText(str(vobj.Object.TopCover))
00344     obj.form.bentLength.setText(str(vobj.Object.BentLength))
00345     obj.form.bentAngle.setValue(vobj.Object.BentAngle)
00346     obj.form.rounding.setValue(vobj.Object.Rounding)
00347     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00348     if vobj.Object.AmountCheck:
00349         obj.form.amount.setValue(vobj.Object.Amount)
00350     else:
00351         obj.form.amount_radio.setChecked(False)
00352         obj.form.spacing_radio.setChecked(True)
00353         obj.form.amount.setDisabled(True)
00354         obj.form.spacing.setEnabled(True)
00355         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00356     #obj.form.PickSelectedFace.setVisible(False)
00357     FreeCADGui.Control.showDialog(obj)
00358
```

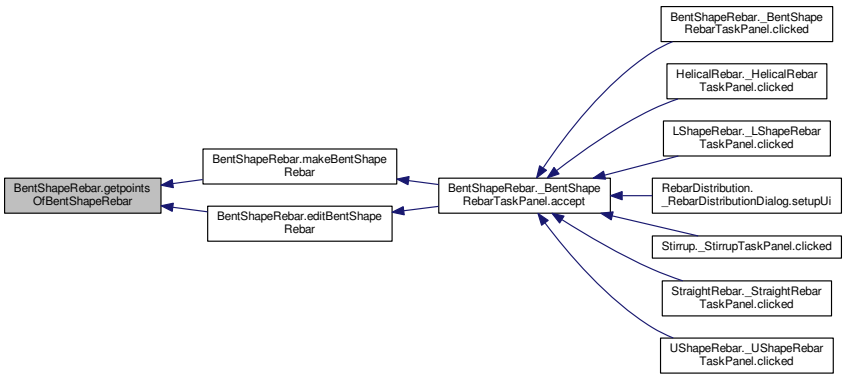
6.1.1.4 def BentShapeRebar.getpointsOfBentShapeRebar (*FacePRM, l_cover, r_cover, b_cover, t_cover, bentLength, bentAngle, orientation*)

```
getpointsOfBentShapeRebar(FacePRM, LeftCover, RightCover, BottomCover, TopCover, BentLength, BentAngle, Orient
Return points of the LShape rebar in the form of array for sketch.
It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
```

Definition at line 40 of file BentShapeRebar.py.

```
00040 def getpointsOfBentShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover,
      bentLength, bentAngle, orientation):
00041     """ getpointsOfBentShapeRebar(FacePRM, LeftCover, RightCover, BottomCover, TopCover, BentLength,
       BentAngle, Orientation):
00042     Return points of the LShape rebar in the form of array for sketch.
00043     It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
00044     """
00045     if orientation == "Bottom":
00046         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00047         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00048         x2 = x1 + bentLength
00049         y2 = y1
00050         dis = (FacePRM[0][1] - t_cover - b_cover) * math.tan(math.radians(bentAngle - 90))
00051         x3 = x2 + dis
00052         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00053         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover - bentLength - dis
00054         y4 = y3
00055         x5 = x4 + dis
00056         y5 = y2
00057         x6 = x5 + bentLength
00058         y6 = y5
00059     elif orientation == "Top":
00060         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00061         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00062         x2 = x1 + bentLength
00063         y2 = y1
00064         dis = (FacePRM[0][1] - t_cover - b_cover) * math.tan(math.radians(bentAngle - 90))
00065         x3 = x2 + dis
00066         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00067         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover - bentLength - dis
00068         y4 = y3
00069         x5 = x4 + dis
00070         y5 = y2
00071         x6 = x5 + bentLength
00072         y6 = y5
00073     elif orientation == "Left":
00074         x1 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00075         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00076         x2 = x1
00077         y2 = y1 - bentLength
00078         dis = (FacePRM[0][0] - r_cover - l_cover) * math.tan(math.radians(bentAngle - 90))
00079         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00080         y3 = y2 - dis
00081         x4 = x3
00082         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover + bentLength + dis
00083         x5 = x2
00084         y5 = y4 - dis
00085         x6 = x5
00086         y6 = y5 - bentLength
00087     elif orientation == "Right":
00088         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00089         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00090         x2 = x1
00091         y2 = y1 - bentLength
00092         dis = (FacePRM[0][0] - r_cover - l_cover) * math.tan(math.radians(bentAngle - 90))
00093         x3 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00094         y3 = y2 - dis
00095         x4 = x3
00096         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover + bentLength + dis
00097         x5 = x2
00098         y5 = y4 - dis
00099         x6 = x5
00100         y6 = y5 - bentLength
00101     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0),\
00102             FreeCAD.Vector(x3, y3, 0), FreeCAD.Vector(x4, y4, 0),\
00103             FreeCAD.Vector(x5, y5, 0), FreeCAD.Vector(x6, y6, 0)]
00104
```

Here is the caller graph for this function:



6.1.1.5 def BentShapeRebar.makeBentShapeRebar (*f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation =* "Bottom Left", *structure =* None, *facename =* None)

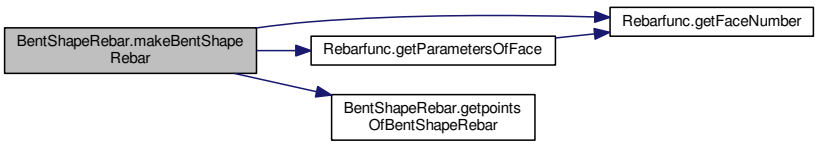
```
makeBentShapeRebar(FrontCover, BottomCover, LeftCover, RightCover, Diameter, TopCover, BentLength, BentAngle,
AmountSpacingCheck, AmountSpacingValue, Orientation, Structure, Facename): Adds the Bent-Shape reinforcement b
selected structural object.
It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
```

Definition at line 200 of file BentShapeRebar.py.

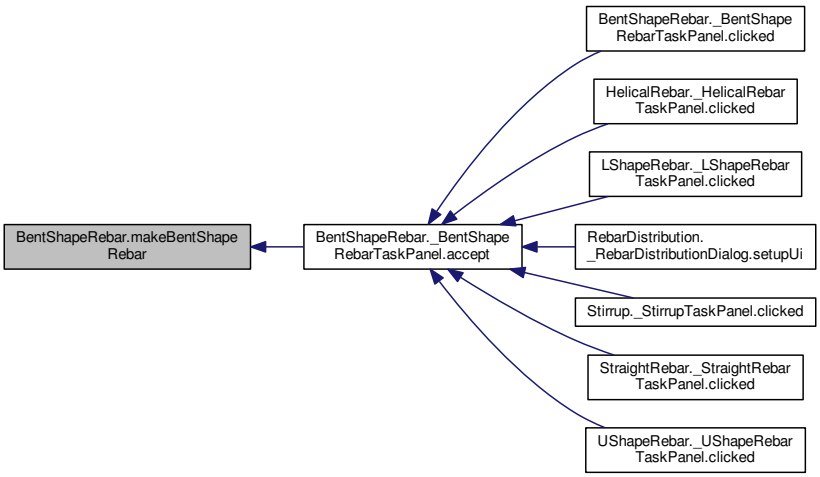
```
00200 def makeBentShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength,
      bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation = "Bottom Left", structure =
      None, facename = None):
00201     """ makeBentShapeRebar(FrontCover, BottomCover, LeftCover, RightCover, Diameter, TopCover, BentLength,
      BentAngle, Rounding,
00202     AmountSpacingCheck, AmountSpacingValue, Orientation, Structure, Facename): Adds the Bent-Shape
      reinforcement bar to the
00203     selected structural object.
00204     It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
00205     """
00206     if not structure and not facename:
00207         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00208         structure = selected_obj.Object
00209         facename = selected_obj.SubElementNames[0]
00210     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00211     #StructurePRM = getTrueParametersOfStructure(structure)
00212     FacePRM = getParametersOfFace(structure, facename)
00213     if not FacePRM:
00214         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
      derived\n")
00215         return
00216     # Get points of L-Shape rebar
00217     points = getpointsOfBentShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover
      , bentLength, bentAngle, orientation)
00218     import Part
00219     import Arch
00220     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00221     sketch.MapMode = "FlatFace"
00222     sketch.Support = [(structure, facename)]
00223     FreeCAD.ActiveDocument.recompute()
00224     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00225     sketch.addGeometry(Part.LineSegment(points[1], points[2]), False)
00226     sketch.addGeometry(Part.LineSegment(points[2], points[3]), False)
00227     sketch.addGeometry(Part.LineSegment(points[3], points[4]), False)
00228     sketch.addGeometry(Part.LineSegment(points[4], points[5]), False)
00229     import Sketcher
00230     if amount_spacing_check:
00231         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
```

```
00232         FreeCAD.ActiveDocument.recompute()
00233     else:
00234         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00235         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
    f_cover)
00236     rebar.Rounding = rounding
00237     # Adds properties to the rebar object
00238     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Shape of rebar")).RebarShape = "BentShapeRebar"
00239     rebar.ViewObject.setEditorMode("RebarShape", 2)
00240     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Front cover of rebar")).FrontCover = f_cover
00241     rebar.setEditorMode("FrontCover", 2)
00242     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
    ", "Left Side cover of rebar")).LeftCover = l_cover
00243     rebar.setEditorMode("LeftCover", 2)
00244     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Right Side cover of rebar")).RightCover = r_cover
00245     rebar.setEditorMode("RightCover", 2)
00246     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00247     rebar.setEditorMode("BottomCover", 2)
00248     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
     "Amount radio button is checked")).AmountCheck
00249     rebar.setEditorMode("AmountCheck", 2)
00250     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
    , "Top cover of rebar")).TopCover = t_cover
00251     rebar.setEditorMode("TopCover", 2)
00252     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00253     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
    ", "Shape of rebar")).Orientation = orientation
00254     rebar.setEditorMode("Orientation", 2)
00255     rebar.setEditorMode("TrueSpacing", 2)
00256     rebar.addProperty("App::PropertyDistance", "BentLength", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "BentLength cover of rebar")).BentLength = bentLength
00257     rebar.setEditorMode("BentLength", 2)
00258     rebar.addProperty("App::PropertyDistance", "BentAngle", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
    ", "Bent Angle of rebar")).BentAngle = bentAngle
00259     rebar.setEditorMode("BentAngle", 2)
00260
00261     if amount_spacing_check:
00262         rebar.AmountCheck = True
00263     else:
00264         rebar.AmountCheck = False
00265         rebar.TrueSpacing = amount_spacing_value
00266     rebar.Label = "BentShapeRebar"
00267     FreeCAD.ActiveDocument.recompute()
00268     return rebar
00269
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.1.2 Variable Documentation

6.1.2.1 string BentShapeRebar.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file BentShapeRebar.py.

6.1.2.2 string BentShapeRebar.__title__ = "BentShapeRebar" [private]

Definition at line 24 of file BentShapeRebar.py.

6.1.2.3 string BentShapeRebar.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file BentShapeRebar.py.

6.2 HelicalRebar Namespace Reference

Classes

- class _HelicalRebarTaskPanel

Functions

- def getpointsOfHelicalRebar (FacePRM, s_cover, b_cover, t_cover, pitch, edges, diameter, size, direction)
- def createHelicalWire (FacePRM, s_cover, b_cover, t_cover, pitch, size, direction, helix=None)
- def makeHelicalRebar (s_cover, b_cover, diameter, t_cover, pitch, structure=None, facename=None)
- def editHelicalRebar (Rebar, s_cover, b_cover, diameter, t_cover, pitch, structure=None, facename=None)
- def editDialog (vobj)
- def CommandHelicalRebar ()

Variables

- string __title__ = "HelicalRebar"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

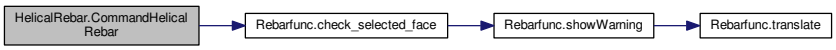
6.2.1 Function Documentation

6.2.1.1 def HelicalRebar.CommandHelicalRebar ( )

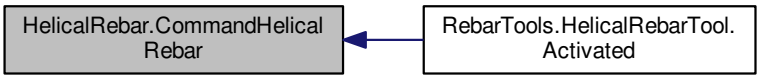
Definition at line 241 of file HelicalRebar.py.

```
00241 def CommandHelicalRebar():
00242     selected_obj = check_selected_face()
00243     if selected_obj:
00244         FreeCADGui.Control.showDialog(_HelicalRebarTaskPanel())
00245
```

Here is the call graph for this function:



Here is the caller graph for this function:



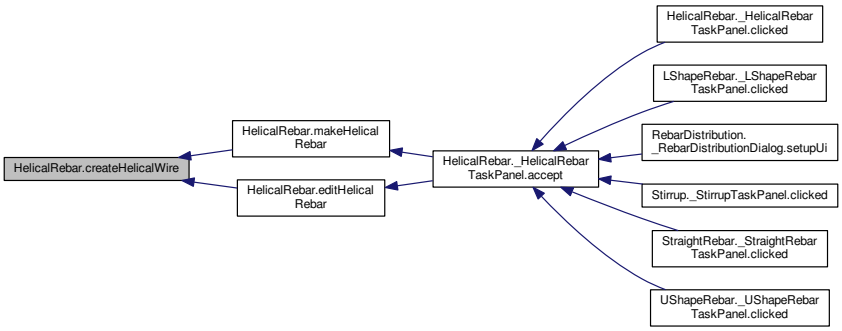
6.2.1.2 def HelicalRebar.createHelicalWire (*FacePRM, s_cover, b_cover, t_cover, pitch, size, direction, helix* = None)

```
createHelicalWire(FacePRM, SideCover, BottomCover, TopCover, Pitch, Size, Direction, Helix = None):
It creates a helical wire.
```

Definition at line 75 of file HelicalRebar.py.

```
00075 def createHelicalWire(FacePRM, s_cover, b_cover, t_cover, pitch, size, direction, helix =
      None):
00076     """ createHelicalWire(FacePRM, SideCover, BottomCover, TopCover, Pitch, Size, Direction, Helix = None):
00077     It creates a helical wire."""
00078     import Part
00079     if not helix:
00080         helix = FreeCAD.ActiveDocument.addObject("Part::Helix","Helix")
00081     helix.Pitch = pitch
00082     helix.Radius = FacePRM[0][0] / 2 - s_cover
00083     helix.Angle = 0
00084     helix.LocalCoord = 0
00085     helix.Height = size - b_cover - t_cover
00086     if round(direction.x) == 1:
00087         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0] - b_cover, FacePRM[1][1], FacePRM[1][2])
00088         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, -1, 0), 90)
00089     elif round(direction.x) == -1:
00090         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0] + t_cover, FacePRM[1][1], FacePRM[1][2])
00091         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, -1, 0), -90)
00092     elif round(direction.y) == 1:
00093         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1] - b_cover, FacePRM[1][2])
00094         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(1, 0, 0), 90)
00095     elif round(direction.y) == -1:
00096         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1] + t_cover, FacePRM[1][2])
00097         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(-1, 0, 0), 90)
00098     elif round(direction.z) == 1:
00099         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1], FacePRM[1][2] - size + b_cover)
00100         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, 0, 1), 0)
00101     elif round(direction.z) == -1:
00102         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1], FacePRM[1][2] + b_cover)
00103         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, 0, -1), 0)
00104     FreeCAD.ActiveDocument.recompute()
00105     return helix
00106
```

Here is the caller graph for this function:



6.2.1.3 def HelicalRebar.editDialog (*vobj*)

Definition at line 231 of file HelicalRebar.py.

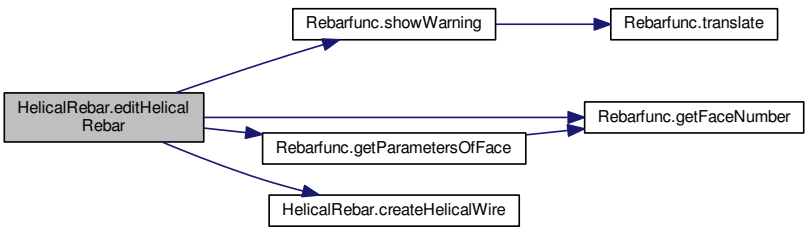
```
00231 def editDialog(vobj):
00232     FreeCADGui.Control.closeDialog()
00233     obj = _HelicalRebarTaskPanel(vobj.Object)
00234     obj.form.sideCover.setText(str(vobj.Object.SideCover))
00235     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00236     obj.form.diameter.setText(str(vobj.Object.Diameter))
00237     obj.form.topCover.setText(str(vobj.Object.TopCover))
00238     obj.form.pitch.setText(str(vobj.Object.Pitch))
00239     FreeCADGui.Control.showDialog(obj)
00240
```

6.2.1.4 def HelicalRebar.editHelicalRebar (*Rebar, s_cover, b_cover, diameter, t_cover, pitch, structure* = None, *facename* = None)

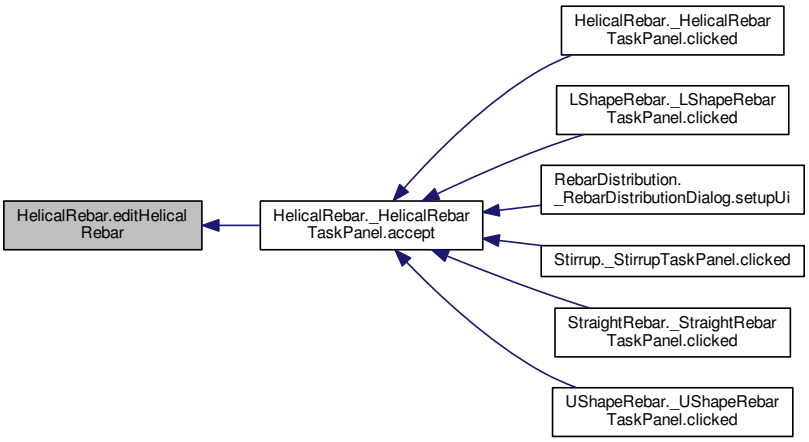
Definition at line 203 of file HelicalRebar.py.

```
00203 def editHelicalRebar(Rebar, s_cover, b_cover, diameter, t_cover, pitch, structure = None,
      facename = None):
00204     sketch = Rebar.Base
00205     if structure and facename:
00206         sketch.Support = [(structure, facename)]
00207     # Check if sketch support is empty.
00208     if not sketch.Support:
00209         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00210         return
00211     # Assigned values
00212     facename = sketch.Support[0][1][0]
00213     structure = sketch.Support[0][0]
00214     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00215     #StructurePRM = getTrueParametersOfStructure(structure)
00216     # Get parameters of the face where sketch of rebar is drawn
00217     FacePRM = getParametersOfFace(structure, facename, False)
00218     size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00219     normal = face.normalAt(0,0)
00220     #normal = face.Placement.Rotation.inverted().multVec(normal)
00221     helix = createHelicalWire(FacePRM, s_cover, b_cover, t_cover, pitch, size, normal,
      Rebar.Base)
00222     FreeCAD.ActiveDocument.recompute()
00223     Rebar.Diameter = diameter
00224     Rebar.SideCover = s_cover
00225     Rebar.BottomCover = b_cover
00226     Rebar.TopCover = t_cover
00227     Rebar.Pitch = pitch
00228     FreeCAD.ActiveDocument.recompute()
00229     return Rebar
00230
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.2.1.5 def HelicalRebar.getpointsOfHelicalRebar (*FacePRM, s_cover, b_cover, t_cover, pitch, edges, diameter, size, direction*)

```
getpointsOfHelicalRebar(FacePRM, s_cover, b_cover, t_cover):
Return points of the LShape rebar in the form of array for sketch.
```

Definition at line 39 of file HelicalRebar.py.

```
00039 def getpointsOfHelicalRebar(FacePRM, s_cover, b_cover, t_cover, pitch, edges,
      diameter, size, direction):
00040     """ getpointsOfHelicalRebar(FacePRM, s_cover, b_cover, t_cover):
00041     Return points of the LShape rebar in the form of array for sketch."""
00042     dx = s_cover + diameter / 2
00043     dz = float(pitch) / edges
00044     R = diameter / 2 - dx
00045     R = FacePRM[0][0] / 2 - s_cover
00046     points = []
00047     if direction[2] in {-1,1}:
00048         z = 0
00049         l = 0
00050         if direction[2] == 1:
00051             zz = FacePRM[1][2] - t_cover
00052         elif direction[2] == -1:
00053             zz = FacePRM[1][2] + b_cover
00054         count = 0
00055         flag = False
00056         while (round(z) < abs(size - b_cover - t_cover)):
00057             for i in range(0, int(edges) + 1):
00058                 if not i and flag:
00059                     continue
00060                 if not flag:
00061                     z -= dz
00062                     flag = True
00063                 iAngle = i * 360 / edges
00064                 x =  FacePRM[1][0] + R * math.cos(math.radians(iAngle))
00065                 y =  FacePRM[1][1] + R * math.sin(math.radians(iAngle))
00066                 points.append(FreeCAD.Vector(x, y, zz))
00067                 count += 1
00068                 if direction[2] == 1:
00069                     zz -= dz
00070                 elif direction[2] == -1:
00071                     zz += dz
00072                 z += dz
00073     return points
00074
```

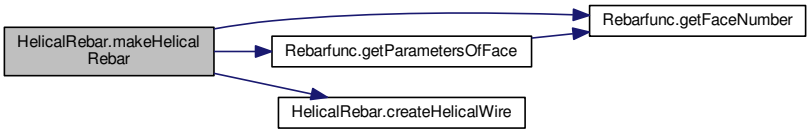
6.2.1.6 def HelicalRebar.makeHelicalRebar (*s_cover, b_cover, diameter, t_cover, pitch, structure* = None, *facename* = None)

```
makeHelicalRebar(SideCover, BottomCover, Diameter, TopCover, Pitch, Structure, Facename):
Adds the Helical reinforcement bar to the selected structural object.
```

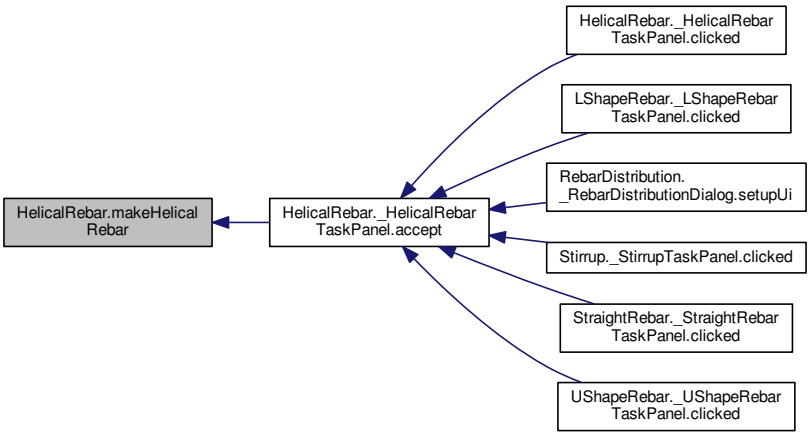
Definition at line 165 of file HelicalRebar.py.

```
00165 def makeHelicalRebar(s_cover, b_cover, diameter, t_cover, pitch, structure = None, facename
 = None):
00166     """ makeHelicalRebar(SideCover, BottomCover, Diameter, TopCover, Pitch, Structure, Facename):
00167     Adds the Helical reinforcement bar to the selected structural object."""
00168     if not structure and not facename:
00169         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00170         structure = selected_obj.Object
00171         facename = selected_obj.SubElementNames[0]
00172     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00173     #StructurePRM = getTrueParametersOfStructure(structure)
00174     FacePRM = getParametersOfFace(structure, facename, False)
00175     if not FacePRM:
00176         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
 derived\n")
00177         return
00178     size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00179     normal = face.normalAt(0,0)
00180     #normal = face.Placement.Rotation.inverted().multVec(normal)
00181     import Arch
00182     helix = createHelicalWire(FacePRM, s_cover, b_cover, t_cover, pitch, size, normal)
00183     helix.Support = [(structure, facename)]
00184     rebar = Arch.makeRebar(structure, helix, diameter, 1, 0)
00185     rebar.OffsetStart = 0
00186     rebar.OffsetEnd = 0
00187     FreeCAD.ActiveDocument.recompute()
00188     # Adds properties to the rebar object
00189     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Shape of rebar")).RebarShape = "HelicalRebar"
00190     rebar.ViewObject.setEditorMode("RebarShape", 2)
00191     rebar.addProperty("App::PropertyDistance", "SideCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
", "Front cover of rebar")).SideCover = s_cover
00192     rebar.setEditorMode("SideCover", 2)
00193     rebar.addProperty("App::PropertyDistance", "Pitch", "RebarDialog", QT_TRANSLATE_NOOP("App::Property", "
Left Side cover of rebar")).Pitch = pitch
00194     rebar.setEditorMode("Pitch", 2)
00195     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00196     rebar.setEditorMode("BottomCover", 2)
00197     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
, "Top cover of rebar")).TopCover = t_cover
00198     rebar.setEditorMode("TopCover", 2)
00199     rebar.Label = "HelicalRebar"
00200     FreeCAD.ActiveDocument.recompute()
00201     return rebar
00202
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.2.2 Variable Documentation

6.2.2.1 string HelicalRebar.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file HelicalRebar.py.

6.2.2.2 string HelicalRebar.__title__ = "HelicalRebar" [private]

Definition at line 24 of file HelicalRebar.py.

6.2.2.3 string HelicalRebar.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file HelicalRebar.py.

6.3 LShapeRebar Namespace Reference

Classes

- class _LShapeRebarTaskPanel

Functions

- def getpointsOfLShapeRebar (FacePRM, l_cover, r_cover, b_cover, t_cover, orientation)
- def makeLShapeRebar (f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding, amount_spacing_↩ check, amount_spacing_value, orientation="Bottom Left", structure=None, facename=None)
- def editLShapeRebar (Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding, amount_↩ spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def editDialog (vobj)
- def CommandLShapeRebar ()

Variables

- string __title__ = "LShapeRebar"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

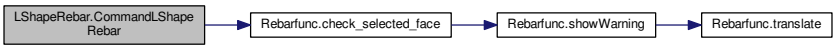
6.3.1 Function Documentation

6.3.1.1 def LShapeRebar.CommandLShapeRebar ( )

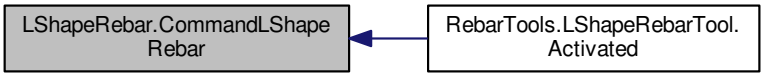
Definition at line 300 of file LShapeRebar.py.

```
00300 def CommandLShapeRebar():
00301     selected_obj = check_selected_face()
00302     if selected_obj:
00303         FreeCADGui.Control.showDialog(_LShapeRebarTaskPanel())
00304
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.3.1.2 def LShapeRebar.editDialog (*vobj*)

Definition at line 278 of file LShapeRebar.py.

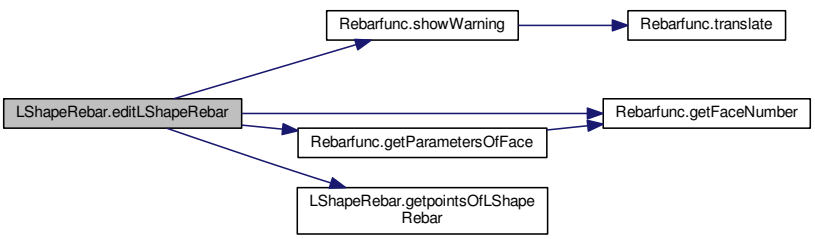
```
00278 def editDialog(vobj):
00279     FreeCADGui.Control.closeDialog()
00280     obj = _LShapeRebarTaskPanel(vobj.Object)
00281     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00282     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00283     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00284     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00285     obj.form.diameter.setText(str(vobj.Object.Diameter))
00286     obj.form.topCover.setText(str(vobj.Object.TopCover))
00287     obj.form.rounding.setValue(vobj.Object.Rounding)
00288     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00289     if vobj.Object.AmountCheck:
00290         obj.form.amount.setValue(vobj.Object.Amount)
00291     else:
00292         obj.form.amount_radio.setChecked(False)
00293         obj.form.spacing_radio.setChecked(True)
00294         obj.form.amount.setDisabled(True)
00295         obj.form.spacing.setEnabled(True)
00296         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00297     #obj.form.PickSelectedFace.setVisible(False)
00298     FreeCADGui.Control.showDialog(obj)
00299
```

6.3.1.3 def LShapeRebar.editLShapeRebar (*Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation, structure* = None, *facename* = None)

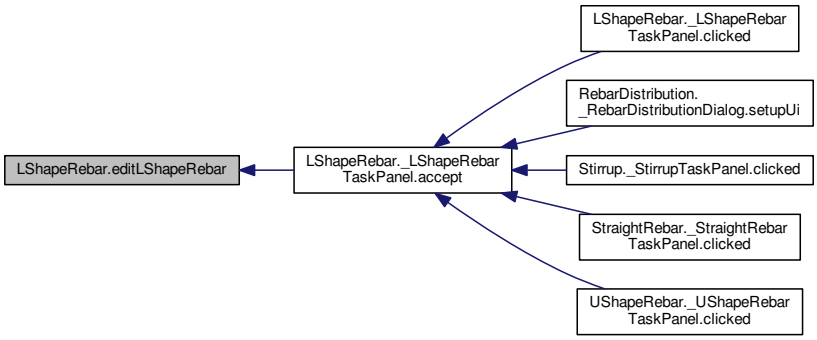
Definition at line 230 of file LShapeRebar.py.

```
00230 def editLShapeRebar(Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding,
      amount_spacing_check, amount_spacing_value, orientation, structure = None, facename = None):
00231     sketch = Rebar.Base
00232     if structure and facename:
00233         sketch.Support = [(structure, facename)]
00234     # Check if sketch support is empty.
00235     if not sketch.Support:
00236         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00237         return
00238     # Assigned values
00239     facename = sketch.Support[0][1][0]
00240     structure = sketch.Support[0][0]
00241     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00242     #StructurePRM = getTrueParametersOfStructure(structure)
00243     # Get parameters of the face where sketch of rebar is drawn
00244     FacePRM = getParametersOfFace(structure, facename)
00245     # Get points of L-Shape rebar
00246     points = getpointsOfLShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover,
      orientation)
00247     sketch.movePoint(0, 1, points[0], 0)
00248     FreeCAD.ActiveDocument.recompute()
00249     sketch.movePoint(0, 2, points[1], 0)
00250     FreeCAD.ActiveDocument.recompute()
00251     sketch.movePoint(1, 1, points[1], 0)
00252     FreeCAD.ActiveDocument.recompute()
00253     sketch.movePoint(1, 2, points[2], 0)
00254     FreeCAD.ActiveDocument.recompute()
00255     Rebar.OffsetStart = f_cover
00256     Rebar.OffsetEnd = f_cover
00257     if amount_spacing_check:
00258         Rebar.Amount = amount_spacing_value
00259         FreeCAD.ActiveDocument.recompute()
00260         Rebar.AmountCheck = True
00261     else:
00262         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00263         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00264         FreeCAD.ActiveDocument.recompute()
00265         Rebar.AmountCheck = False
00266     Rebar.Diameter = diameter
00267     Rebar.FrontCover = f_cover
00268     Rebar.LeftCover = l_cover
00269     Rebar.RightCover = r_cover
00270     Rebar.BottomCover = b_cover
00271     Rebar.TopCover = t_cover
00272     Rebar.Rounding = rounding
00273     Rebar.TrueSpacing = amount_spacing_value
00274     Rebar.Orientation = orientation
00275     FreeCAD.ActiveDocument.recompute()
00276     return Rebar
00277
```

Here is the call graph for this function:



Here is the caller graph for this function:



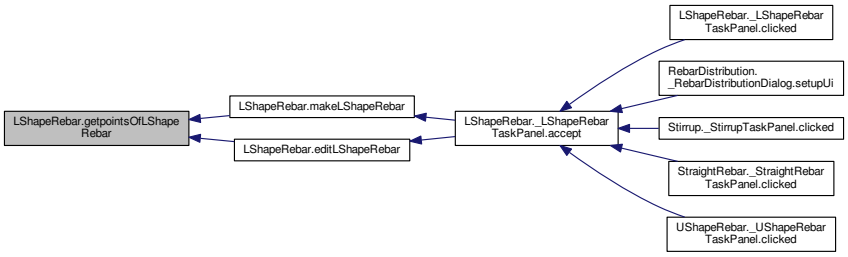
6.3.1.4 def LShapeRebar.getpointsOfLShapeRebar (*FacePRM, l_cover, r_cover, b_cover, t_cover, orientation*)

```
getpointsOfLShapeRebar(FacePRM, LeftCover, RightCover, BottomCover, TopCover, Orientation):
Return points of the LShape rebar in the form of array for sketch.
It takes four different orientations input i.e. 'Bottom Left', 'Bottom Right ', 'Top Left', 'Top Right'.
```

Definition at line 40 of file LShapeRebar.py.

```
00040 def getpointsOfLShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover, orientation):
00041     """ getpointsOfLShapeRebar(FacePRM, LeftCover, RightCover, BottomCover, TopCover, Orientation):
00042     Return points of the LShape rebar in the form of array for sketch.
00043     It takes four different orientations input i.e. 'Bottom Left', 'Bottom Right ', 'Top Left', 'Top Right'
      .
00044     """
00045     if orientation == "Bottom Left":
00046         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00047         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00048         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00049         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00050         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00051         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00052     elif orientation == "Bottom Right":
00053         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00054         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00055         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00056         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00057         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00058         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00059     elif orientation == "Top Left":
00060         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00061         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00062         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00063         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00064         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00065         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00066     elif orientation == "Top Right":
00067         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00068         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00069         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00070         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00071         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00072         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00073     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0),\
00074             FreeCAD.Vector(x3, y3, 0)]
00075
```

Here is the caller graph for this function:



6.3.1.5 def LShapeRebar.makeLShapeRebar (*f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation* = "Bottom Left", *structure* = None, *facename* = None)

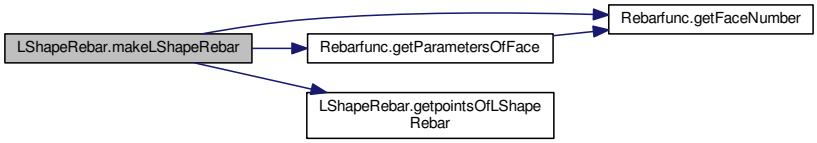
```
makeLShapeRebar(FrontCover, BottomCover, LeftCover, RightCover, Diameter, TopCover, Rounding, AmountSpacingChe
Orientation, Structure, Facename): Adds the L-Shape reinforcement bar to the selected structural object.
It takes four different orientations input i.e. 'Bottom Left', 'Bottom Right ', 'Top Left', 'Top Right'.
```

Definition at line 169 of file LShapeRebar.py.

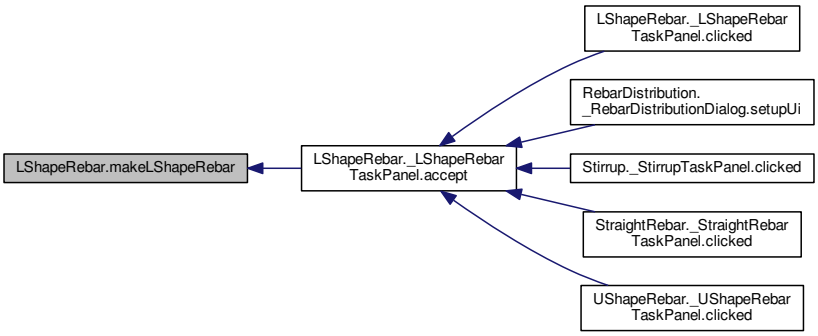
```
00169 def makeLShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding,
      amount_spacing_check, amount_spacing_value, orientation = "Bottom Left", structure = None, facename = None):
00170     """ makeLShapeRebar(FrontCover, BottomCover, LeftCover, RightCover, Diameter, TopCover, Rounding,
       AmountSpacingCheck, AmountSpacingValue,
00171     Orientation, Structure, Facename): Adds the L-Shape reinforcement bar to the selected structural
       object.
00172     It takes four different orientations input i.e. 'Bottom Left', 'Bottom Right ', 'Top Left', 'Top Right'
      .
00173     """
00174     if not structure and not facename:
00175         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00176         structure = selected_obj.Object
00177         facename = selected_obj.SubElementNames[0]
00178     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00179     #StructurePRM = getTrueParametersOfStructure(structure)
00180     FacePRM = getParametersOfFace(structure, facename)
00181     if not FacePRM:
00182         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
       derived\n")
00183         return
00184     # Get points of L-Shape rebar
00185     points = getpointsOfLShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover,
      orientation)
00186     import Part
00187     import Arch
00188     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00189     sketch.MapMode = "FlatFace"
00190     sketch.Support = [(structure, facename)]
00191     FreeCAD.ActiveDocument.recompute()
00192     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00193     sketch.addGeometry(Part.LineSegment(points[1], points[2]), False)
00194     import Sketcher
00195     if amount_spacing_check:
00196         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00197         FreeCAD.ActiveDocument.recompute()
00198     else:
00199         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00200         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
      f_cover)
00201     rebar.Rounding = rounding
00202     # Adds properties to the rebar object
00203     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Shape of rebar")).RebarShape = "LShapeRebar"
00204     rebar.ViewObject.setEditorMode("RebarShape", 2)
00205     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
```

```
      App::Property", "Front cover of rebar")).FrontCover = f_cover
00206     rebar.setEditorMode("FrontCover", 2)
00207     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Left Side cover of rebar")).LeftCover = l_cover
00208     rebar.setEditorMode("LeftCover", 2)
00209     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Right Side cover of rebar")).RightCover = r_cover
00210     rebar.setEditorMode("RightCover", 2)
00211     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00212     rebar.setEditorMode("BottomCover", 2)
00213     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
       "Amount radio button is checked")).AmountCheck
00214     rebar.setEditorMode("AmountCheck", 2)
00215     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
      , "Top cover of rebar")).TopCover = t_cover
00216     rebar.setEditorMode("TopCover", 2)
00217     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00218     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Shape of rebar")).Orientation = orientation
00219     rebar.setEditorMode("Orientation", 2)
00220     rebar.setEditorMode("TrueSpacing", 2)
00221     if amount_spacing_check:
00222         rebar.AmountCheck = True
00223     else:
00224         rebar.AmountCheck = False
00225         rebar.TrueSpacing = amount_spacing_value
00226     rebar.Label = "LShapeRebar"
00227     FreeCAD.ActiveDocument.recompute()
00228     return rebar
00229
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.3.2 Variable Documentation

6.3.2.1 string LShapeRebar.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file LShapeRebar.py.

6.3.2.2 string LShapeRebar.__title__ = "LShapeRebar" [private]

Definition at line 24 of file LShapeRebar.py.

6.3.2.3 string LShapeRebar.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file LShapeRebar.py.

6.4 PopUpImage Namespace Reference

Classes

- class PopUpImage

Functions

- def showPopUpImageDialog (img)

Variables

- string __title__ = "PopUpImage"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

6.4.1 Function Documentation

6.4.1.1 def PopUpImage.showPopUpImageDialog (*img*)

```
showPopUpImageDialog(image): This function will show a given image in a pop-up
dialog box.
```

Definition at line 43 of file PopUpImage.py.

```
00043 def showPopUpImageDialog(img):
00044     """ showPopUpImageDialog(image): This function will show a given image in a pop-up
00045     dialog box."""
00046     dialog = PopUpImage(img)
00047     dialog.exec_()
00048
```

6.4.2 Variable Documentation

6.4.2.1 string PopUpImage.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file PopUpImage.py.

6.4.2.2 string PopUpImage.__title__ = "PopUpImage" [private]

Definition at line 24 of file PopUpImage.py.

6.4.2.3 string PopUpImage.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file PopUpImage.py.

6.5 RebarDistribution Namespace Reference

Classes

- class _RebarDistributionDialog

Functions

- def getCustomSpacingString (amount1, spacing1, amount2, spacing2, amount3, spacing3, frontCover, size)
- def getupleOfCustomSpacing (span_string)
- def runRebarDistribution (self)
- def removeRebarDistribution (self)

Variables

- string __title__ = "DialogDistribution"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"
- CustomSpacing

6.5.1 Function Documentation

6.5.1.1 def RebarDistribution.getCustomSpacingString (*amount1, spacing1, amount2, spacing2, amount3, spacing3, frontCover, size*)

Definition at line 63 of file RebarDistribution.py.

```
00063 def getCustomSpacingString(amount1, spacing1, amount2, spacing2, amount3, spacing3,
      frontCover, size):
00064     seg1_area = amount1 * spacing1 - spacing1 / 2
00065     seg3_area = amount3 * spacing3 - spacing3 / 2
00066     seg2_area = size - seg1_area - seg3_area - 2 * frontCover
00067     if seg2_area < 0:
00068         FreeCAD.Console.PrintError("Sum of length of segment 1 and segment 2 is greater than length of
       rebar expands.\n")
00069         return
00070     if spacing1 and spacing2 and spacing3 and amount1 and amount2 and amount3:
00071         pass
00072     else:
00073         if spacing1 and spacing2 and spacing3:
00074             amount2 = math.ceil(seg2_area / spacing2)
00075             spacing2 = seg2_area / amount2
00076         elif amount1 and amount2 and amount3:
00077             spacing2 = math.floor(seg2_area / amount2)
00078     CustomSpacing = str(amount1) + "@" + str(spacing1) + "+" + str(int(amount2)) + "@" + str(spacing2) + "+
      " + str(amount3) + "@" + str(spacing3)
00079     return CustomSpacing
00080
```

6.5.1.2 def RebarDistribution.getupleOfCustomSpacing (*span_string*)

```
gettupleOfCustomSpacing(span_string): This function take input
in specific syntax and return output in the form of list. For eg.
Input: "3@100+2@200+3@100"
Output: [(3, 100), (2, 200), (3, 100)]
```

Definition at line 81 of file RebarDistribution.py.

```
00081 def getupleOfCustomSpacing(span_string):
00082     """ gettupleOfCustomSpacing(span_string): This function take input
00083     in specific syntax and return output in the form of list. For eg.
00084     Input: "3@100+2@200+3@100"
00085     Output: [(3, 100), (2, 200), (3, 100)]"""
00086     import string
00087     span_st = string.strip(span_string)
00088     span_sp = string.split(span_st, '+')
00089     index = 0
00090     spacinglist = []
00091     while index < len(span_sp):
00092         # Find "@" recursively in span_sp array.
00093         in_sp = string.split(span_sp[index], '@')
00094         spacinglist.append((int(in_sp[0]),float(in_sp[1])))
00095         index += 1
00096     return spacinglist
00097
```

6.5.1.3 def RebarDistribution.removeRebarDistribution (*self*)

Definition at line 108 of file RebarDistribution.py.

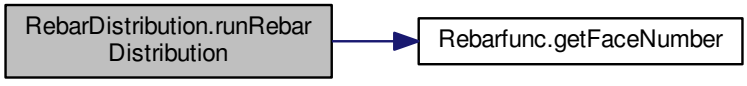
```
00108 def removeRebarDistribution(self):
00109     self.CustomSpacing = ""
00110     self.Rebar.CustomSpacing = ""
00111     FreeCAD.ActiveDocument.recompute()
00112
00113 #runRebarDistribution(App.ActiveDocument.Rebar)
00114
```

6.5.1.4 def RebarDistribution.runRebarDistribution (*self*)

Definition at line 98 of file RebarDistribution.py.

```
00098 def runRebarDistribution(self):
00099     frontCover = self.form.frontCover.text()
00100     frontCover = FreeCAD.Units.Quantity(frontCover).Value
00101     face = self.SelectedObj.Shape.Faces[getFaceNumber(self.FaceName) - 1]
00102     size = (ArchCommands.projectToVector(self.SelectedObj.Shape.copy(), face.normalAt(0, 0))).Length
00103     dialog = _RebarDistributionDialog(frontCover, size)
00104     dialog.setupUi()
00105     dialog.form.exec_()
00106     self.CustomSpacing = dialog.CustomSpacing
00107
```

Here is the call graph for this function:



6.5.2 Variable Documentation

6.5.2.1 string RebarDistribution.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file RebarDistribution.py.

6.5.2.2 string RebarDistribution.__title__ = "DialogDistribution" [private]

Definition at line 24 of file RebarDistribution.py.

6.5.2.3 string RebarDistribution.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file RebarDistribution.py.

6.5.2.4 RebarDistribution.CustomSpacing

Definition at line 106 of file RebarDistribution.py.

6.6 Rebarfunc Namespace Reference

Functions

- def getEdgesAngle (edge1, edge2)
- def checkRectangle (edges)
- def getBaseStructuralObject (obj)
- def getBaseObject (obj)
- def getFaceNumber (s)
- def facenormalDirection (structure=None, facename=None)
- def getTrueParametersOfStructure (obj)
- def getParametersOfFace (structure, facename, sketch=True)
- def extendedTangentPartLength (rounding, diameter, angle)
- def extendedTangentLength (rounding, diameter, angle)
- def check_selected_face ()
- def getSelectedFace (self)
- def showWarning (message)
- def translate (context, text, disambig=None)

Variables

- string __title__ = "GenericRebarFuctions"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"
- SelectedObj
- FaceName

6.6.1 Function Documentation

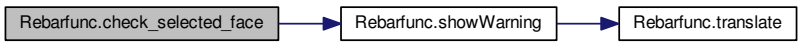
6.6.1.1 def Rebarfunc.check_selected_face ( )

```
check_selected_face(): This function checks whether user have selected
    any face or not.
```

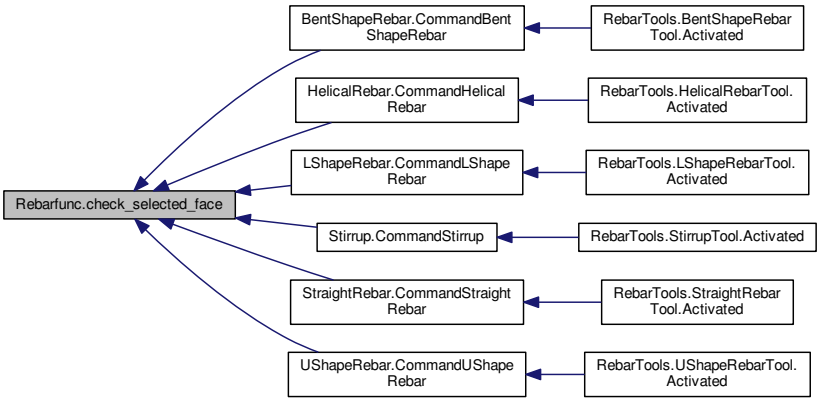
Definition at line 255 of file Rebarfunc.py.

```
00255 def check_selected_face():
00256     """ check_selected_face(): This function checks whether user have selected
00257         any face or not."""
00258     selected_objs = FreeCADGui.Selection.getSelectionEx()
00259     if not selected_objs:
00260         showWarning("Select any face of the structural element.")
00261         selected_obj = None
00262     else:
00263         selected_face_names = selected_objs[0].SubElementNames
00264         if not selected_face_names:
00265             selected_obj = None
00266             showWarning("Select any face of the structural element.")
00267         elif "Face" in selected_face_names[0]:
00268             if len(selected_face_names) > 1:
00269                 showWarning("You have selected more than one face of the structural element.")
00270                 selected_obj = None
00271             elif len(selected_face_names) == 1:
00272                 selected_obj = selected_objs[0]
00273         else:
00274             showWarning("Select any face of the selected the face.")
00275             selected_obj = None
00276     return selected_obj
00277
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.6.1.2 def Rebarfunc.checkRectangle (*edges*)

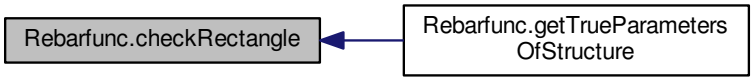
```
checkRectangle(edges=[]): This function checks whether the given form rectangle
    or not. It will return True when edges form rectangular shape or return False
    when edges not form a rectangular.
```

Definition at line 46 of file Rebarfunc.py.

```
00046 def checkRectangle(edges):
00047     """ checkRectangle(edges=[]): This function checks whether the given form rectangle
00048         or not. It will return True when edges form rectangular shape or return False
00049         when edges not form a rectangular."""
00050     angles = [round(getEdgesAngle(edges[0], edges[1])), round(
      getEdgesAngle(edges[0], edges[2])),
00051               round(getEdgesAngle(edges[0], edges[3]))]
00052     if angles.count(90) == 2 and (angles.count(180) == 1 or angles.count(0) == 1):
00053         return True
00054     else:
00055         return False
00056
```

Here is the call graph for this function:

Here is the caller graph for this function:



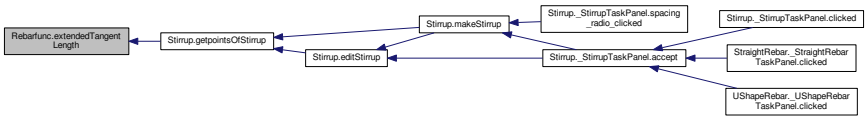
6.6.1.3 def Rebarfunc.extendedTangentLength (*rounding, diameter, angle*)

```
extendedTangentLength(rounding, diameter, angle): Get a extended
length of rounding at the end of Stirrup for bent.
```

Definition at line 243 of file Rebarfunc.py.

```
00243 def extendedTangentLength(rounding, diameter, angle):
00244     """ extendedTangentLength(rounding, diameter, angle): Get a extended
00245     length of rounding at the end of Stirrup for bent."""
00246     radius = rounding * diameter
00247     x1 = radius / math.sin(math.radians(angle))
00248     x2 = radius * math.tan(math.radians(90 - angle))
00249     return x1 + x2
00250
00251 # -------------------------------------------------------------------------
00252 # Warning / Alert functions when user do something wrong.
00253 #--------------------------------------------------------------------------
00254
```

Here is the caller graph for this function:



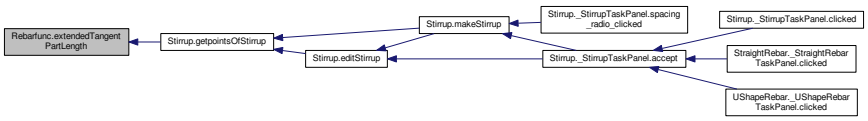
6.6.1.4 def Rebarfunc.extendedTangentPartLength (*rounding, diameter, angle*)

```
extendedTangentPartLength(rounding, diameter, angle): Get a extended
length of rounding on corners.
```

Definition at line 235 of file Rebarfunc.py.

```
00235 def extendedTangentPartLength(rounding, diameter, angle):
00236     """ extendedTangentPartLength(rounding, diameter, angle): Get a extended
00237     length of rounding on corners."""
00238     radius = rounding * diameter
00239     x1 = radius / math.tan(math.radians(angle))
00240     x2 = radius / math.cos(math.radians(90 - angle)) - radius
00241     return x1 + x2
00242
```

Here is the caller graph for this function:

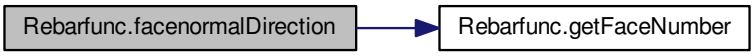


6.6.1.5 def Rebarfunc.facenormalDirection (*structure* = None, *facename* = None)

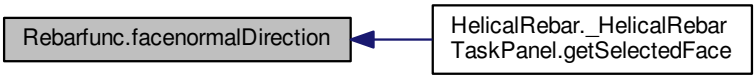
Definition at line 81 of file Rebarfunc.py.

```
00081 def facenormalDirection(structure = None, facename = None):
00082     if not structure and not facename:
00083         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00084         structure = selected_obj.Object
00085         facename = selected_obj.SubElementNames[0]
00086     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00087     normal = face.normalAt(0,0)
00088     normal = face.Placement.Rotation.inverted().multVec(normal)
00089     return normal
00090
00091 # ------------------------------------------------------------------------
00092 # Main functions which is use while creating any rebar.
00093 # ------------------------------------------------------------------------
00094
```

Here is the call graph for this function:



Here is the caller graph for this function:



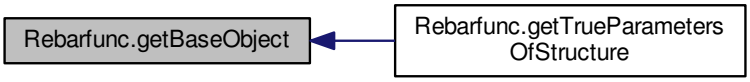
6.6.1.6 def Rebarfunc.getBaseObject (*obj*)

`getBaseObject(obj): This function will return last base object.`

Definition at line 65 of file Rebarfunc.py.

```
00065 def getBaseObject(obj):
00066     """ getBaseObject(obj): This function will return last base object."""
00067     if hasattr(obj, "Base"):
00068         return getBaseObject(obj.Base)
00069     else:
00070         return obj
00071
```

Here is the caller graph for this function:



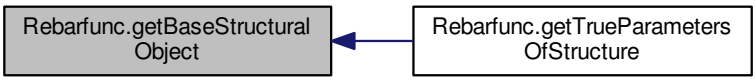
6.6.1.7 def Rebarfunc.getBaseStructuralObject (*obj*)

```
getBaseStructuralObject(obj): This function will return last base
    structural object.
```

Definition at line 57 of file Rebarfunc.py.

```
00057 def getBaseStructuralObject(obj):
00058     """ getBaseStructuralObject(obj): This function will return last base
00059         structural object."""
00060     if not obj.Base:
00061         return obj
00062     else:
00063         return getBaseStructuralObject(obj.Base)
00064
```

Here is the caller graph for this function:



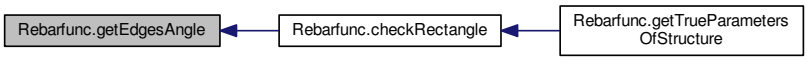
6.6.1.8 def Rebarfunc.getEdgesAngle (*edge1, edge2*)

```
getEdgesAngle(edge1, edge2): returns a angle between two edges.
```

Definition at line 38 of file Rebarfunc.py.

```
00038 def getEdgesAngle(edge1, edge2):
00039     """ getEdgesAngle(edge1, edge2): returns a angle between two edges."""
00040     vec1 = vec(edge1)
00041     vec2 = vec(edge2)
00042     angle = vec1.getAngle(vec2)
00043     angle = math.degrees(angle)
00044     return angle
00045
```

Here is the caller graph for this function:



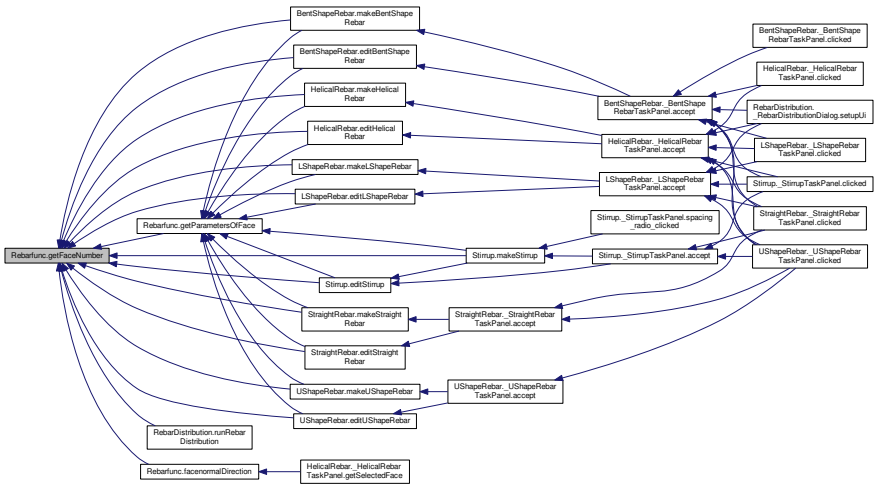
6.6.1.9 def Rebarfunc.getFaceNumber (*s*)

```
getFaceNumber(facename): This will return a face number from face name.
For eg.:
    Input: "Face12"
    Output: 12
```

Definition at line 72 of file Rebarfunc.py.

```
00072 def getFaceNumber(s):
00073     """ getFaceNumber(facename): This will return a face number from face name.
00074     For eg.:
00075         Input: "Face12"
00076         Output: 12"""
00077     head = s.rstrip('0123456789')
00078     tail = s[len(head):]
00079     return int(tail)
00080
```

Here is the caller graph for this function:



6.6.1.10 def Rebarfunc.getParametersOfFace (*structure, facename, sketch* = `True`)

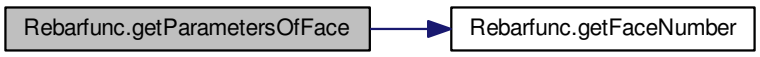
```
getParametersOfFace(structure, facename, sketch = True): This function will return
length, width and points of center of mass of a given face according to the sketch
value in the form of list.

For eg.:
Case 1: When sketch is True: We use True when we want to create rebars from sketch
    (planar rebars) and the sketch is strictly based on 2D so we neglected the normal
    axis of the face.
    Output: [(FaceLength, FaceWidth), (CenterOfMassX, CenterOfMassY)]

Case 2: When sketch is False: When we want to create non-planar rebars(like stirrup)
    or we want to create rebar from a wire. Also for creating rebar from wire
    we will require three coordinates (x, y, z).
    Output: [(FaceLength, FaceWidth), (CenterOfMassX, CenterOfMassY, CenterOfMassZ)]
```

Definition at line 126 of file Rebarfunc.py.

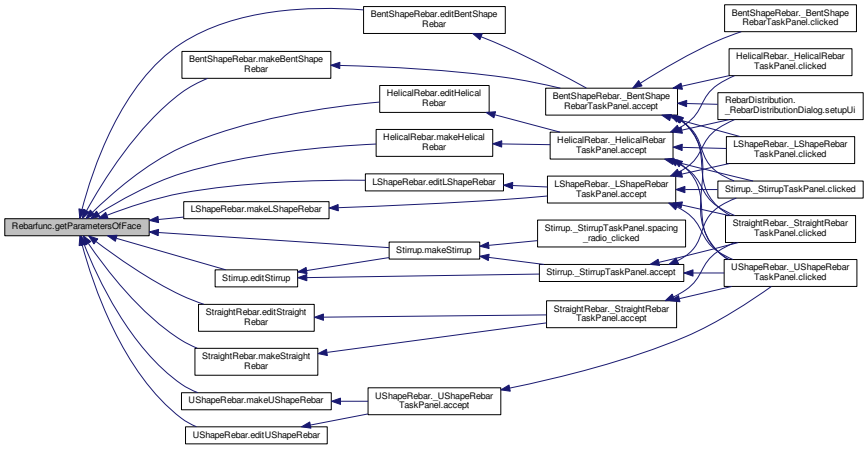
```
00126 def getParametersOfFace(structure, facename, sketch = True):
00127     """ getParametersOfFace(structure, facename, sketch = True): This function will return
00128     length, width and points of center of mass of a given face according to the sketch
00129     value in the form of list.
00130
00131     For eg.:
00132     Case 1: When sketch is True: We use True when we want to create rebars from sketch
00133         (planar rebars) and the sketch is strictly based on 2D so we neglected the normal
00134         axis of the face.
00135         Output: [(FaceLength, FaceWidth), (CenterOfMassX, CenterOfMassY)]
00136
00137     Case 2: When sketch is False: When we want to create non-planar rebars(like stirrup)
00138         or we want to create rebar from a wire. Also for creating rebar from wire
00139         we will require three coordinates (x, y, z).
00140         Output: [(FaceLength, FaceWidth), (CenterOfMassX, CenterOfMassY, CenterOfMassZ)]"""
00141     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00142     center_of_mass = face.CenterOfMass
00143     #center_of_mass = center_of_mass.sub(getBaseStructuralObject(structure).Placement.Base)
00144     center_of_mass = center_of_mass.sub(structure.Placement.Base)
00145     Edges = []
00146     facePRM = []
00147     # When structure is cubic. It support all structure is derived from
00148     # any other object (like a sketch, wire etc).
00149     if isCubic(structure.Shape):
00150         print 423
00151         for edge in face.Edges:
00152             if not Edges:
00153                 Edges.append(edge)
00154             else:
00155                 # Checks whether similar edges is already present in Edges list
00156                 # or not.
00157                 if round((vec(edge)).Length) not in [round((vec(x)).Length) for x in Edges]:
00158                     Edges.append(edge)
00159         if len(Edges) == 1:
00160             Edges.append(edge)
00161         # facePRM holds length of a edges.
00162         facePRM = [(vec(edge)).Length for edge in Edges]
00163         # Find the orientation of the face. Also eliminating normal axes
00164         # to the edge/face.
00165         # When edge is parallel to x-axis
00166         if round(Edges[0].tangentAt(0)[0]) in {1,-1}:
00167             x = center_of_mass[0]
00168             if round(Edges[1].tangentAt(0)[1]) in {1, -1}:
00169                 y = center_of_mass[1]
00170             else:
00171                 y = center_of_mass[2]
00172         # When edge is parallel to y-axis
00173         elif round(Edges[0].tangentAt(0)[1]) in {1,-1}:
00174             x = center_of_mass[1]
00175             if round(Edges[1].tangentAt(0)[0]) in {1, -1}:
00176                 # Change order when edge along x-axis is at second place.
00177                 facePRM.reverse()
00178                 y = center_of_mass[1]
00179             else:
00180                 y = center_of_mass[2]
00181         elif round(Edges[0].tangentAt(0)[2]) in {1,-1}:
00182             y = center_of_mass[2]
00183             if round(Edges[1].tangentAt(0)[0]) in {1, -1}:
00184                 x = center_of_mass[0]
00185             else:
00186                 x = center_of_mass[1]
00187             facePRM.reverse()
00188         facelength = facePRM[0]
00189         facewidth = facePRM[1]
00190     # When structure is not cubic. For founding parameters of given face
00191     # I have used bounding box.
00192     else:
00193         boundbox = face.BoundBox
00194         # Check that one length of bounding box is zero. Here bounding box
00195         # looks like a plane.
00196         if 0 in {round(boundbox.XLength), round(boundbox.YLength), round(boundbox.ZLength)}:
00197             normal = face.normalAt(0,0)
00198             normal = face.Placement.Rotation.inverted().multVec(normal)
00199             #print "x: ", boundbox.XLength
00200             #print "y: ", boundbox.YLength
00201             #print "z: ", boundbox.ZLength
00202             # Set length and width of user selected face of structural element
00203             flag = True
00204             # FIXME: Improve below logic.
00205             for i in range(len(normal)):
00206                 if round(normal[i]) == 0:
00207                     if flag and i == 0:
00208                         x = center_of_mass[i]
00209                         facelength =  boundbox.XLength
00210                         flag = False
00211                     elif flag and i == 1:
00212                         x = center_of_mass[i]
```

```
00213                         facelength = boundbox.YLength
00214                         flag = False
00215                     if i == 1:
00216                         y = center_of_mass[i]
00217                         facewidth = boundbox.YLength
00218                     elif i == 2:
00219                         y = center_of_mass[i]
00220                         facewidth = boundbox.ZLength
00221             #print [(facelength, facewidth), (x, y)]
00222     # Return parameter of the face when rebar is not created from the sketch.
00223     # For eg. non-planar rebars like stirrup etc.
00224     if not sketch:
00225         center_of_mass = face.CenterOfMass
00226         return [(facelength, facewidth), center_of_mass]
00227     #TODO: Add support when bounding box have depth. Here bounding box looks
00228     # like cuboid. If we given curved face.
00229     return [(facelength, facewidth), (x, y)]
00230
00231 # ------------------------------------------------------------------------
00232 # Functions which is mainly used while creating stirrup.
00233 # ------------------------------------------------------------------------
00234
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.6.1.11 def Rebarfunc.getSelectedFace (*self*)

Definition at line 278 of file Rebarfunc.py.

```
00278 def getSelectedFace(self):
00279     selected_objs = FreeCADGui.Selection.getSelectionEx()
00280     if selected_objs:
```

```
00281         if len(selected_objs[0].SubObjects) == 1:
00282             if "Face" in selected_objs[0].SubElementNames[0]:
00283                 self.SelectedObj = selected_objs[0].Object
00284                 self.FaceName = selected_objs[0].SubElementNames[0]
00285                 self.form.PickSelectedFaceLabel.setText("Selected face is " + self.FaceName)
00286             else:
00287                 showWarning("Select any face of the structural element.")
00288         else:
00289             showWarning("Select only one face of the structural element.")
00290     else:
00291         showWarning("Select any face of the structural element.")
00292
```

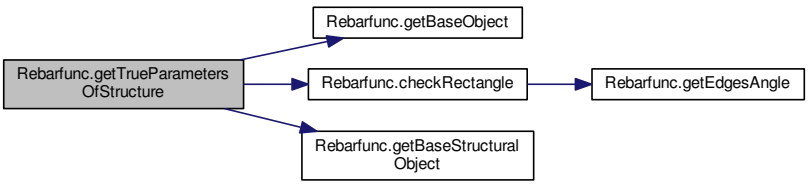
6.6.1.12 def Rebarfunc.getTrueParametersOfStructure (*obj*)

```
getTrueParametersOfStructure(obj): This function return actual length,
width and height of the structural element in the form of array like
[Length, Width, Height]
```

Definition at line 95 of file Rebarfunc.py.

```
00095 def getTrueParametersOfStructure(obj):
00096     """ getTrueParametersOfStructure(obj): This function return actual length,
00097     width and height of the structural element in the form of array like
00098     [Length, Width, Height]"""
00099     baseObject = getBaseObject(obj)
00100     # If selected_obj is not derived from any base object
00101     if baseObject:
00102         # If selected_obj is derived from SketchObject
00103         if baseObject.isDerivedFrom("Sketcher::SketchObject"):
00104             edges = baseObject.Shape.Edges
00105             if checkRectangle(edges):
00106                 for edge in edges:
00107                     # Representation vector of edge
00108                     rep_vector = edge.Vertexes[1].Point.sub(edge.Vertexes[0].Point)
00109                     rep_vector_angle = round(math.degrees(rep_vector.getAngle(FreeCAD.Vector(1,0,0))))
00110                     if rep_vector_angle in {0, 180}:
00111                         length = edge.Length
00112                     else:
00113                         width = edge.Length
00114             else:
00115                 return None
00116         else:
00117             return None
00118         height = obj.Height.Value
00119     else:
00120         structuralBaseObject = getBaseStructuralObject(obj)
00121         length = structuralBaseObject.Length.Value
00122         width = structuralBaseObject.Width.Value
00123         height = structuralBaseObject.Height.Value
00124     return [length, width, height]
00125
```

Here is the call graph for this function:



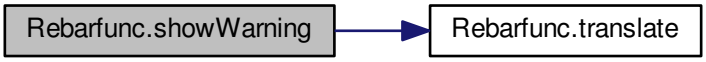
6.6.1.13 def Rebarfunc.showWarning (*message*)

```
showWarning(message): This function is used to produce warning
message for the user.
```

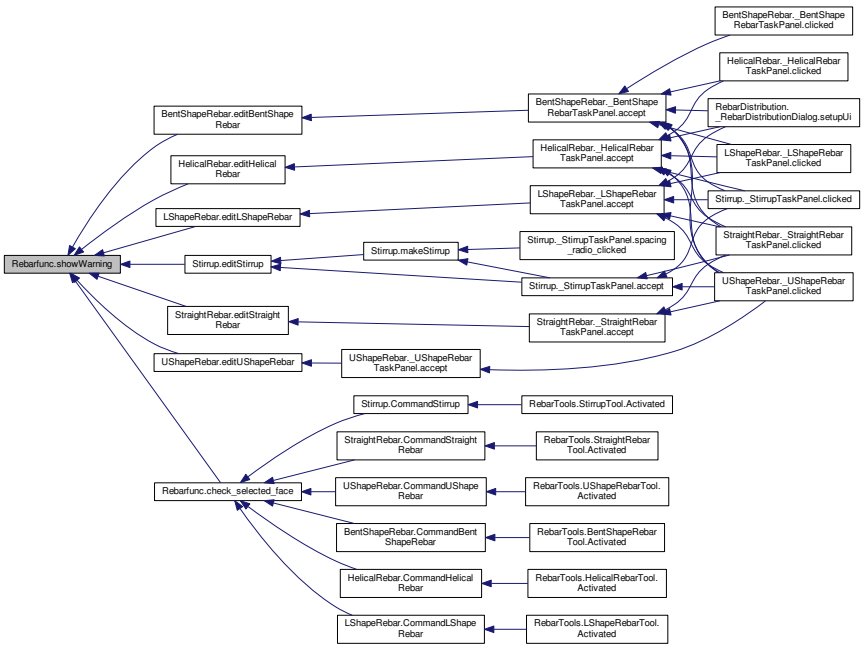
Definition at line 293 of file Rebarfunc.py.

```
00293 def showWarning(message):
00294     """ showWarning(message): This function is used to produce warning
00295     message for the user."""
00296     msg = QtGui.QMessageBox()
00297     msg.setIcon(QtGui.QMessageBox.Warning)
00298     msg.setText(translate("RebarAddon", message))
00299     msg.setStandardButtons(QtGui.QMessageBox.Ok)
00300     msg.exec_()
00301
00302 # Qt tanslation handling
```

Here is the call graph for this function:



Here is the caller graph for this function:

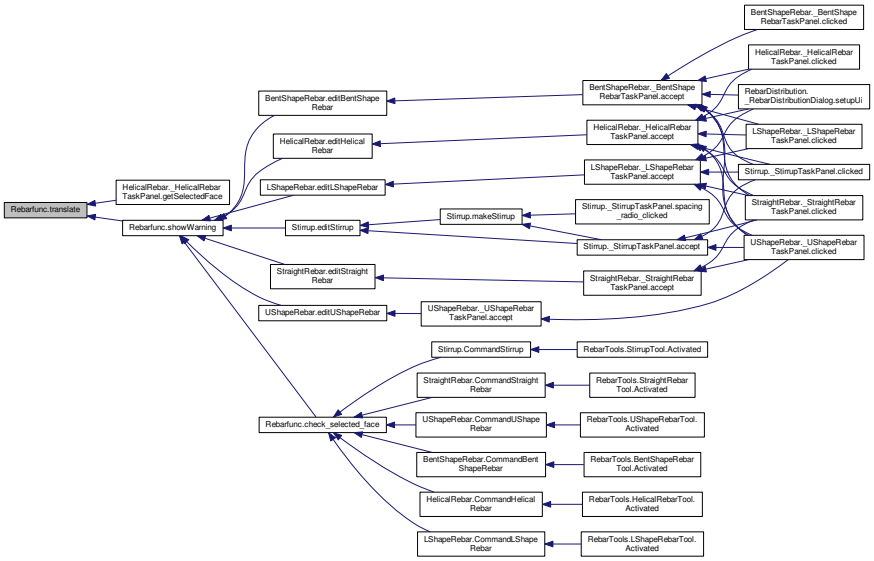


6.6.1.14 def Rebarfunc.translate (*context, text, disambig* = None)

Definition at line 303 of file Rebarfunc.py.

```
00303 def translate(context, text, disambig=None):
00304     return QtCore.QCoreApplication.translate(context, text, disambig)
00305
```

Here is the caller graph for this function:



6.6.2 Variable Documentation

6.6.2.1 string Rebarfunc.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file Rebarfunc.py.

6.6.2.2 string Rebarfunc.__title__ = "GenericRebarFuctions" [private]

Definition at line 24 of file Rebarfunc.py.

6.6.2.3 string Rebarfunc.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file Rebarfunc.py.

6.6.2.4 Rebarfunc.FaceName

Definition at line 284 of file Rebarfunc.py.

6.6.2.5 Rebarfunc.SelectedObj

Definition at line 283 of file Rebarfunc.py.

6.7 RebarTools Namespace Reference

Classes

- class BentShapeRebarTool
- class HelicalRebarTool
- class LShapeRebarTool
- class StirrupTool
- class StraightRebarTool
- class UShapeRebarTool

Variables

- string __title__ = "RebarCommands"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"
- list RebarCommands = ["Arch_Rebar_Straight", "Arch_Rebar_UShape", "Arch_Rebar_LShape", "Arch_Rebar_Stirrup", "Arch_Rebar_BentShape", "Arch_Rebar_Helical"]

6.7.1 Variable Documentation

6.7.1.1 string RebarTools.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file RebarTools.py.

6.7.1.2 string RebarTools.__title__ = "RebarCommands" [private]

Definition at line 24 of file RebarTools.py.

6.7.1.3 string RebarTools.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file RebarTools.py.

6.7.1.4 list RebarTools.RebarCommands = ["Arch_Rebar_Straight", "Arch_Rebar_UShape", "Arch_Rebar_LShape", "Arch_Rebar_Stirrup", "Arch_Rebar_BentShape", "Arch_Rebar_Helical"]

Definition at line 148 of file RebarTools.py.

6.8 Stirrup Namespace Reference

Classes

- class _StirrupTaskPanel

Functions

- def getpointsOfStirrup (FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle, bentFactor, diameter, rounding, facenormal)
- def makeStirrup (l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter, rounding, amount_spacing_check, amount_spacing_value, structure=None, facename=None)
- def editStirrup (Rebar, l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter, rounding, amount_spacing_check, amount_spacing_value, structure=None, facename=None)
- def editDialog (vobj)
- def CommandStirrup ()

Variables

- string __title__ = "StirrupRebar"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

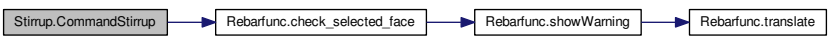
6.8.1 Function Documentation

6.8.1.1 def Stirrup.CommandStirrup ( )

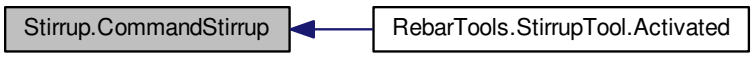
Definition at line 350 of file Stirrup.py.

```
00350 def CommandStirrup():
00351     selected_obj = check_selected_face()
00352     if selected_obj:
00353         FreeCADGui.Control.showDialog(_StirrupTaskPanel())
00354
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.8.1.2 def Stirrup.editDialog (*vobj*)

Definition at line 327 of file Stirrup.py.

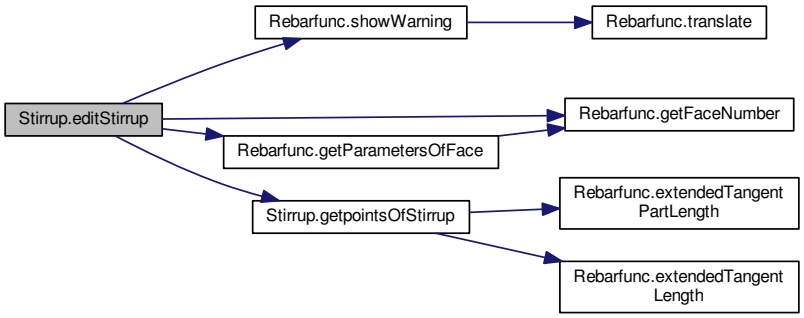
```
00327 def editDialog(vobj):
00328     FreeCADGui.Control.closeDialog()
00329     obj = _StirrupTaskPanel(vobj.Object)
00330     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00331     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00332     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00333     obj.form.t_sideCover.setText(str(vobj.Object.TopCover))
00334     obj.form.b_sideCover.setText(str(vobj.Object.BottomCover))
00335     obj.form.diameter.setText(str(vobj.Object.Diameter))
00336     obj.form.bentAngle.setCurrentIndex(obj.form.bentAngle.findText(str(vobj.Object.BentAngle)))
00337     obj.form.bentFactor.setValue(vobj.Object.BentFactor)
00338     obj.form.rounding.setValue(vobj.Object.Rounding)
00339     if vobj.Object.AmountCheck:
00340         obj.form.amount.setValue(vobj.Object.Amount)
00341     else:
00342         obj.form.amount_radio.setChecked(False)
00343         obj.form.spacing_radio.setChecked(True)
00344         obj.form.amount.setDisabled(True)
00345         obj.form.spacing.setEnabled(True)
00346         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00347     #obj.form.PickSelectedFace.setVisible(False)
00348     FreeCADGui.Control.showDialog(obj)
00349
```

6.8.1.3 def Stirrup.editStirrup (*Rebar, l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter, rounding, amount_spacing_check, amount_spacing_value, structure* = None, *facename* = None)

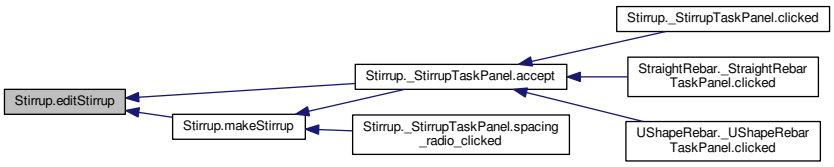
Definition at line 281 of file Stirrup.py.

```
00281         amount_spacing_check, amount_spacing_value, structure = None, facename = None):
00282     sketch = Rebar.Base
00283     if structure and facename:
00284         sketch.Support = [(structure, facename)]
00285     # Check if sketch support is empty.
00286     if not sketch.Support:
00287         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
      unchecked Edit->Preferences->Arch.")
00288         return
00289     # Assigned values
00290     facename = sketch.Support[0][1][0]
00291     structure = sketch.Support[0][0]
00292     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00293     #StructurePRM = getTrueParametersOfStructure(structure)
00294     # Get parameters of the face where sketch of rebar is drawn
00295     FacePRM = getParametersOfFace(structure, facename, False)
00296     FaceNormal = face.normalAt(0, 0)
00297     #FaceNormal = face.Placement.Rotation.inverted().multVec(FaceNormal)
00298     # Calculate the coordinates value of Stirrup rebar
00299     points = getpointsOfStirrup(FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle,
     bentFactor, diameter, rounding, FaceNormal)
00300     Rebar.Base.Points = points
00301     FreeCAD.ActiveDocument.recompute()
00302     Rebar.Direction = FaceNormal.negative()
00303     Rebar.OffsetStart = f_cover
00304     Rebar.OffsetEnd = f_cover
00305     Rebar.BentAngle = bentAngle
00306     Rebar.BentFactor = bentFactor
00307     Rebar.Rounding = rounding
00308     Rebar.Diameter = diameter
00309     if amount_spacing_check:
00310         Rebar.Amount = amount_spacing_value
00311         FreeCAD.ActiveDocument.recompute()
00312         Rebar.AmountCheck = True
00313     else:
00314         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00315         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00316         FreeCAD.ActiveDocument.recompute()
00317         Rebar.AmountCheck = False
00318     Rebar.FrontCover = f_cover
00319     Rebar.LeftCover = l_cover
00320     Rebar.RightCover = r_cover
00321     Rebar.TopCover = t_cover
00322     Rebar.BottomCover = b_cover
00323     Rebar.TrueSpacing = amount_spacing_value
00324     FreeCAD.ActiveDocument.recompute()
00325     return Rebar
00326
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.8.1.4 def Stirrup.getpointsOfStirrup (*FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle, bentFactor, diameter, rounding, facenormal*)

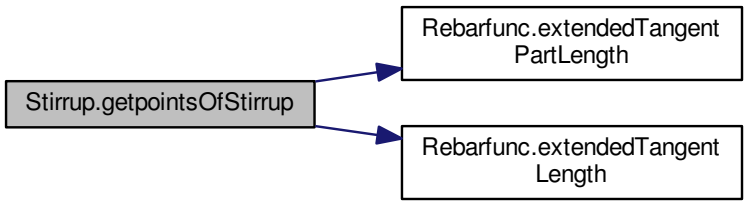
```
getpointsOfStirrup(FacePRM, LeftCover, RightCover, TopCover, BottomCover, BentAngle, BentFactor, Diameter, Rou
Return the coordinates points of the Stirrup in the form of array.
```

Definition at line 40 of file Stirrup.py.

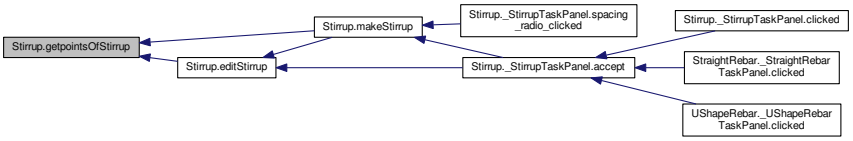
```
00040 def getpointsOfStirrup(FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle,
      bentFactor, diameter, rounding, facenormal):
00041     """ getpointsOfStirrup(FacePRM, LeftCover, RightCover, TopCover, BottomCover, BentAngle, BentFactor,
       Diameter, Rounding, FaceNormal):
00042     Return the coordinates points of the Stirrup in the form of array."""
00043     angle = 180 - bentAngle
00044     tangent_part_length = extendedTangentPartLength(rounding, diameter, angle)
00045     tangent_length = extendedTangentLength(rounding, diameter, angle)
00046     if round(facenormal[0]) in {1,-1}:
00047         x1 = FacePRM[1][0]
00048         y1 = FacePRM[1][1] - FacePRM[0][0] / 2 + l_cover
00049         z1 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover + tangent_part_length
00050         y2 = FacePRM[1][1] - FacePRM[0][0] / 2 + l_cover
00051         z2 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00052         y3 = FacePRM[1][1] + FacePRM[0][0] / 2 - r_cover
00053         z3 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00054         y4 = FacePRM[1][1] + FacePRM[0][0] / 2 - r_cover
00055         z4 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00056         y5 = FacePRM[1][1] - FacePRM[0][0] / 2 + l_cover - tangent_part_length
00057         z5 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00058         side_length = abs(y5 - y4) - tangent_part_length
00059         normal_dis = (diameter * (side_length + tangent_part_length)) / side_length
00060         x2 = x1 - normal_dis / 4
```

```
00061         x3 = x2 - normal_dis / 4
00062         x4 = x3 - normal_dis / 4
00063         x5 = x4 - normal_dis / 4
00064         x0 = x1 + normal_dis / 4
00065         y0 = y1 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(angle))
00066         z0 = z1 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(angle))
00067         x6 = x5 - normal_dis / 4
00068         y6 = y5 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(90 - angle))
00069         z6 = z5 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(90 - angle))
00070     elif round(facenormal[1]) in {1,-1}:
00071         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00072         y1 = FacePRM[1][1]
00073         z1 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover + tangent_part_length
00074         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00075         z2 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00076         x3 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00077         z3 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00078         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00079         z4 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00080         x5 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover - tangent_part_length
00081         z5 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00082         side_length = abs(x5 - x4) - tangent_part_length
00083         normal_dis = (diameter * (side_length + tangent_part_length)) / side_length
00084         y2 = y1 - normal_dis / 4
00085         y3 = y2 - normal_dis / 4
00086         y4 = y3 - normal_dis / 4
00087         y5 = y4 - normal_dis / 4
00088         y0 = y1 + normal_dis / 4
00089         x0 = x1 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(angle))
00090         z0 = z1 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(angle))
00091         x6 = x5 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(90 - angle))
00092         y6 = y5 - normal_dis / 4
00093         z6 = z5 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(90 - angle))
00094     elif round(facenormal[2]) in {1,-1}:
00095         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00096         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover + tangent_part_length
00097         z1 = FacePRM[1][2]
00098         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00099         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00100         x3 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00101         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00102         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00103         y4 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00104         x5 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover - tangent_part_length
00105         y5 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00106         side_length = abs(x5 - x4) - tangent_part_length
00107         normal_dis = (diameter * (side_length + tangent_part_length)) / side_length
00108         z2 = z1 - normal_dis / 4
00109         z3 = z2 - normal_dis / 4
00110         z4 = z3 - normal_dis / 4
00111         z5 = z4 - normal_dis / 4
00112         z0 = z1 + normal_dis / 4
00113         x0 = x1 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(angle))
00114         y0 = y1 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(angle))
00115         x6 = x5 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(90 - angle))
00116         y6 = y5 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(90 - angle))
00117         z6 = z5 - normal_dis / 4
00118     return [FreeCAD.Vector(x0, y0, z0), FreeCAD.Vector(x1, y1, z1),\
00119             FreeCAD.Vector(x2, y2, z2), FreeCAD.Vector(x3, y3, z3),\
00120             FreeCAD.Vector(x4, y4, z4), FreeCAD.Vector(x5, y5, z5),\
00121             FreeCAD.Vector(x6, y6, z6)]
00122
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.8.1.5 def Stirrup.makeStirrup (*l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter, rounding, amount_spacing_check, amount_spacing_value, structure* = None, *facename* = None)

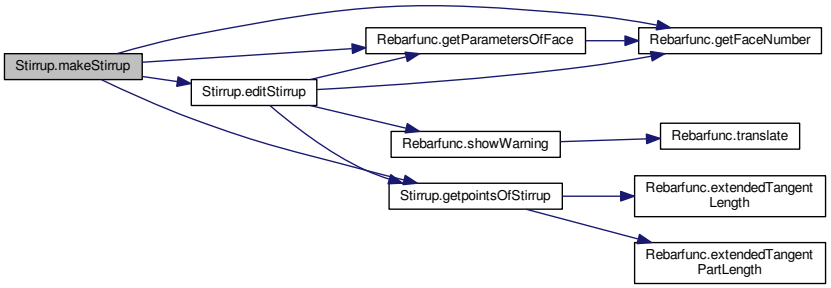
```
makeStirrup(LeftCover, RightCover, TopCover, BottomCover, FrontCover, BentAngle,
BentFactor, Diameter, Rounding, AmountSpacingCheck, AmountSpacingValue, Structure, Facename):
Adds the Stirrup reinforcement bar to the selected structural object.
```

Definition at line 210 of file Stirrup.py.

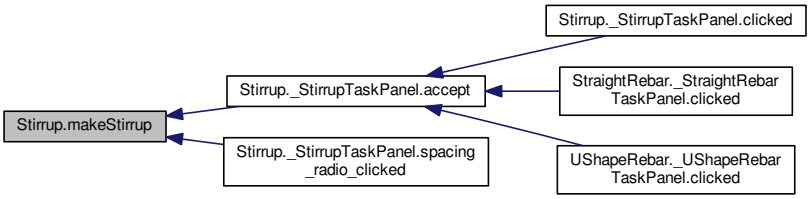
```
00210         amount_spacing_check, amount_spacing_value, structure = None, facename = None):
00211     """ makeStirrup(LeftCover, RightCover, TopCover, BottomCover, FrontCover, BentAngle,
00212     BentFactor, Diameter, Rounding, AmountSpacingCheck, AmountSpacingValue, Structure, Facename):
00213     Adds the Stirrup reinforcement bar to the selected structural object."""
00214     if not structure and not facename:
00215         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00216         structure = selected_obj.Object
00217         facename = selected_obj.SubElementNames[0]
00218     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00219     #StructurePRM = getTrueParametersOfStructure(structure)
00220     FacePRM = getParametersOfFace(structure, facename, False)
00221     FaceNormal = face.normalAt(0,0)
00222     #FaceNormal = face.Placement.Rotation.inverted().multVec(FaceNormal)
00223     if not FacePRM:
00224         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
      derived\n")
00225         return
00226     # Calculate the coordinate values of Stirrup
00227     points = getpointsOfStirrup(FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle,
    bentFactor, diameter, rounding, FaceNormal)
00228     import Draft
00229     line = Draft.makeWire(points, closed = False, face = True, support = None)
00230     import Arch
00231     line.Support = [(structure, facename)]
00232     if amount_spacing_check:
00233         rebar = Arch.makeRebar(structure, line, diameter, amount_spacing_value, f_cover)
00234     else:
00235         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00236         rebar = Arch.makeRebar(structure, line, diameter,\
00237             int((size - diameter) / amount_spacing_value), f_cover)
00238     rebar.Direction = FaceNormal.negative()
00239     rebar.Rounding = rounding
00240     # Adds properties to the rebar object
00241     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog",\
00242         QT_TRANSLATE_NOOP("App::Property","Shape of rebar")).RebarShape = "Stirrup"
00243     rebar.ViewObject.setEditorMode("RebarShape", 2)
00244     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog",\
00245         QT_TRANSLATE_NOOP("App::Property", "Left Side cover of rebar")).LeftCover = l_cover
00246     rebar.setEditorMode("LeftCover", 2)
00247     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog",\
00248         QT_TRANSLATE_NOOP("App::Property", "Right Side cover of rebar")).RightCover = r_cover
00249     rebar.setEditorMode("RightCover", 2)
00250     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog",\
00251         QT_TRANSLATE_NOOP("App::Property", "Top Side cover of rebar")).TopCover = t_cover
00252     rebar.setEditorMode("TopCover", 2)
00253     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog",\
00254         QT_TRANSLATE_NOOP("App::Property", "Bottom Side cover of rebar")).BottomCover = b_cover
00255     rebar.setEditorMode("BottomCover", 2)
00256     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog",\
00257         QT_TRANSLATE_NOOP("App::Property", "Top cover of rebar")).FrontCover = f_cover
00258     rebar.setEditorMode("FrontCover", 2)
00259     rebar.addProperty("App::PropertyInteger", "BentAngle", "RebarDialog",\
```

```
00260         QT_TRANSLATE_NOOP("App::Property", "Bent angle between at the end of rebar")).BentAngle = bentAngle
00261     rebar.setEditorMode("BentAngle", 2)
00262     rebar.addProperty("App::PropertyInteger", "BentFactor", "RebarDialog",\
00263         QT_TRANSLATE_NOOP("App::Property", "Bent Length is the equal to BentFactor * Diameter")).BentFactor
    = bentFactor
00264     rebar.setEditorMode("BentFactor", 2)
00265     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog",\
00266         QT_TRANSLATE_NOOP("App::Property", "Amount radio button is checked")).AmountCheck
00267     rebar.setEditorMode("AmountCheck", 2)
00268     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog",\
00269         QT_TRANSLATE_NOOP("App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00270     rebar.setEditorMode("TrueSpacing", 2)
00271     if amount_spacing_check:
00272         rebar.AmountCheck = True
00273     else:
00274         rebar.AmountCheck = False
00275         rebar.TrueSpacing = amount_spacing_value
00276     rebar.Label = "Stirrup"
00277     FreeCAD.ActiveDocument.recompute()
00278     return rebar
00279
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.8.2 Variable Documentation

6.8.2.1 string Stirrup.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file Stirrup.py.

6.8.2.2 string Stirrup.__title__ = "StirrupRebar" [private]

Definition at line 24 of file Stirrup.py.

6.8.2.3 string Stirrup.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file Stirrup.py.

6.9 StraightRebar Namespace Reference

Classes

- class _StraightRebarTaskPanel

Functions

- def getpointsOfStraightRebar (FacePRM, rt_cover, lb_cover, coverAlong, orientation)
- def makeStraightRebar (f_cover, coverAlong, rt_cover, lb_cover, diameter, amount_spacing_check, amount_spacing_value, orientation="Horizontal", structure=None, facename=None)
- def editStraightRebar (Rebar, f_cover, coverAlong, rt_cover, lb_cover, diameter, amount_spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def editDialog (vobj)
- def CommandStraightRebar ()

Variables

- string __title__ = "StraightRebar"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

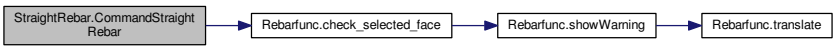
6.9.1 Function Documentation

6.9.1.1 def StraightRebar.CommandStraightRebar ( )

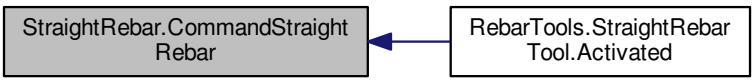
Definition at line 320 of file StraightRebar.py.

```
00320 def CommandStraightRebar():
00321     selected_obj = check_selected_face()
00322     if selected_obj:
00323         FreeCADGui.Control.showDialog(_StraightRebarTaskPanel())
00324
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.9.1.2 def StraightRebar.editDialog (*vobj*)

Definition at line 299 of file StraightRebar.py.

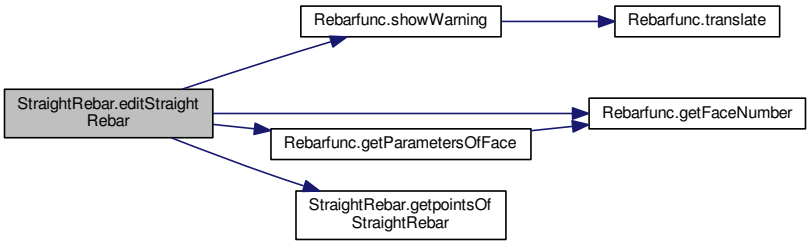
```
00299 def editDialog(vobj):
00300     FreeCADGui.Control.closeDialog()
00301     obj = _StraightRebarTaskPanel(vobj.Object)
00302     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00303     obj.form.r_sideCover.setText(str(vobj.Object.RightTopCover))
00304     obj.form.l_sideCover.setText(str(vobj.Object.LeftBottomCover))
00305     obj.form.bottomCover.setText(str(vobj.Object.Cover))
00306     obj.form.diameter.setText(str(vobj.Object.Diameter))
00307     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00308     obj.form.coverAlong.setCurrentIndex(obj.form.coverAlong.findText(str(vobj.Object.CoverAlong)))
00309     if vobj.Object.AmountCheck:
00310         obj.form.amount.setValue(vobj.Object.Amount)
00311     else:
00312         obj.form.amount_radio.setChecked(False)
00313         obj.form.spacing_radio.setChecked(True)
00314         obj.form.amount.setDisabled(True)
00315         obj.form.spacing.setEnabled(True)
00316         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00317     #obj.form.PickSelectedFace.setVisible(False)
00318     FreeCADGui.Control.showDialog(obj)
00319
```

6.9.1.3 def StraightRebar.editStraightRebar (*Rebar, f_cover, coverAlong, rt_cover, lb_cover, diameter, amount_spacing_check, amount_spacing_value, orientation, structure* = None, *facename* = None)

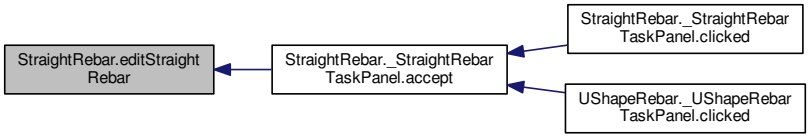
Definition at line 255 of file StraightRebar.py.

```
00255 def editStraightRebar(Rebar, f_cover, coverAlong, rt_cover, lb_cover, diameter,
      amount_spacing_check, amount_spacing_value, orientation, structure = None, facename = None):
00256     sketch = Rebar.Base
00257     if structure and facename:
00258         sketch.Support = [(structure, facename)]
00259         FreeCAD.ActiveDocument.recompute()
00260     # Check if sketch support is empty.
00261     if not sketch.Support:
00262         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00263         return
00264     # Assigned values
00265     facename = sketch.Support[0][1][0]
00266     structure = sketch.Support[0][0]
00267     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00268     #StructurePRM = getTrueParametersOfStructure(structure)
00269     # Get parameters of the face where sketch of rebar is drawn
00270     FacePRM = getParametersOfFace(structure, facename)
00271     # Get points of Striaght rebar
00272     points = getpointsOfStraightRebar(FacePRM, rt_cover, lb_cover, coverAlong,
      orientation)
00273     sketch.movePoint(0, 1, points[0], 0)
00274     FreeCAD.ActiveDocument.recompute()
00275     sketch.movePoint(0, 2, points[1], 0)
00276     FreeCAD.ActiveDocument.recompute()
00277     Rebar.OffsetStart = f_cover
00278     Rebar.OffsetEnd = f_cover
00279     if amount_spacing_check:
00280         Rebar.Amount = amount_spacing_value
00281         FreeCAD.ActiveDocument.recompute()
00282         Rebar.AmountCheck = True
00283     else:
00284         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00285         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00286         FreeCAD.ActiveDocument.recompute()
00287         Rebar.AmountCheck = False
00288     Rebar.FrontCover = f_cover
00289     Rebar.RightTopCover = rt_cover
00290     Rebar.LeftBottomCover = lb_cover
00291     Rebar.CoverAlong = coverAlong[0]
00292     Rebar.Cover = coverAlong[1]
00293     Rebar.TrueSpacing = amount_spacing_value
00294     Rebar.Diameter = diameter
00295     Rebar.Orientation = orientation
00296     FreeCAD.ActiveDocument.recompute()
00297     return Rebar
00298
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.9.1.4 def StraightRebar.getpointsOfStraightRebar (*FacePRM, rt_cover, lb_cover, coverAlong, orientation*)

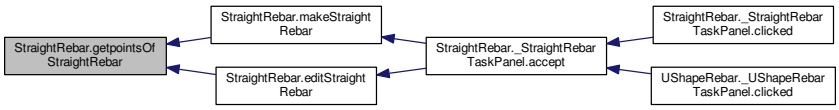
```
getpointsOfStraightRebar(FacePRM, RightTopcover, LeftBottomcover, CoverAlong, Orientation):
Return points of the Straight rebar in the form of array for sketch.

Case I: When Orientation is 'Horizontal':
    We have two option in CoverAlong i.e. 'Bottom Side' or 'Top Side'
Case II: When Orientation is 'Vertical':
    We have two option in CoverAlong i.e. 'Left Side' or 'Right Side'
```

Definition at line 40 of file StraightRebar.py.

```
00040 def getpointsOfStraightRebar(FacePRM, rt_cover, lb_cover, coverAlong, orientation):
00041     """ getpointsOfStraightRebar(FacePRM, RightTopcover, LeftBottomcover, CoverAlong, Orientation):
00042     Return points of the Straight rebar in the form of array for sketch.
00043
00044     Case I: When Orientation is 'Horizontal':
00045         We have two option in CoverAlong i.e. 'Bottom Side' or 'Top Side'
00046     Case II: When Orientation is 'Vertical':
00047         We have two option in CoverAlong i.e. 'Left Side' or 'Right Side'
00048     """
00049     if orientation == "Horizontal":
00050         if coverAlong[0] == "Bottom Side":
00051             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + lb_cover
00052             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + coverAlong[1]
00053             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - rt_cover
00054             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + coverAlong[1]
00055         elif coverAlong[0] == "Top Side":
00056             cover = FacePRM[0][1] - coverAlong[1]
00057             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + lb_cover
00058             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + cover
00059             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - rt_cover
00060             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + cover
00061     elif orientation == "Vertical":
00062         if coverAlong[0] == "Left Side":
```

```
00063             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + coverAlong[1]
00064             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + lb_cover
00065             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + coverAlong[1]
00066             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + FacePRM[0][1] - rt_cover
00067         elif coverAlong[0] == "Right Side":
00068             cover = FacePRM[0][0] - coverAlong[1]
00069             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + cover
00070             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + lb_cover
00071             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + cover
00072             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + FacePRM[0][1] - rt_cover
00073     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0)]
00074
```

Here is the caller graph for this function:



6.9.1.5 def StraightRebar.makeStraightRebar (*f_cover, coverAlong, rt_cover, lb_cover, diameter, amount_spacing_check, amount_spacing_value, orientation* = "Horizontal", *structure* = None, *facename* = None)

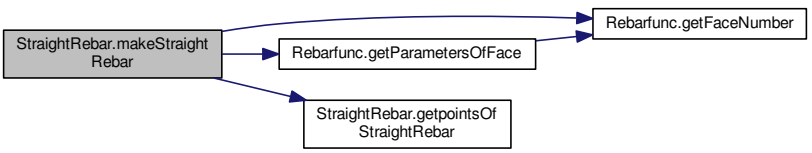
```
Adds the straight reinforcement bar to the selected structural object.

Case I: When orientation of straight rebar is 'Horizontal':
    makeStraightRebar(FrontCover, CoverAlong, RightCover, LeftCover, Diameter, AmountSpacingCheck, AmountSpaci
    Structure, Facename)
    Note: Type of CoverAlong argument is a tuple. Syntax: (<Along>, <Value>). Here we have horizontal orientat
    and Bottom Side to <Along> arguments.
    For eg. ("Top Side", 20) and ("Bottom Side", 20)

Case II: When orientation of straight rebar is 'Vertical':
    makeStraightRebar(FrontCover, CoverAlong, TopCover, BottomCover, Diameter, AmountSpacingCheck, AmountSpaci
    Structure, Facename)
    Note: Type of CoverAlong argument is a tuple. Syntax: (<Along>, <Value>). Here we have vertical orientatio
    and Right Side to <Along> arguments.
    For eg. ("Left Side", 20) and ("Right Side", 20)
```

Definition at line 185 of file StraightRebar.py.

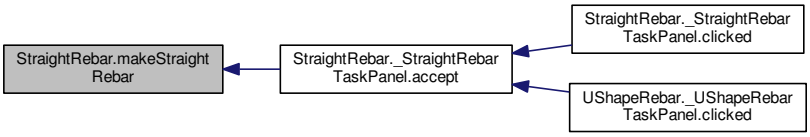
```
00185 def makeStraightRebar(f_cover, coverAlong, rt_cover, lb_cover, diameter,
      amount_spacing_check, amount_spacing_value, orientation = "Horizontal", structure = None, facename = None):
00186     """ Adds the straight reinforcement bar to the selected structural object.
00187
00188     Case I: When orientation of straight rebar is 'Horizontal':
00189         makeStraightRebar(FrontCover, CoverAlong, RightCover, LeftCover, Diameter, AmountSpacingCheck,
      AmountSpacingValue, Orientation = "Horizontal",
00190         Structure, Facename)
00191         Note: Type of CoverAlong argument is a tuple. Syntax: (<Along>, <Value>). Here we have horizontal
      orientation so we can pass Top Side
00192         and Bottom Side to <Along> arguments.
00193         For eg. ("Top Side", 20) and ("Bottom Side", 20)
00194
00195     Case II: When orientation of straight rebar is 'Vertical':
00196         makeStraightRebar(FrontCover, CoverAlong, TopCover, BottomCover, Diameter, AmountSpacingCheck,
      AmountSpacingValue, Orientation = "Horizontal",
00197         Structure, Facename)
00198         Note: Type of CoverAlong argument is a tuple. Syntax: (<Along>, <Value>). Here we have vertical
      orientation so we can pass Left Side
00199         and Right Side to <Along> arguments.
00200         For eg. ("Left Side", 20) and ("Right Side", 20)
00201     """
00202     if not structure and not facename:
00203         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
```

```
00204         structure = selected_obj.Object
00205         facename = selected_obj.SubElementNames[0]
00206     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00207     #StructurePRM = getTrueParametersOfStructure(structure)
00208     FacePRM = getParametersOfFace(structure, facename)
00209     if not FacePRM:
00210         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
      derived\n")
00211         return
00212     # Get points of Striaght rebar
00213     points = getpointsOfStraightRebar(FacePRM, rt_cover, lb_cover, coverAlong,
     orientation)
00214     import Part
00215     import Arch
00216     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00217     sketch.MapMode = "FlatFace"
00218     sketch.Support = [(structure, facename)]
00219     FreeCAD.ActiveDocument.recompute()
00220     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00221     if amount_spacing_check:
00222         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00223         FreeCAD.ActiveDocument.recompute()
00224     else:
00225         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00226         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
     f_cover)
00227     # Adds properties to the rebar object
00228     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
     App::Property", "Shape of rebar")).RebarShape = "StraightRebar"
00229     rebar.ViewObject.setEditorMode("RebarShape", 2)
00230     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
     App::Property", "Front cover of rebar")).FrontCover = f_cover
00231     rebar.setEditorMode("FrontCover", 2)
00232     rebar.addProperty("App::PropertyDistance", "RightTopCover", "RebarDialog", QT_TRANSLATE_NOOP("
     App::Property", "Right/Top Side cover of rebar")).RightTopCover = rt_cover
00233     rebar.setEditorMode("RightTopCover", 2)
00234     rebar.addProperty("App::PropertyDistance", "LeftBottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
     App::Property", "Left/Bottom Side cover of rebar")).LeftBottomCover = lb_cover
00235     rebar.setEditorMode("LeftBottomCover", 2)
00236     rebar.addProperty("App::PropertyString", "CoverAlong", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
     , "Cover along")).CoverAlong = coverAlong[0]
00237     rebar.setEditorMode("CoverAlong", 2)
00238     rebar.addProperty("App::PropertyDistance", "Cover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property", "
     Cover of rebar along user selected side")).Cover = coverAlong[1]
00239     rebar.setEditorMode("Cover", 2)
00240     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
      "Amount radio button is checked")).AmountCheck
00241     rebar.setEditorMode("AmountCheck", 2)
00242     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
     App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00243     rebar.setEditorMode("TrueSpacing", 2)
00244     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
     ", "Shape of rebar")).Orientation = orientation
00245     rebar.setEditorMode("Orientation", 2)
00246     if amount_spacing_check:
00247         rebar.AmountCheck = True
00248     else:
00249         rebar.AmountCheck = False
00250         rebar.TrueSpacing = amount_spacing_value
00251     rebar.Label = "StraightRebar"
00252     FreeCAD.ActiveDocument.recompute()
00253     return rebar
00254
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.9.2 Variable Documentation

6.9.2.1 string StraightRebar.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file StraightRebar.py.

6.9.2.2 string StraightRebar.__title__ = "StraightRebar" [private]

Definition at line 24 of file StraightRebar.py.

6.9.2.3 string StraightRebar.__url__ = "https://www.freecadweb.org" [private]

Definition at line 26 of file StraightRebar.py.

6.10 UShapeRebar Namespace Reference

Classes

- class _UShapeRebarTaskPanel

Functions

- def getpointsOfUShapeRebar (FacePRM, r_cover, l_cover, b_cover, t_cover, orientation)
- def makeUShapeRebar (f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding, amount_spacing_↩ check, amount_spacing_value, orientation="Bottom", structure=None, facename=None)
- def editUShapeRebar (Rebar, f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding, amount_↩ spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def editDialog (vobj)
- def CommandUShapeRebar ()

Variables

- string __title__ = "UShapeRebar"
- string __author__ = "Amritpal Singh"
- string __url__ = "https://www.freecadweb.org"

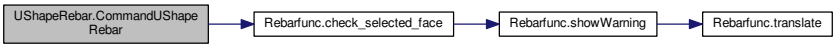
6.10.1 Function Documentation

6.10.1.1 def UShapeRebar.CommandUShapeRebar ( )

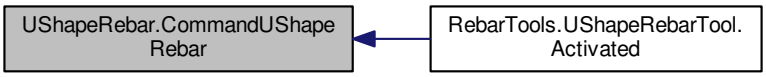
Definition at line 313 of file UShapeRebar.py.

```
00313 def CommandUShapeRebar():
00314     selected_obj = check_selected_face()
00315     if selected_obj:
00316         FreeCADGui.Control.showDialog(_UShapeRebarTaskPanel())
00317
```

Here is the call graph for this function:



Here is the caller graph for this function:



6.10.1.2 def UShapeRebar.editDialog (*vobj*)

Definition at line 291 of file UShapeRebar.py.

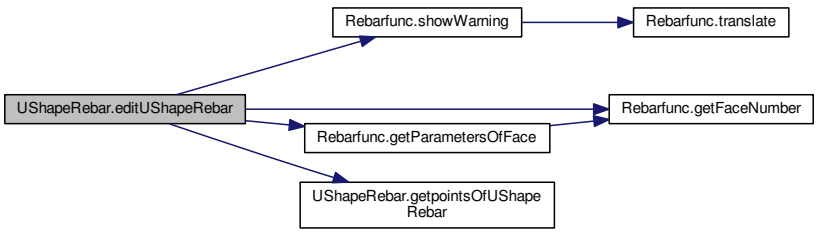
```
00291 def editDialog(vobj):
00292     FreeCADGui.Control.closeDialog()
00293     obj = _UShapeRebarTaskPanel(vobj.Object)
00294     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00295     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00296     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00297     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00298     obj.form.diameter.setText(str(vobj.Object.Diameter))
00299     obj.form.topCover.setText(str(vobj.Object.TopCover))
00300     obj.form.rounding.setValue(vobj.Object.Rounding)
00301     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00302     if vobj.Object.AmountCheck:
00303         obj.form.amount.setValue(vobj.Object.Amount)
00304     else:
00305         obj.form.amount_radio.setChecked(False)
00306         obj.form.spacing_radio.setChecked(True)
00307         obj.form.amount.setDisabled(True)
00308         obj.form.spacing.setEnabled(True)
00309         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00310     #obj.form.PickSelectedFace.setVisible(False)
00311     FreeCADGui.Control.showDialog(obj)
00312
```

6.10.1.3 def UShapeRebar.editUShapeRebar (*Rebar, f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation, structure* = None, *facename* = None)

Definition at line 239 of file UShapeRebar.py.

```
00239 def editUShapeRebar(Rebar, f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding,
      amount_spacing_check, amount_spacing_value, orientation, structure = None, facename = None):
00240     sketch = Rebar.Base
00241     if structure and facename:
00242         sketch.Support = [(structure, facename)]
00243     # Check if sketch support is empty.
00244     if not sketch.Support:
00245         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00246         return
00247     # Assigned values
00248     facename = sketch.Support[0][1][0]
00249     structure = sketch.Support[0][0]
00250     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00251     #StructurePRM = getTrueParametersOfStructure(structure)
00252     # Get parameters of the face where sketch of rebar is drawn
00253     FacePRM = getParametersOfFace(structure, facename)
00254     # Get points of U-Shape rebar
00255     points = getpointsOfUShapeRebar(FacePRM, r_cover, l_cover, b_cover, t_cover,
      orientation)
00256     sketch.movePoint(0, 1, points[0], 0)
00257     FreeCAD.ActiveDocument.recompute()
00258     sketch.movePoint(0, 2, points[1], 0)
00259     FreeCAD.ActiveDocument.recompute()
00260     sketch.movePoint(1, 1, points[1], 0)
00261     FreeCAD.ActiveDocument.recompute()
00262     sketch.movePoint(1, 2, points[2], 0)
00263     FreeCAD.ActiveDocument.recompute()
00264     sketch.movePoint(2, 1, points[2], 0)
00265     FreeCAD.ActiveDocument.recompute()
00266     sketch.movePoint(2, 2, points[3], 0)
00267     FreeCAD.ActiveDocument.recompute()
00268     Rebar.OffsetStart = f_cover
00269     Rebar.OffsetEnd = f_cover
00270     if amount_spacing_check:
00271         Rebar.Amount = amount_spacing_value
00272         FreeCAD.ActiveDocument.recompute()
00273         Rebar.AmountCheck = True
00274     else:
00275         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00276         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00277         FreeCAD.ActiveDocument.recompute()
00278         Rebar.AmountCheck = False
00279     Rebar.Diameter = diameter
00280     Rebar.FrontCover = f_cover
00281     Rebar.RightCover = r_cover
00282     Rebar.LeftCover = l_cover
00283     Rebar.BottomCover = b_cover
00284     Rebar.TopCover = t_cover
00285     Rebar.Rounding = rounding
00286     Rebar.TrueSpacing = amount_spacing_value
00287     Rebar.Orientation = orientation
00288     FreeCAD.ActiveDocument.recompute()
00289     return Rebar
00290
```

Here is the call graph for this function:



Here is the caller graph for this function:

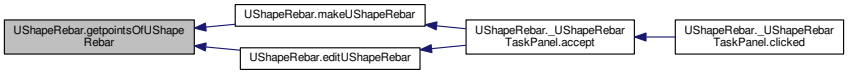
6.10.1.4 def UShapeRebar.getpointsOfUShapeRebar (*FacePRM, r_cover, l_cover, b_cover, t_cover, orientation*)

```
getpointsOfUShapeRebar(FacePRM, RightCover, LeftCover, BottomCover, TopCover, Orientation):
Return points of the UShape rebar in the form of array for sketch.
It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
```

Definition at line 40 of file UShapeRebar.py.

```
00040 def getpointsOfUShapeRebar(FacePRM, r_cover, l_cover, b_cover, t_cover, orientation):
00041     """ getpointsOfUShapeRebar(FacePRM, RightCover, LeftCover, BottomCover, TopCover, Orientation):
00042     Return points of the UShape rebar in the form of array for sketch.
00043     It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
00044     """
00045     if orientation == "Bottom":
00046         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00047         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00048         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00049         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00050         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00051         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00052         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00053         y4 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00054     elif orientation == "Top":
00055         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00056         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00057         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00058         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00059         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00060         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00061         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00062         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00063     elif orientation == "Left":
00064         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00065         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00066         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00067         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00068         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00069         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00070         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00071         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00072     elif orientation == "Right":
00073         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00074         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00075         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00076         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00077         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00078         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00079         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00080         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00081     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0),\
00082             FreeCAD.Vector(x3, y3, 0), FreeCAD.Vector(x4, y4, 0)]
00083
```

Here is the caller graph for this function:



6.10.1.5 def UShapeRebar.makeUShapeRebar (*f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation* = "Bottom", *structure* = None, *facename* = None)

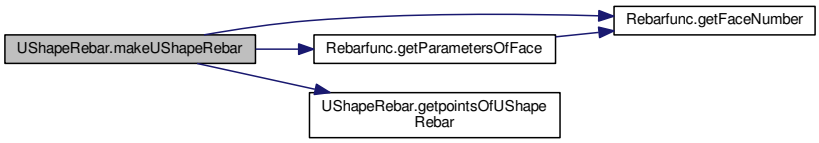
makeUShapeRebar(FrontCover, BottomCover, RightCover, LeftCover, Diameter, Topcover, Rounding, AmountSpacingChe
Orientation, Structure, Facename): Adds the U-Shape reinforcement bar to the selected structural object.
It takes four different types of orientations as input i.e 'Bottom', 'Top', 'Right', 'Left'.

Definition at line 177 of file UShapeRebar.py.

```
00177 def makeUShapeRebar(f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding,
      amount_spacing_check, amount_spacing_value, orientation = "Bottom", structure = None, facename = None):
00178     """ makeUShapeRebar(FrontCover, BottomCover, RightCover, LeftCover, Diameter, Topcover, Rounding,
       AmountSpacingCheck, AmountSpacingValue,
00179     Orientation, Structure, Facename): Adds the U-Shape reinforcement bar to the selected structural
       object.
00180     It takes four different types of orientations as input i.e 'Bottom', 'Top', 'Right', 'Left'.
00181     """
00182     if not structure and not facename:
00183         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00184         structure = selected_obj.Object
00185         facename = selected_obj.SubElementNames[0]
00186     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00187     #StructurePRM = getTrueParametersOfStructure(structure)
00188     FacePRM = getParametersOfFace(structure, facename)
00189     if not FacePRM:
00190         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
       derived\n")
00191         return
00192     # Get points of U-Shape rebar
00193     points = getpointsOfUShapeRebar(FacePRM, r_cover, l_cover, b_cover, t_cover,
      orientation)
00194     import Part
00195     import Arch
00196     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00197     sketch.MapMode = "FlatFace"
00198     sketch.Support = [(structure, facename)]
00199     FreeCAD.ActiveDocument.recompute()
00200     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00201     sketch.addGeometry(Part.LineSegment(points[1], points[2]), False)
00202     import Sketcher
00203     sketch.addGeometry(Part.LineSegment(points[2], points[3]), False)
00204     if amount_spacing_check:
00205         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00206         FreeCAD.ActiveDocument.recompute()
00207     else:
00208         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00209         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
      f_cover)
00210     rebar.Rounding = rounding
00211     # Adds properties to the rebar object
00212     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Shape of rebar")).RebarShape = "UShapeRebar"
00213     rebar.ViewObject.setEditorMode("RebarShape", 2)
00214     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Front cover of rebar")).FrontCover = f_cover
00215     rebar.setEditorMode("FrontCover", 2)
00216     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Right Side cover of rebar")).RightCover = r_cover
00217     rebar.setEditorMode("RightCover", 2)
00218     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Left Side cover of rebar")).LeftCover = l_cover
00219     rebar.setEditorMode("LeftCover", 2)
00220     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00221     rebar.setEditorMode("BottomCover", 2)
00222     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
       "Amount radio button is checked")).AmountCheck
00223     rebar.setEditorMode("AmountCheck", 2)
00224     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
      , "Top cover of rebar")).TopCover = t_cover
00225     rebar.setEditorMode("TopCover", 2)
00226     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00227     rebar.setEditorMode("TrueSpacing", 2)
00228     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Shape of rebar")).Orientation = orientation
00229     rebar.setEditorMode("Orientation", 2)
00230     if amount_spacing_check:
00231         rebar.AmountCheck = True
00232     else:
```

```
00233         rebar.AmountCheck = False
00234         rebar.TrueSpacing = amount_spacing_value
00235     rebar.Label = "UShapeRebar"
00236     FreeCAD.ActiveDocument.recompute()
00237     return rebar
00238
```

Here is the call graph for this function:



Here is the caller graph for this function:

6.10.2 Variable Documentation

6.10.2.1 string UShapeRebar.__author__ = "Amritpal Singh" [private]

Definition at line 25 of file UShapeRebar.py.

6.10.2.2 string UShapeRebar.__title__ = "UShapeRebar" [private]

Definition at line 24 of file UShapeRebar.py.

6.10.2.3 string UShapeRebar.__url__ = "https://www.freecadweb.org" [private]

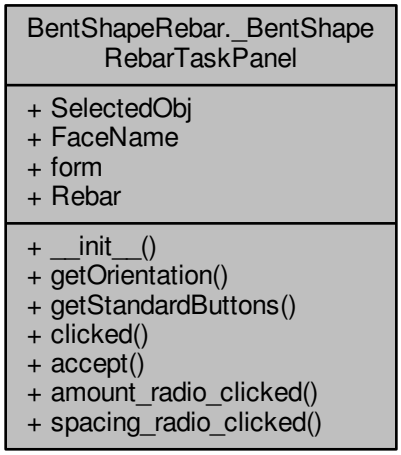
Definition at line 26 of file UShapeRebar.py.

Chapter 7

Class Documentation

7.1 BentShapeRebar._BentShapeRebarTaskPanel Class Reference

Collaboration diagram for BentShapeRebar._BentShapeRebarTaskPanel:



Public Member Functions

- def __init__ (self, Rebar=None)
- def getOrientation (self)
- def getStandardButtons (self)
- def clicked (self, button)
- def accept (self, signal=None)
- def amount_radio_clicked (self)
- def spacing_radio_clicked (self)

Public Attributes

- SelectedObj
- FaceName
- form
- Rebar

7.1.1 Detailed Description

Definition at line 105 of file BentShapeRebar.py.

7.1.2 Constructor & Destructor Documentation

7.1.2.1 def BentShapeRebar._BentShapeRebarTaskPanel.__init__ (*self, Rebar* = None)

Definition at line 106 of file BentShapeRebar.py.

```
00106     def __init__(self, Rebar = None):
00107         if not Rebar:
00108             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00109             self.SelectedObj = selected_obj.Object
00110             self.FaceName = selected_obj.SubElementNames[0]
00111         else:
00112             self.FaceName = Rebar.Base.Support[0][1][0]
00113             self.SelectedObj = Rebar.Base.Support[0][0]
00114         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00115         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "Bent Shape Rebar", None))
00116         self.form.orientation.addItems(["Bottom", "Top", "Right", "Left"])
00117         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00118         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00119         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00120         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00121         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00122         self.form.orientation.currentIndexChanged.connect(self.getOrientation)
00123         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/BentShapeRebar.svg"))
00124         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00125         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/BentShapeRebarDetailed.svg"))
00126         self.Rebar = Rebar
00127
```

7.1.3 Member Function Documentation

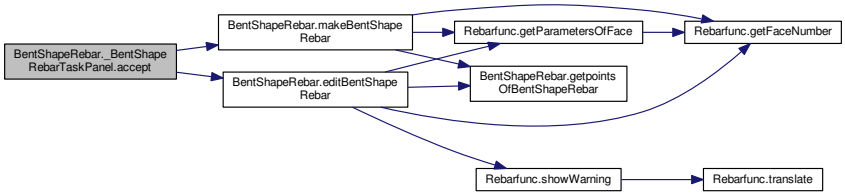
7.1.3.1 def BentShapeRebar._BentShapeRebarTaskPanel.accept (*self, signal* = None)

Definition at line 146 of file BentShapeRebar.py.

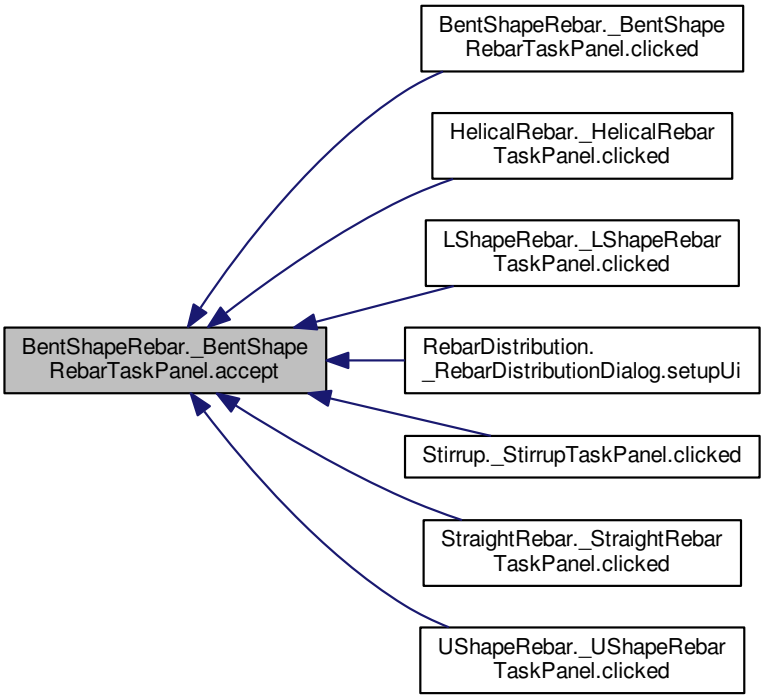
```
00146     def accept(self, signal = None):
00147         f_cover = self.form.frontCover.text()
00148         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00149         b_cover = self.form.bottomCover.text()
00150         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00151         l_cover = self.form.l_sideCover.text()
00152         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00153         r_cover = self.form.r_sideCover.text()
00154         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00155         t_cover = self.form.topCover.text()
00156         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00157         bentLength = self.form.bentLength.text()
00158         bentLength = FreeCAD.Units.Quantity(bentLength).Value
```

```
00159         bentAngle = self.form.bentAngle.value()
00160         diameter = self.form.diameter.text()
00161         diameter = FreeCAD.Units.Quantity(diameter).Value
00162         rounding = self.form.rounding.value()
00163         orientation = self.form.orientation.currentText()
00164         amount_check = self.form.amount_radio.isChecked()
00165         spacing_check = self.form.spacing_radio.isChecked()
00166         if not self.Rebar:
00167             if amount_check:
00168                 amount = self.form.amount.value()
00169                 rebar = makeBentShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, bentLength, bentAngle, rounding, True, amount, orientation, self.
      SelectedObj, self.FaceName)
00170             elif spacing_check:
00171                 spacing = self.form.spacing.text()
00172                 spacing = FreeCAD.Units.Quantity(spacing).Value
00173                 rebar = makeBentShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, bentLength, bentAngle, rounding, False, spacing, orientation, self.
      SelectedObj, self.FaceName)
00174         else:
00175             if amount_check:
00176                 amount = self.form.amount.value()
00177                 rebar = editBentShapeRebar(self.Rebar, f_cover, b_cover, l_cover,
      r_cover, diameter, t_cover, bentLength, bentAngle, rounding, True, amount, orientation, self.
      SelectedObj, self.FaceName)
00178             elif spacing_check:
00179                 spacing = self.form.spacing.text()
00180                 spacing = FreeCAD.Units.Quantity(spacing).Value
00181                 rebar = editBentShapeRebar(self.Rebar, f_cover, b_cover, l_cover,
      r_cover, diameter, t_cover, bentLength, bentAngle, rounding, False, spacing, orientation, self.
      SelectedObj, self.FaceName)
00182         if self.CustomSpacing:
00183             rebar.CustomSpacing = self.CustomSpacing
00184             FreeCAD.ActiveDocument.recompute()
00185         self.Rebar = rebar
00186         if signal == int(QtGui.QDialogButtonBox.Apply):
00187             pass
00188         else:
00189             FreeCADGui.Control.closeDialog(self)
00190
```

Here is the call graph for this function:



Here is the caller graph for this function:



7.1.3.2 def BentShapeRebar._BentShapeRebarTaskPanel.amount_radio_clicked (*self*)

Definition at line 191 of file BentShapeRebar.py.

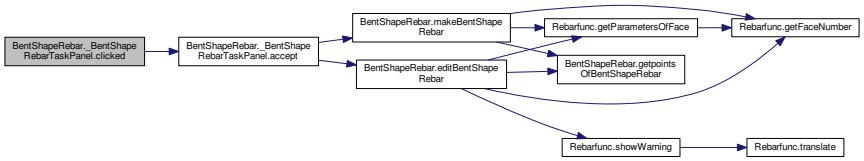
```
00191     def amount_radio_clicked(self):
00192         self.form.spacing.setEnabled(False)
00193         self.form.amount.setEnabled(True)
00194
```

7.1.3.3 def BentShapeRebar._BentShapeRebarTaskPanel.clicked (*self, button*)

Definition at line 142 of file BentShapeRebar.py.

```
00142     def clicked(self, button):
00143         if button == int(QtGui.QDialogButtonBox.Apply):
00144             self.accept(button)
00145
```

Here is the call graph for this function:



7.1.3.4 def BentShapeRebar._BentShapeRebarTaskPanel.getOrientation (*self*)

Definition at line 128 of file BentShapeRebar.py.

```
00128     def getOrientation(self):
00129         orientation = self.form.orientation.currentText()
00130         #if orientation == "Bottom":
00131         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
      "/icons/LShapeRebarBR.svg"))
00132         #elif orientation == "Top":
00133         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
      "/icons/LShapeRebarBL.svg"))
00134         #elif orientation == "Right":
00135         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
      "/icons/LShapeRebarTR.svg"))
00136         #else:
00137         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
      "/icons/LShapeRebarTL.svg"))
00138
```

7.1.3.5 def BentShapeRebar._BentShapeRebarTaskPanel.getStandardButtons (*self*)

Definition at line 139 of file BentShapeRebar.py.

```
00139     def getStandardButtons(self):
00140         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
     QtGui.QDialogButtonBox.Cancel)
00141
```

7.1.3.6 def BentShapeRebar._BentShapeRebarTaskPanel.spacing_radio_clicked (*self*)

Definition at line 195 of file BentShapeRebar.py.

```
00195     def spacing_radio_clicked(self):
00196         self.form.amount.setEnabled(False)
00197         self.form.spacing.setEnabled(True)
00198
00199
```

7.1.4 Member Data Documentation

7.1.4.1 BentShapeRebar._BentShapeRebarTaskPanel.FaceName

Definition at line 110 of file BentShapeRebar.py.

7.1.4.2 BentShapeRebar._BentShapeRebarTaskPanel.form

Definition at line 114 of file BentShapeRebar.py.

7.1.4.3 BentShapeRebar._BentShapeRebarTaskPanel.Rebar

Definition at line 126 of file BentShapeRebar.py.

7.1.4.4 BentShapeRebar._BentShapeRebarTaskPanel.SelectedObj

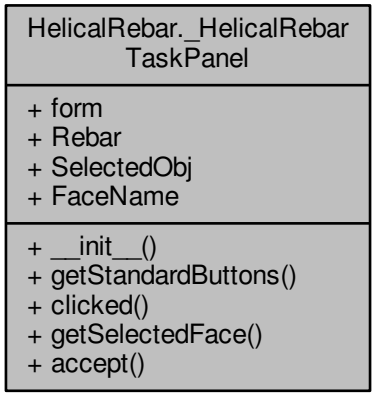
Definition at line 109 of file BentShapeRebar.py.

The documentation for this class was generated from the following file:

- BentShapeRebar.py

7.2 HelicalRebar._HelicalRebarTaskPanel Class Reference

Collaboration diagram for HelicalRebar._HelicalRebarTaskPanel:



Public Member Functions

- def __init__ (self, Rebar=None)
- def getStandardButtons (self)
- def clicked (self, button)
- def getSelectedFace (self)
- def accept (self, signal=None)

Public Attributes

- form
- Rebar
- SelectedObj
- FaceName

7.2.1 Detailed Description

Definition at line 107 of file HelicalRebar.py.

7.2.2 Constructor & Destructor Documentation

7.2.2.1 def HelicalRebar._HelicalRebarTaskPanel.__init__ (*self*, *Rebar* = None)

Definition at line 108 of file HelicalRebar.py.

```
00108     def __init__(self, Rebar = None):
00109         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00110         self.form.setWindowTitle(QtGui.QApplication.translate("Arch", "Helical Rebar", None))
00111         if not Rebar:
00112             normal = facenormalDirection()
00113         else:
00114             normal = facenormalDirection(Rebar.Base.Support[0][0], Rebar.Base.Support[0]
      [1][0])
00115         if not round(normal.z) in {1, -1}:
00116             self.form.topCoverLabel.setText(translate("RebarAddon", "Left Cover"))
00117             self.form.bottomCoverLabel.setText(translate("RebarAddon", "Right Cover"))
00118         self.form.PickSelectedFace.clicked.connect(self.getSelectedFace)
00119         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/HelicalRebar.svg"))
00120         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/HelicalRebarDetailed.svg"))
00121         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00122         self.Rebar = Rebar
00123         self.SelectedObj = None
00124         self.FaceName = None
00125
```

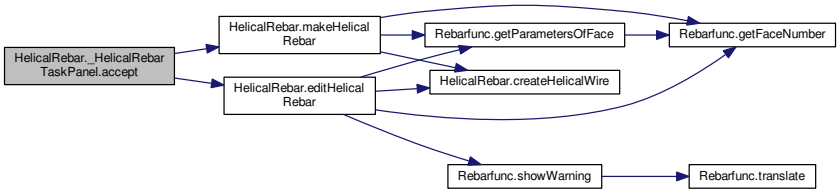
7.2.3 Member Function Documentation

7.2.3.1 def HelicalRebar._HelicalRebarTaskPanel.accept (*self*, *signal* = None)

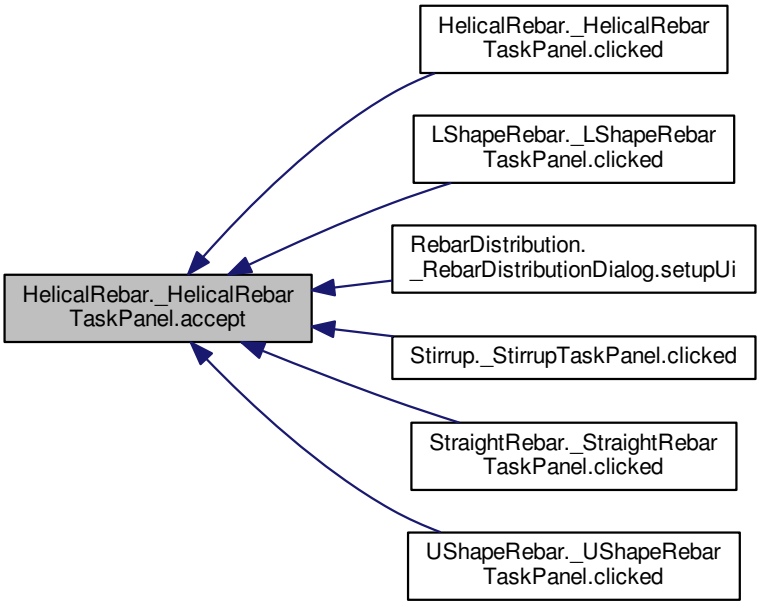
Definition at line 144 of file HelicalRebar.py.

```
00144     def accept(self, signal = None):
00145         b_cover = self.form.bottomCover.text()
00146         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00147         s_cover = self.form.sideCover.text()
00148         s_cover = FreeCAD.Units.Quantity(s_cover).Value
00149         t_cover = self.form.topCover.text()
00150         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00151         pitch = self.form.pitch.text()
00152         pitch = FreeCAD.Units.Quantity(pitch).Value
00153         diameter = self.form.diameter.text()
00154         diameter = FreeCAD.Units.Quantity(diameter).Value
00155         if not self.Rebar:
00156             rebar = makeHelicalRebar(s_cover, b_cover, diameter, t_cover, pitch, self.
      SelectedObj, self.FaceName)
00157         else:
00158             rebar = editHelicalRebar(self.Rebar, s_cover, b_cover, diameter, t_cover,
      pitch, self.SelectedObj, self.FaceName)
00159         self.Rebar = rebar
00160         if signal == int(QtGui.QDialogButtonBox.Apply):
00161             pass
00162         else:
00163             FreeCADGui.Control.closeDialog(self)
00164
```

Here is the call graph for this function:



Here is the caller graph for this function:

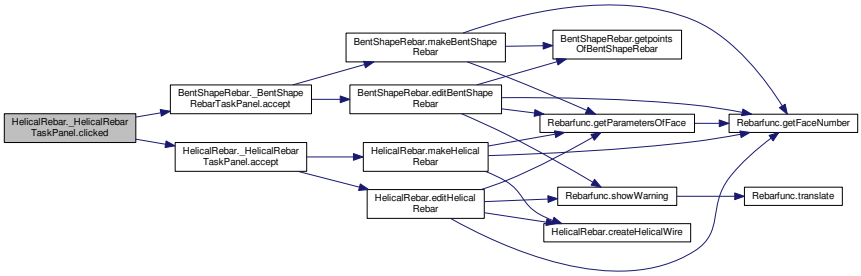


7.2.3.2 def HelicalRebar._HelicalRebarTaskPanel.clicked (*self, button*)

Definition at line 129 of file HelicalRebar.py.

```
00129     def clicked(self, button):
00130         if button == int(QtGui.QDialogButtonBox.Apply):
00131             self.accept(button)
00132
```

Here is the call graph for this function:

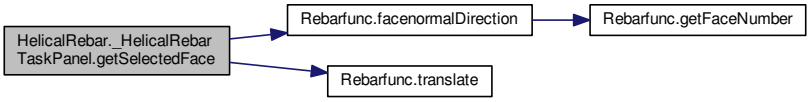


7.2.3.3 def HelicalRebar._HelicalRebarTaskPanel.getSelectedFace (*self*)

Definition at line 133 of file HelicalRebar.py.

```
00133     def getSelectedFace(self):
00134         getSelectedFace(self)
00135         normal = facenormalDirection()
00136         if not round(normal.z) in {1, -1}:
00137             self.form.topCoverLabel.setText(translate("RebarAddon", "Left Cover"))
00138             self.form.bottomCoverLabel.setText(translate("RebarAddon", "Right Cover"))
00139         else:
00140             self.form.topCoverLabel.setText(translate("RebarAddon", "Top Cover"))
00141             self.form.bottomCoverLabel.setText(translate("RebarAddon", "Bottom Cover"))
00142 
00143 
```

Here is the call graph for this function:



7.2.3.4 def HelicalRebar._HelicalRebarTaskPanel.getStandardButtons (*self*)

Definition at line 126 of file HelicalRebar.py.

```
00126     def getStandardButtons(self):
00127         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00128 
```

7.2.4 Member Data Documentation

7.2.4.1 HelicalRebar._HelicalRebarTaskPanel.FaceName

Definition at line 124 of file HelicalRebar.py.

7.2.4.2 HelicalRebar._HelicalRebarTaskPanel.form

Definition at line 109 of file HelicalRebar.py.

7.2.4.3 HelicalRebar._HelicalRebarTaskPanel.Rebar

Definition at line 122 of file HelicalRebar.py.

7.2.4.4 HelicalRebar._HelicalRebarTaskPanel.SelectedObj

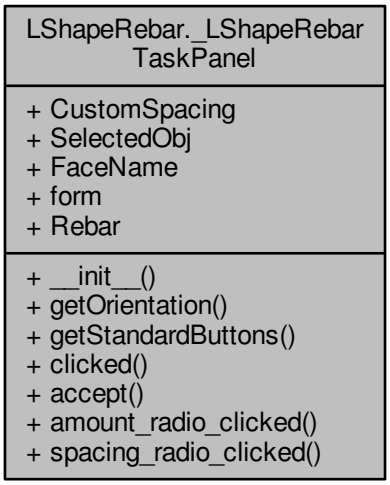
Definition at line 123 of file HelicalRebar.py.

The documentation for this class was generated from the following file:

- HelicalRebar.py

7.3 LShapeRebar._LShapeRebarTaskPanel Class Reference

Collaboration diagram for LShapeRebar._LShapeRebarTaskPanel:



Public Member Functions

- def __init__ (self, Rebar=None)
- def getOrientation (self)
- def getStandardButtons (self)
- def clicked (self, button)
- def accept (self, signal=None)
- def amount_radio_clicked (self)
- def spacing_radio_clicked (self)

Public Attributes

- CustomSpacing
- SelectedObj
- FaceName
- form
- Rebar

7.3.1 Detailed Description

Definition at line 76 of file LShapeRebar.py.

7.3.2 Constructor & Destructor Documentation

7.3.2.1 def LShapeRebar._LShapeRebarTaskPanel.__init__ (*self, Rebar* = None)

Definition at line 77 of file LShapeRebar.py.

```
00077     def __init__(self, Rebar = None):
00078         self.CustomSpacing = None
00079         if not Rebar:
00080             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00081             self.SelectedObj = selected_obj.Object
00082             self.FaceName = selected_obj.SubElementNames[0]
00083         else:
00084             self.FaceName = Rebar.Base.Support[0][1][0]
00085             self.SelectedObj = Rebar.Base.Support[0][0]
00086         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00087         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "L-Shape Rebar", None))
00088         self.form.orientation.addItems(["Bottom Right", "Bottom Left", "Top Right", "Top Left"])
00089         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00090         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00091         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00092         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00093         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00094         self.form.orientation.currentIndexChanged.connect(self.getOrientation)
00095         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarBR.svg"))
00096         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00097         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/LShapeRebarDetailed.svg"))
00098         self.Rebar = Rebar
00099
```

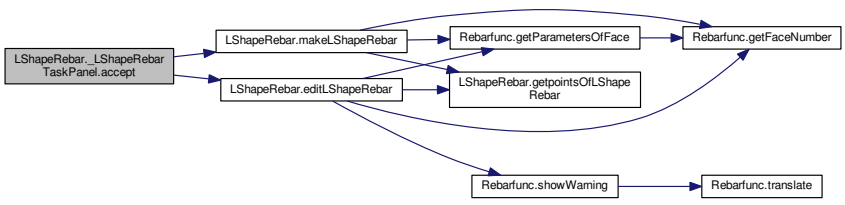
7.3.3 Member Function Documentation

7.3.3.1 def LShapeRebar._LShapeRebarTaskPanel.accept (*self, signal* = None)

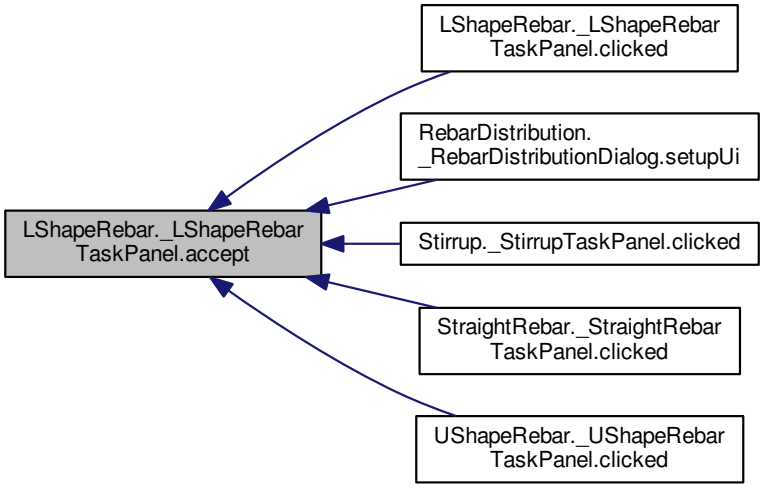
Definition at line 118 of file LShapeRebar.py.

```
00118     def accept(self, signal = None):
00119         f_cover = self.form.frontCover.text()
00120         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00121         b_cover = self.form.bottomCover.text()
00122         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00123         l_cover = self.form.l_sideCover.text()
00124         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00125         r_cover = self.form.r_sideCover.text()
00126         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00127         t_cover = self.form.topCover.text()
00128         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00129         diameter = self.form.diameter.text()
00130         diameter = FreeCAD.Units.Quantity(diameter).Value
00131         rounding = self.form.rounding.value()
00132         orientation = self.form.orientation.currentText()
00133         amount_check = self.form.amount_radio.isChecked()
00134         spacing_check = self.form.spacing_radio.isChecked()
00135         if not self.Rebar:
00136             if amount_check:
00137                 amount = self.form.amount.value()
00138                 rebar = makeLShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00139             elif spacing_check:
00140                 spacing = self.form.spacing.text()
00141                 spacing = FreeCAD.Units.Quantity(spacing).Value
00142                 rebar = makeLShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00143         else:
00144             if amount_check:
00145                 amount = self.form.amount.value()
00146                 rebar = editLShapeRebar(self.Rebar, f_cover, b_cover, l_cover, r_cover,
       diameter, t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00147             elif spacing_check:
00148                 spacing = self.form.spacing.text()
00149                 spacing = FreeCAD.Units.Quantity(spacing).Value
00150                 rebar = editLShapeRebar(self.Rebar, f_cover, b_cover, l_cover, r_cover,
       diameter, t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00151         if self.CustomSpacing:
00152             rebar.CustomSpacing = self.CustomSpacing
00153             FreeCAD.ActiveDocument.recompute()
00154         self.Rebar = rebar
00155         if signal == int(QtGui.QDialogButtonBox.Apply):
00156             pass
00157         else:
00158             FreeCADGui.Control.closeDialog(self)
00159
```

Here is the call graph for this function:



Here is the caller graph for this function:



7.3.3.2 def LShapeRebar._LShapeRebarTaskPanel.amount_radio_clicked (*self*)

Definition at line 160 of file LShapeRebar.py.

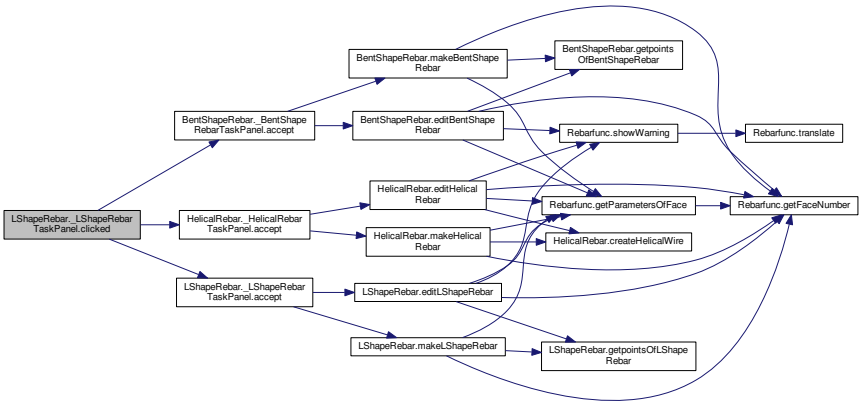
```
00160     def amount_radio_clicked(self):
00161         self.form.spacing.setEnabled(False)
00162         self.form.amount.setEnabled(True)
00163
```

7.3.3.3 def LShapeRebar._LShapeRebarTaskPanel.clicked (*self, button*)

Definition at line 114 of file LShapeRebar.py.

```
00114     def clicked(self, button):
00115         if button == int(QtGui.QDialogButtonBox.Apply):
00116             self.accept(button)
00117
```

Here is the call graph for this function:



7.3.3.4 def LShapeRebar._LShapeRebarTaskPanel.getOrientation (*self*)

Definition at line 100 of file LShapeRebar.py.

```
00100     def getOrientation(self):
00101         orientation = self.form.orientation.currentText()
00102         if orientation == "Bottom Right":
00103             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarBR.svg"))
00104         elif orientation == "Bottom Left":
00105             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarBL.svg"))
00106         elif orientation == "Top Right":
00107             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarTR.svg"))
00108         else:
00109             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarTL.svg"))
00110
```

7.3.3.5 def LShapeRebar._LShapeRebarTaskPanel.getStandardButtons (*self*)

Definition at line 111 of file LShapeRebar.py.

```
00111     def getStandardButtons(self):
00112         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00113
```

7.3.3.6 def LShapeRebar._LShapeRebarTaskPanel.spacing_radio_clicked (*self*)

Definition at line 164 of file LShapeRebar.py.

```
00164     def spacing_radio_clicked(self):
00165         self.form.amount.setEnabled(False)
00166         self.form.spacing.setEnabled(True)
00167
00168
```

7.3.4 Member Data Documentation

7.3.4.1 LShapeRebar._LShapeRebarTaskPanel.CustomSpacing

Definition at line 78 of file LShapeRebar.py.

7.3.4.2 LShapeRebar._LShapeRebarTaskPanel.FaceName

Definition at line 82 of file LShapeRebar.py.

7.3.4.3 LShapeRebar._LShapeRebarTaskPanel.form

Definition at line 86 of file LShapeRebar.py.

7.3.4.4 LShapeRebar._LShapeRebarTaskPanel.Rebar

Definition at line 98 of file LShapeRebar.py.

7.3.4.5 LShapeRebar._LShapeRebarTaskPanel.SelectedObj

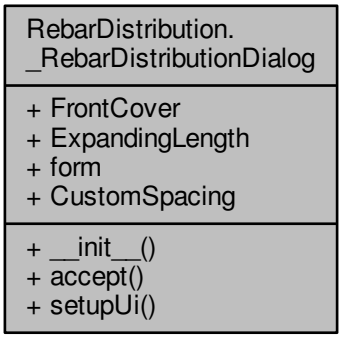
Definition at line 81 of file LShapeRebar.py.

The documentation for this class was generated from the following file:

- LShapeRebar.py

7.4 RebarDistribution._RebarDistributionDialog Class Reference

Collaboration diagram for RebarDistribution._RebarDistributionDialog:



Public Member Functions

- def __init__ (self, frontCover, size)
- def accept (self)
- def setupUi (self)

Public Attributes

- FrontCover
- ExpandingLength
- form
- CustomSpacing

7.4.1 Detailed Description

Definition at line 38 of file RebarDistribution.py.

7.4.2 Constructor & Destructor Documentation

7.4.2.1 def RebarDistribution._RebarDistributionDialog.__init__ (*self, frontCover, size*)

Definition at line 39 of file RebarDistribution.py.

```
00039     def __init__(self, frontCover, size):
00040         self.FrontCover = frontCover
00041         self.ExpandingLength = size
00042         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00043         self.form.setWindowTitle(QtGui.QApplication.translate("Arch", "Rebar Distribution", None))
00044         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/RebarDistribution.svg"))
00045
```

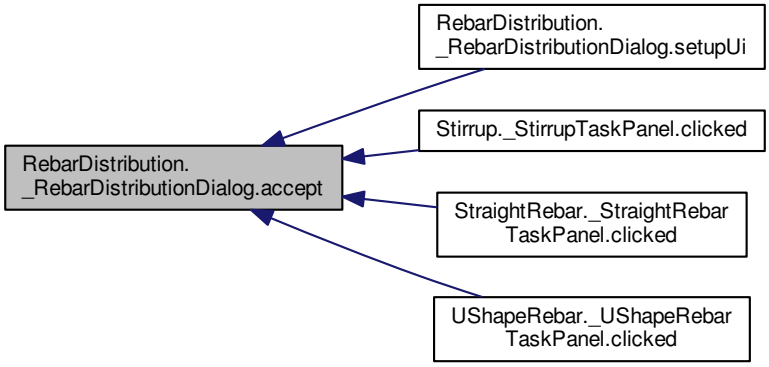
7.4.3 Member Function Documentation

7.4.3.1 def RebarDistribution._RebarDistributionDialog.accept (*self*)

Definition at line 46 of file RebarDistribution.py.

```
00046     def accept(self):
00047         amount1 = self.form.amount1.value()
00048         spacing1 = self.form.spacing1.text()
00049         spacing1 = FreeCAD.Units.Quantity(spacing1).Value
00050         amount2 = self.form.amount2.value()
00051         spacing2 = self.form.spacing2.text()
00052         spacing2 = FreeCAD.Units.Quantity(spacing2).Value
00053         amount3 = self.form.amount3.value()
00054         spacing3 = self.form.spacing3.text()
00055         spacing3 = FreeCAD.Units.Quantity(spacing3).Value
00056         self.CustomSpacing = getCustomSpacingString(amount1, spacing1,
      amount2, spacing2, amount3, spacing3, self.FrontCover, self.
      ExpandingLength)
00057
```

Here is the caller graph for this function:

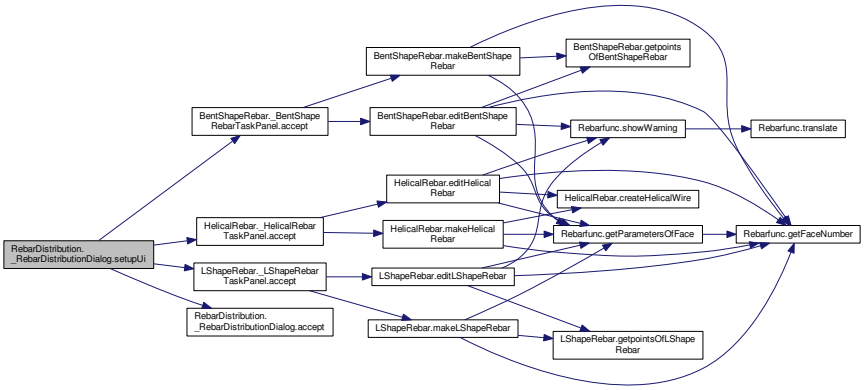


7.4.3.2 def RebarDistribution._RebarDistributionDialog.setupUi (*self*)

Definition at line 58 of file RebarDistribution.py.

```
00058     def setupUi(self):
00059         # Connect Signals and Slots
00060         self.form.buttonBox.accepted.connect(self.accept)
00061         pass
00062
```

Here is the call graph for this function:



7.4.4 Member Data Documentation

7.4.4.1 RebarDistribution._RebarDistributionDialog.CustomSpacing

Definition at line 56 of file RebarDistribution.py.

7.4.4.2 RebarDistribution._RebarDistributionDialog.ExpandingLength

Definition at line 41 of file RebarDistribution.py.

7.4.4.3 RebarDistribution._RebarDistributionDialog.form

Definition at line 42 of file RebarDistribution.py.

7.4.4.4 RebarDistribution._RebarDistributionDialog.FrontCover

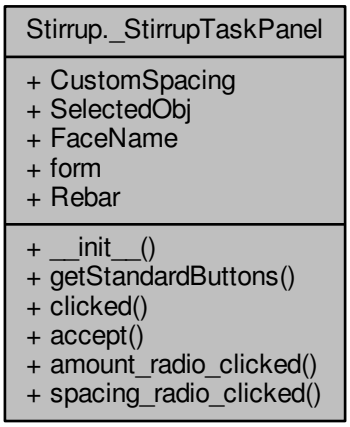
Definition at line 40 of file RebarDistribution.py.

The documentation for this class was generated from the following file:

- RebarDistribution.py

7.5 Stirrup._StirrupTaskPanel Class Reference

Collaboration diagram for Stirrup._StirrupTaskPanel:



Public Member Functions

- def __init__ (self, Rebar=None)
- def getStandardButtons (self)
- def clicked (self, button)
- def accept (self, signal=None)
- def amount_radio_clicked (self)
- def spacing_radio_clicked (self)

Public Attributes

- CustomSpacing
- SelectedObj
- FaceName
- form
- Rebar

7.5.1 Detailed Description

Definition at line 123 of file Stirrup.py.

7.5.2 Constructor & Destructor Documentation

7.5.2.1 def Stirrup._StirrupTaskPanel.__init__ (*self, Rebar* = None)

Definition at line 124 of file Stirrup.py.

```
00124     def __init__(self, Rebar = None):
00125         self.CustomSpacing = None
00126         if not Rebar:
00127             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00128             self.SelectedObj = selected_obj.Object
00129             self.FaceName = selected_obj.SubElementNames[0]
00130         else:
00131             self.FaceName = Rebar.Base.Support[0][1][0]
00132             self.SelectedObj = Rebar.Base.Support[0][0]
00133         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00134         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "Stirrup Rebar", None))
00135         self.form.bentAngle.addItems(["135", "90"])
00136         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00137         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00138         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0]+"
      /icons/Stirrup.svg"))
00139         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00140         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00141         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00142         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00143         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/StirrupDetailed.svg"))
00144         self.Rebar = Rebar
00145
```

7.5.3 Member Function Documentation

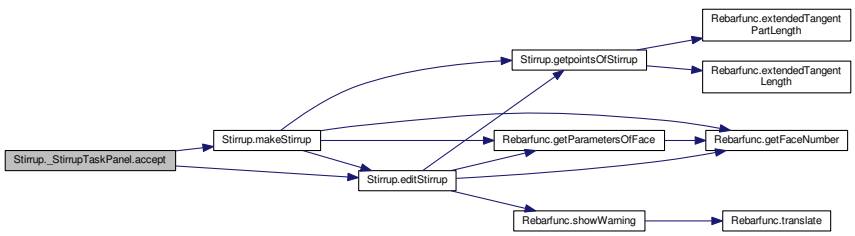
7.5.3.1 def Stirrup._StirrupTaskPanel.accept (*self, signal* = None)

Definition at line 153 of file Stirrup.py.

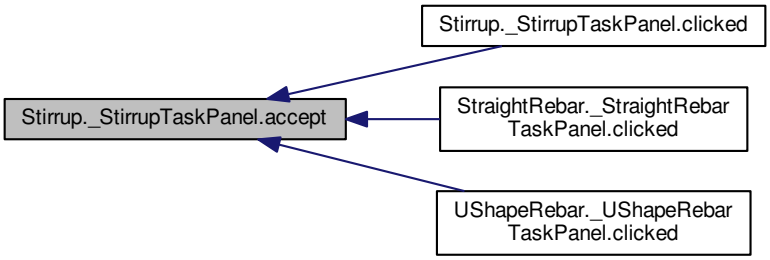
```
00153     def accept(self, signal = None):
00154         l_cover = self.form.l_sideCover.text()
00155         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00156         r_cover = self.form.r_sideCover.text()
00157         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00158         t_cover = self.form.t_sideCover.text()
00159         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00160         b_cover = self.form.b_sideCover.text()
00161         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00162         f_cover = self.form.frontCover.text()
00163         f_cover = FreeCAD.Units.Quantity(f_cover).Value
```

```
00164         diameter = self.form.diameter.text()
00165         diameter = FreeCAD.Units.Quantity(diameter).Value
00166         bentAngle = int(self.form.bentAngle.currentText())
00167         bentFactor = self.form.bentFactor.value()
00168         rounding = self.form.rounding.value()
00169         amount_check = self.form.amount_radio.isChecked()
00170         spacing_check = self.form.spacing_radio.isChecked()
00171         if not self.Rebar:
00172             if amount_check:
00173                 amount = self.form.amount.value()
00174                 rebar = makeStirrup(l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle,
      bentFactor, diameter,\
00175                     rounding, True, amount, self.SelectedObj, self.
      FaceName)
00176             elif spacing_check:
00177                 spacing = self.form.spacing.text()
00178                 spacing = FreeCAD.Units.Quantity(spacing).Value
00179                 rebar = makeStirrup(l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle,
      bentFactor, diameter,\
00180                     rounding, False, spacing, self.SelectedObj, self.
      FaceName)
00181         else:
00182             if amount_check:
00183                 amount = self.form.amount.value()
00184                 rebar = editStirrup(self.Rebar, l_cover, r_cover, t_cover, b_cover, f_cover
      , bentAngle, bentFactor,\
00185                     diameter, rounding, True, amount, self.SelectedObj, self.
      FaceName)
00186             elif spacing_check:
00187                 spacing = self.form.spacing.text()
00188                 spacing = FreeCAD.Units.Quantity(spacing).Value
00189                 rebar = editStirrup(self.Rebar, l_cover, r_cover, t_cover, b_cover, f_cover
      , bentAngle, bentFactor,\
00190                     diameter, rounding, False, spacing, self.SelectedObj, self.
      FaceName)
00191         if self.CustomSpacing:
00192             rebar.CustomSpacing = self.CustomSpacing
00193             FreeCAD.ActiveDocument.recompute()
00194         self.Rebar = rebar
00195         if signal == int(QtGui.QDialogButtonBox.Apply):
00196             pass
00197         else:
00198             FreeCADGui.Control.closeDialog(self)
00199
```

Here is the call graph for this function:



Here is the caller graph for this function:



7.5.3.2 def Stirrup._StirrupTaskPanel.amount_radio_clicked (*self*)

Definition at line 200 of file Stirrup.py.

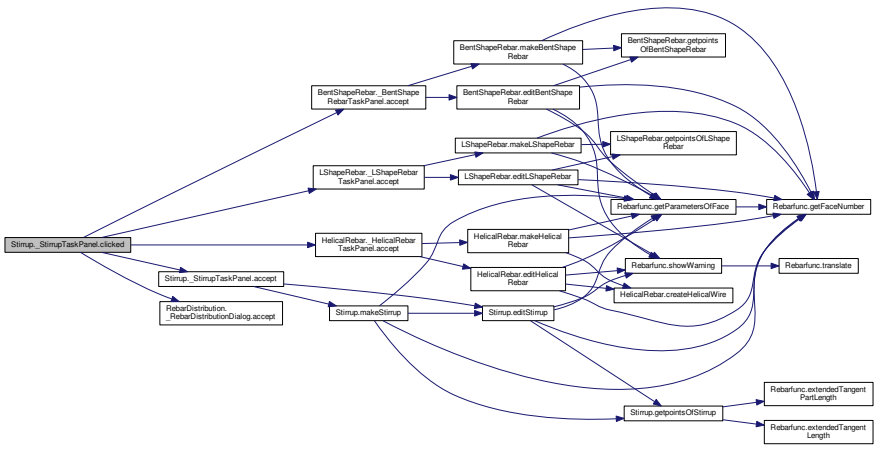
```
00200     def amount_radio_clicked(self):
00201         self.form.spacing.setEnabled(False)
00202         self.form.amount.setEnabled(True)
00203
```

7.5.3.3 def Stirrup._StirrupTaskPanel.clicked (*self, button*)

Definition at line 149 of file Stirrup.py.

```
00149     def clicked(self, button):
00150         if button == int(QtGui.QDialogButtonBox.Apply):
00151             self.accept(button)
00152
```

Here is the call graph for this function:



7.5.3.4 def Stirrup._StirrupTaskPanel.getStandardButtons (*self*)

Definition at line 146 of file Stirrup.py.

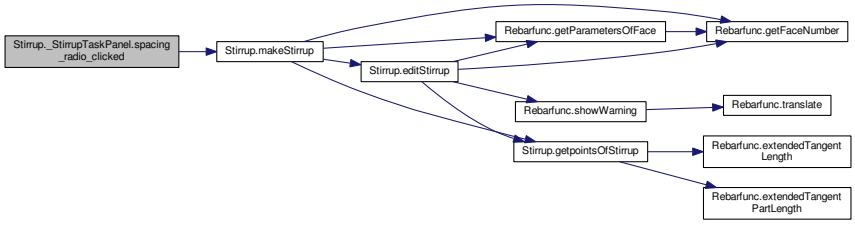
```
00146     def getStandardButtons(self):
00147         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00148
```

7.5.3.5 def Stirrup._StirrupTaskPanel.spacing_radio_clicked (*self*)

Definition at line 204 of file Stirrup.py.

```
00204     def spacing_radio_clicked(self):
00205         self.form.amount.setEnabled(False)
00206         self.form.spacing.setEnabled(True)
00207
00208
```

Here is the call graph for this function:



7.5.4 Member Data Documentation

7.5.4.1 Stirrup._StirrupTaskPanel.CustomSpacing

Definition at line 125 of file Stirrup.py.

7.5.4.2 Stirrup._StirrupTaskPanel.FaceName

Definition at line 129 of file Stirrup.py.

7.5.4.3 Stirrup._StirrupTaskPanel.form

Definition at line 133 of file Stirrup.py.

7.5.4.4 Stirrup._StirrupTaskPanel.Rebar

Definition at line 144 of file Stirrup.py.

7.5.4.5 Stirrup._StirrupTaskPanel.SelectedObj

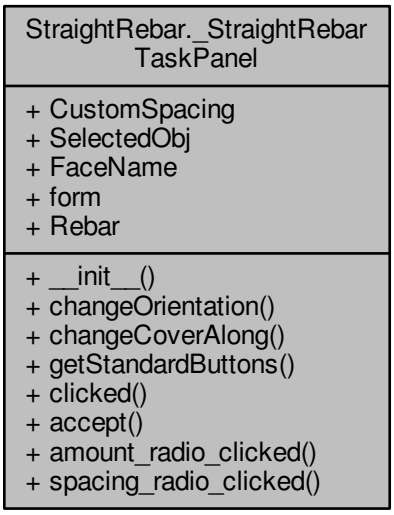
Definition at line 128 of file Stirrup.py.

The documentation for this class was generated from the following file:

- Stirrup.py

7.6 StraightRebar._StraightRebarTaskPanel Class Reference

Collaboration diagram for StraightRebar._StraightRebarTaskPanel:



Public Member Functions

- def __init__ (self, Rebar=None)
- def changeOrientation (self)
- def changeCoverAlong (self)
- def getStandardButtons (self)
- def clicked (self, button)
- def accept (self, signal=None)
- def amount_radio_clicked (self)
- def spacing_radio_clicked (self)

Public Attributes

- CustomSpacing
- SelectedObj
- FaceName
- form
- Rebar

7.6.1 Detailed Description

Definition at line 75 of file StraightRebar.py.

7.6.2 Constructor & Destructor Documentation

7.6.2.1 def StraightRebar._StraightRebarTaskPanel.__init__ (*self, Rebar* = None)

Definition at line 76 of file StraightRebar.py.

```
00076     def __init__(self, Rebar = None):
00077         self.CustomSpacing = None
00078         if not Rebar:
00079             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00080             self.SelectedObj = selected_obj.Object
00081             self.FaceName = selected_obj.SubElementNames[0]
00082         else:
00083             self.FaceName = Rebar.Base.Support[0][1][0]
00084             self.SelectedObj = Rebar.Base.Support[0][0]
00085         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00086         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "Straight Rebar", None))
00087         self.form.orientation.addItems(["Horizontal", "Vertical"])
00088         self.form.coverAlong.addItems(["Bottom Side", "Top Side"])
00089         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00090         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00091         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00092         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00093         self.form.PickSelectedFace.setCheckable(True)
00094         self.form.PickSelectedFace.toggle()
00095         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00096         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/StraightRebarH.svg"))
00097         self.form.orientation.currentIndexChanged.connect(self.changeOrientation)
00098         self.form.coverAlong.currentIndexChanged.connect(self.changeCoverAlong)
00099         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00100         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/StraightRebarDetailed.svg"))
00101         self.Rebar = Rebar
00102
```

7.6.3 Member Function Documentation

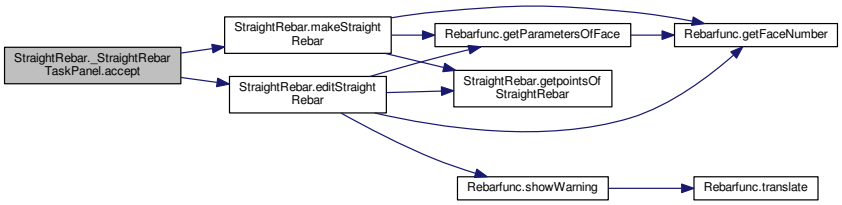
7.6.3.1 def StraightRebar._StraightRebarTaskPanel.accept (*self, signal* = None)

Definition at line 136 of file StraightRebar.py.

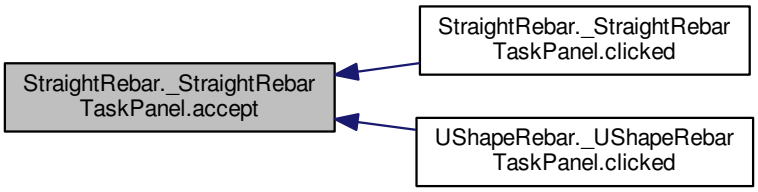
```
00136     def accept(self, signal = None):
00137         f_cover = self.form.frontCover.text()
00138         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00139         cover = self.form.bottomCover.text()
00140         cover = FreeCAD.Units.Quantity(cover).Value
00141         lb_cover = self.form.l_sideCover.text()
00142         lb_cover = FreeCAD.Units.Quantity(lb_cover).Value
00143         rt_cover = self.form.r_sideCover.text()
00144         rt_cover = FreeCAD.Units.Quantity(rt_cover).Value
00145         orientation = self.form.orientation.currentText()
00146         coverAlong = self.form.coverAlong.currentText()
00147         diameter = self.form.diameter.text()
00148         diameter = FreeCAD.Units.Quantity(diameter).Value
00149         amount_check = self.form.amount_radio.isChecked()
00150         spacing_check = self.form.spacing_radio.isChecked()
00151         if not self.Rebar:
00152             if amount_check:
00153                 amount = self.form.amount.value()
00154                 rebar = makeStraightRebar(f_cover, (coverAlong, cover), rt_cover, lb_cover
      , diameter, True, amount, orientation, self.SelectedObj, self.FaceName)
00155             elif spacing_check:
```

```
00156                 spacing = self.form.spacing.text()
00157                 spacing = FreeCAD.Units.Quantity(spacing).Value
00158                 rebar = makeStraightRebar(f_cover, (coverAlong, cover), rt_cover, lb_cover
      , diameter, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00159         else:
00160             if amount_check:
00161                 amount = self.form.amount.value()
00162                 rebar = editStraightRebar(self.Rebar, f_cover, (coverAlong, cover),
      rt_cover, lb_cover, diameter, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00163             elif spacing_check:
00164                 spacing = self.form.spacing.text()
00165                 spacing = FreeCAD.Units.Quantity(spacing).Value
00166                 rebar = editStraightRebar(self.Rebar, f_cover, (coverAlong, cover),
      rt_cover, lb_cover, diameter, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00167         if self.CustomSpacing:
00168             rebar.CustomSpacing = self.CustomSpacing
00169             FreeCAD.ActiveDocument.recompute()
00170         self.Rebar = rebar
00171         if signal == int(QtGui.QDialogButtonBox.Apply):
00172             pass
00173         else:
00174             FreeCADGui.Control.closeDialog(self)
00175
```

Here is the call graph for this function:



Here is the caller graph for this function:



7.6.3.2 def StraightRebar._StraightRebarTaskPanel.amount_radio_clicked (*self*)

Definition at line 176 of file StraightRebar.py.

```
00176     def amount_radio_clicked(self):
00177         self.form.spacing.setEnabled(False)
00178         self.form.amount.setEnabled(True)
00179
```

7.6.3.3 def StraightRebar._StraightRebarTaskPanel.changeCoverAlong (*self*)

Definition at line 118 of file StraightRebar.py.

```
00118     def changeCoverAlong(self):
00119         coverAlong = self.form.coverAlong.currentText()
00120         if coverAlong == "Bottom Side":
00121             self.form.bottomCoverLabel.setText("Bottom Cover")
00122         elif coverAlong == "Top Side":
00123             self.form.bottomCoverLabel.setText("Top Cover")
00124         elif coverAlong == "Right Side":
00125             self.form.bottomCoverLabel.setText("Right Cover")
00126         else:
00127             self.form.bottomCoverLabel.setText("Left Cover")
00128
```

7.6.3.4 def StraightRebar._StraightRebarTaskPanel.changeOrientation (*self*)

Definition at line 103 of file StraightRebar.py.

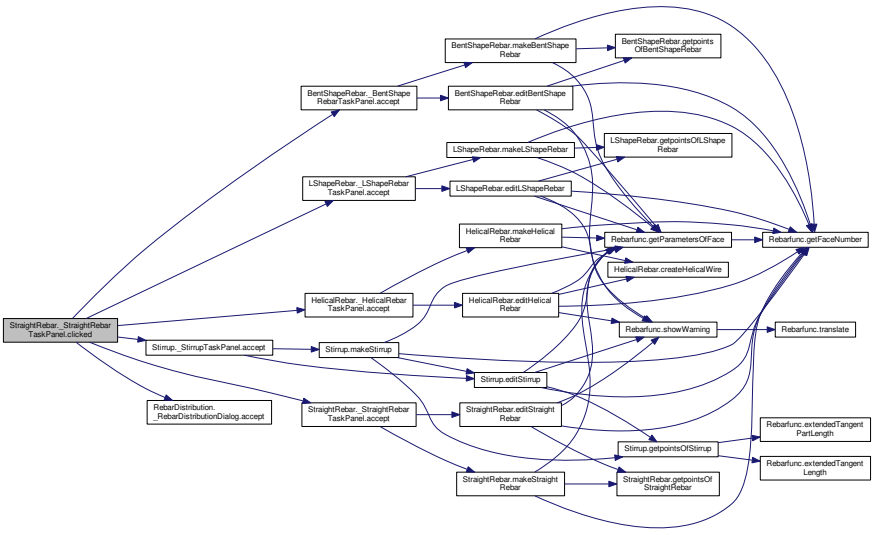
```
00103     def changeOrientation(self):
00104         orientation = self.form.orientation.currentText()
00105         if orientation == "Horizontal":
00106             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/StraightRebarH.svg"))
00107             self.form.r_sideCoverLabel.setText("Right Side Cover")
00108             self.form.l_sideCoverLabel.setText("Left Side Cover")
00109             self.form.coverAlong.clear()
00110             self.form.coverAlong.addItems(["Bottom Side", "Top Side"])
00111         else:
00112             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/StraightRebarV.svg"))
00113             self.form.r_sideCoverLabel.setText("Top Side Cover")
00114             self.form.l_sideCoverLabel.setText("Bottom Side Cover")
00115             self.form.coverAlong.clear()
00116             self.form.coverAlong.addItems(["Right Side", "Left Side"])
00117
```

7.6.3.5 def StraightRebar._StraightRebarTaskPanel.clicked (*self, button*)

Definition at line 132 of file StraightRebar.py.

```
00132     def clicked(self, button):
00133         if button == int(QtGui.QDialogButtonBox.Apply):
00134             self.accept(button)
00135
```

Here is the call graph for this function:



7.6.3.6 def StraightRebar._StraightRebarTaskPanel.getStandardButtons (*self*)

Definition at line 129 of file StraightRebar.py.

```
00129     def getStandardButtons(self):
00130         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00131
```

7.6.3.7 def StraightRebar._StraightRebarTaskPanel.spacing_radio_clicked (*self*)

Definition at line 180 of file StraightRebar.py.

```
00180     def spacing_radio_clicked(self):
00181         self.form.amount.setEnabled(False)
00182         self.form.spacing.setEnabled(True)
00183
00184
```

7.6.4 Member Data Documentation

7.6.4.1 StraightRebar._StraightRebarTaskPanel.CustomSpacing

Definition at line 77 of file StraightRebar.py.

7.6.4.2 StraightRebar._StraightRebarTaskPanel.FaceName

Definition at line 81 of file StraightRebar.py.

7.6.4.3 StraightRebar._StraightRebarTaskPanel.form

Definition at line 85 of file StraightRebar.py.

7.6.4.4 StraightRebar._StraightRebarTaskPanel.Rebar

Definition at line 101 of file StraightRebar.py.

7.6.4.5 StraightRebar._StraightRebarTaskPanel.SelectedObj

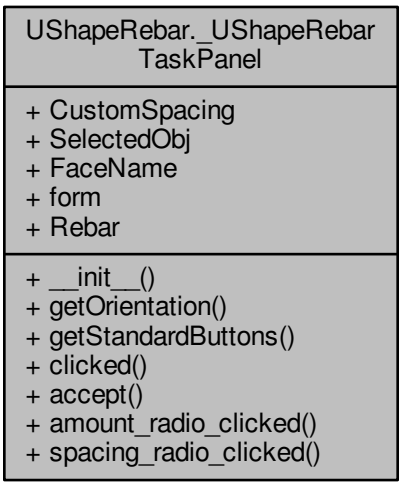
Definition at line 80 of file StraightRebar.py.

The documentation for this class was generated from the following file:

- StraightRebar.py

7.7 UShapeRebar._UShapeRebarTaskPanel Class Reference

Collaboration diagram for UShapeRebar._UShapeRebarTaskPanel:



Public Member Functions

- def __init__ (self, Rebar=None)
- def getOrientation (self)
- def getStandardButtons (self)
- def clicked (self, button)
- def accept (self, signal=None)
- def amount_radio_clicked (self)
- def spacing_radio_clicked (self)

Public Attributes

- CustomSpacing
- SelectedObj
- FaceName
- form
- Rebar

7.7.1 Detailed Description

Definition at line 84 of file UShapeRebar.py.

7.7.2 Constructor & Destructor Documentation

7.7.2.1 def UShapeRebar._UShapeRebarTaskPanel.__init__ (*self, Rebar* = None)

Definition at line 85 of file UShapeRebar.py.

```
00085     def __init__(self, Rebar = None):
00086         self.CustomSpacing = None
00087         if not Rebar:
00088             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00089             self.SelectedObj = selected_obj.Object
00090             self.FaceName = selected_obj.SubElementNames[0]
00091         else:
00092             self.FaceName = Rebar.Base.Support[0][1][0]
00093             self.SelectedObj = Rebar.Base.Support[0][0]
00094         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00095         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "U-Shape Rebar", None))
00096         self.form.orientation.addItems(["Bottom", "Top", "Right", "Left"])
00097         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00098         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00099         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00100         self.form.removeCustomSpacing.clicked.connect(lambda:
    removeRebarDistribution(self))
00101         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00102         self.form.orientation.currentIndexChanged.connect(self.getOrientation)
00103         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
    /icons/UShapeRebarBottom.svg"))
00104         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
    QtGui.QStyle.SP_DialogHelpButton))
00105         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
    (os.path.abspath(__file__))[0] + "/icons/UShapeRebarDetailed.svg"))
00106         self.Rebar = Rebar
00107
```

7.7.3 Member Function Documentation

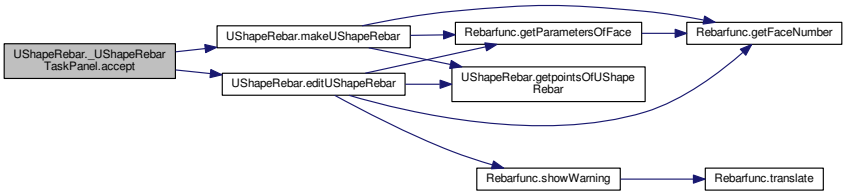
7.7.3.1 def UShapeRebar._UShapeRebarTaskPanel.accept (*self, signal* = None)

Definition at line 126 of file UShapeRebar.py.

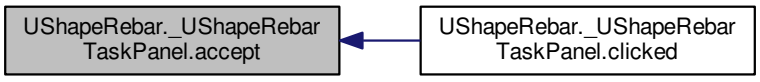
```
00126     def accept(self, signal = None):
00127         f_cover = self.form.frontCover.text()
00128         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00129         b_cover = self.form.bottomCover.text()
00130         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00131         r_cover = self.form.r_sideCover.text()
00132         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00133         l_cover = self.form.l_sideCover.text()
00134         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00135         t_cover = self.form.topCover.text()
00136         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00137         diameter = self.form.diameter.text()
00138         diameter = FreeCAD.Units.Quantity(diameter).Value
00139         rounding = self.form.rounding.value()
00140         orientation = self.form.orientation.currentText()
00141         amount_check = self.form.amount_radio.isChecked()
00142         spacing_check = self.form.spacing_radio.isChecked()
00143         if not self.Rebar:
00144             if amount_check:
00145                 amount = self.form.amount.value()
00146                 rebar = makeUShapeRebar(f_cover, b_cover, r_cover, l_cover, diameter,
    t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
    FaceName)
00147             elif spacing_check:
00148                 spacing = self.form.spacing.text()
00149                 spacing = FreeCAD.Units.Quantity(spacing).Value
00150                 rebar = makeUShapeRebar(f_cover, b_cover, r_cover, l_cover, diameter,
    t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
    FaceName)
00151         else:
00152             if amount_check:
00153                 amount = self.form.amount.value()
00154                 rebar = editUShapeRebar(self.Rebar, f_cover, b_cover, r_cover, l_cover,
     diameter, t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
```

```
      FaceName)
00155             elif spacing_check:
00156                 spacing = self.form.spacing.text()
00157                 spacing = FreeCAD.Units.Quantity(spacing).Value
00158                 rebar = editUShapeRebar(self.Rebar, f_cover, b_cover, r_cover, l_cover,
       diameter, t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00159         if self.CustomSpacing:
00160             rebar.CustomSpacing = self.CustomSpacing
00161             FreeCAD.ActiveDocument.recompute()
00162         self.Rebar = rebar
00163         if signal == int(QtGui.QDialogButtonBox.Apply):
00164             pass
00165         else:
00166             FreeCADGui.Control.closeDialog(self)
00167
```

Here is the call graph for this function:



Here is the caller graph for this function:



7.7.3.2 def UShapeRebar._UShapeRebarTaskPanel.amount_radio_clicked (*self*)

Definition at line 168 of file UShapeRebar.py.

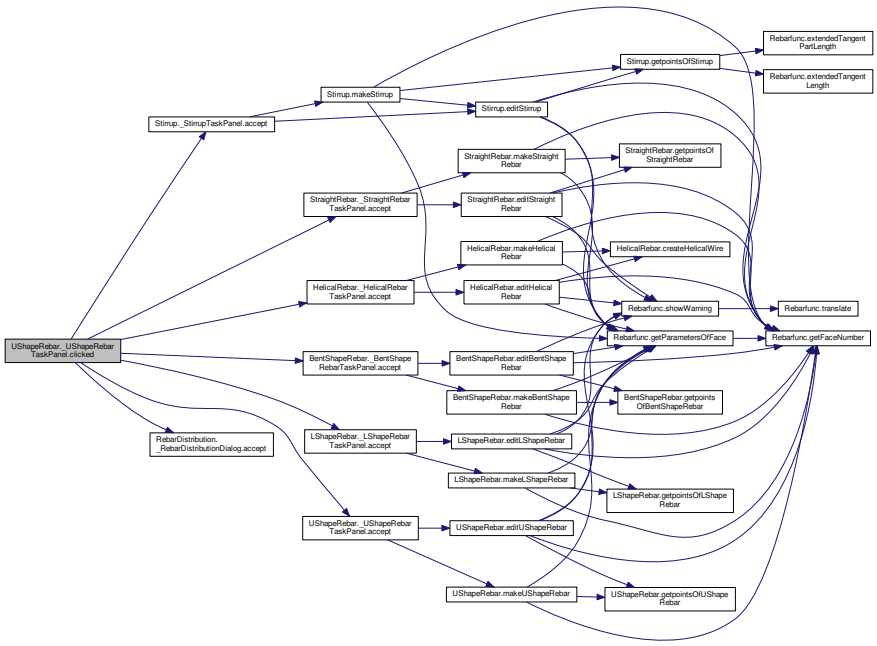
```
00168     def amount_radio_clicked(self):
00169         self.form.spacing.setEnabled(False)
00170         self.form.amount.setEnabled(True)
00171
```

7.7.3.3 def UShapeRebar._UShapeRebarTaskPanel.clicked (*self, button*)

Definition at line 122 of file UShapeRebar.py.

```
00122     def clicked(self, button):
00123         if button == int(QtGui.QDialogButtonBox.Apply):
00124             self.accept(button)
00125
```

Here is the call graph for this function:



7.7.3.4 def UShapeRebar._UShapeRebarTaskPanel.getOrientation (*self*)

Definition at line 108 of file UShapeRebar.py.

```
00108     def getOrientation(self):
00109         orientation = self.form.orientation.currentText()
00110         if orientation == "Bottom":
00111             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarBottom.svg"))
00112         elif orientation == "Top":
00113             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarTop.svg"))
00114         elif orientation == "Right":
00115             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarRight.svg"))
00116         else:
00117             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarLeft.svg"))
00118
```

7.7.3.5 def UShapeRebar._UShapeRebarTaskPanel.getStandardButtons (*self*)

Definition at line 119 of file UShapeRebar.py.

```
00119     def getStandardButtons(self):
00120         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00121
```

7.7.3.6 def UShapeRebar._UShapeRebarTaskPanel.spacing_radio_clicked (*self*)

Definition at line 172 of file UShapeRebar.py.

```
00172     def spacing_radio_clicked(self):
00173         self.form.amount.setEnabled(False)
00174         self.form.spacing.setEnabled(True)
00175
00176
```

7.7.4 Member Data Documentation

7.7.4.1 UShapeRebar._UShapeRebarTaskPanel.CustomSpacing

Definition at line 86 of file UShapeRebar.py.

7.7.4.2 UShapeRebar._UShapeRebarTaskPanel.FaceName

Definition at line 90 of file UShapeRebar.py.

7.7.4.3 UShapeRebar._UShapeRebarTaskPanel.form

Definition at line 94 of file UShapeRebar.py.

7.7.4.4 UShapeRebar._UShapeRebarTaskPanel.Rebar

Definition at line 106 of file UShapeRebar.py.

7.7.4.5 UShapeRebar._UShapeRebarTaskPanel.SelectedObj

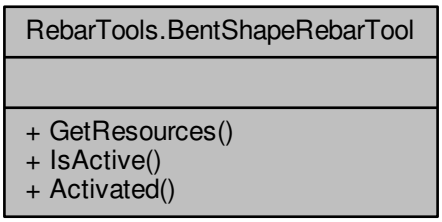
Definition at line 89 of file UShapeRebar.py.

The documentation for this class was generated from the following file:

- UShapeRebar.py

7.8 RebarTools.BentShapeRebarTool Class Reference

Collaboration diagram for RebarTools.BentShapeRebarTool:



Public Member Functions

- def GetResources (self)
- def IsActive (self)
- def Activated (self)

7.8.1 Detailed Description

Definition at line 104 of file RebarTools.py.

7.8.2 Member Function Documentation

7.8.2.1 def RebarTools.BentShapeRebarTool.Activated (*self*)

Definition at line 117 of file RebarTools.py.

```
00117     def Activated(self):
00118         import BentShapeRebar
00119         # Call to CommandBentShaepRebar() function
00120         BentShapeRebar.CommandBentShapeRebar()
00121
```

Here is the call graph for this function:

7.8.2.2 def RebarTools.BentShapeRebarTool.GetResources (*self*)

Definition at line 106 of file RebarTools.py.

```
00106     def GetResources(self):
00107         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/BentShapeRebar.svg',
00108                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Bent-Shape Rebar"),
00109                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a BentShape bar reinforcement from the
       selected face of the Structural element.")}
00110
```

7.8.2.3 def RebarTools.BentShapeRebarTool.IsActive (*self*)

Definition at line 111 of file RebarTools.py.

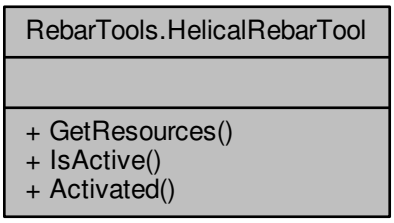
```
00111     def IsActive(self):
00112         if FreeCADGui.ActiveDocument:
00113             return True
00114         else:
00115             return False
00116
```

The documentation for this class was generated from the following file:

- RebarTools.py

7.9 RebarTools.HelicalRebarTool Class Reference

Collaboration diagram for RebarTools.HelicalRebarTool:



Public Member Functions

- def GetResources (self)
- def IsActive (self)
- def Activated (self)

7.9.1 Detailed Description

Definition at line 122 of file RebarTools.py.

7.9.2 Member Function Documentation

7.9.2.1 def RebarTools.HelicalRebarTool.Activated (*self*)

Definition at line 135 of file RebarTools.py.

```
00135     def Activated(self):
00136         import HelicalRebar
00137         # Call to CommandHelicalRebar() function
00138         HelicalRebar.CommandHelicalRebar()
00139 
00140 FreeCADGui.addCommand('Arch_Rebar_Straight', StraightRebarTool())
00141 FreeCADGui.addCommand('Arch_Rebar_UShape', UShapeRebarTool())
00142 FreeCADGui.addCommand('Arch_Rebar_LShape', LShapeRebarTool())
00143 FreeCADGui.addCommand('Arch_Rebar_Stirrup', StirrupTool())
00144 FreeCADGui.addCommand('Arch_Rebar_BentShape', BentShapeRebarTool())
00145 FreeCADGui.addCommand('Arch_Rebar_Helical', HelicalRebarTool())
00146 
00147 # List of all rebar commands
```

Here is the call graph for this function:

7.9.2.2 def RebarTools.HelicalRebarTool.GetResources (*self*)

Definition at line 124 of file RebarTools.py.

```
00124     def GetResources(self):
00125         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
    /icons/dropdown_list/HelixShapeRebar.svg',
00126                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Helical Rebar"),
00127                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a Helical bar reinforcement from the
     selected face of the Structural element.")}
00128
```

7.9.2.3 def RebarTools.HelicalRebarTool.IsActive (*self*)

Definition at line 129 of file RebarTools.py.

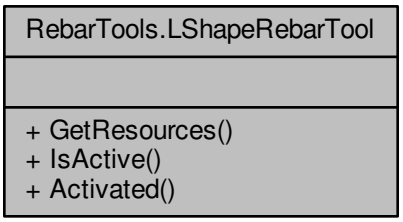
```
00129     def IsActive(self):
00130         if FreeCADGui.ActiveDocument:
00131             return True
00132         else:
00133             return False
00134
```

The documentation for this class was generated from the following file:

- RebarTools.py

7.10 RebarTools.LShapeRebarTool Class Reference

Collaboration diagram for RebarTools.LShapeRebarTool:



Public Member Functions

- def GetResources (self)
- def IsActive (self)
- def Activated (self)

7.10.1 Detailed Description

Definition at line 68 of file RebarTools.py.

7.10.2 Member Function Documentation

7.10.2.1 def RebarTools.LShapeRebarTool.Activated (*self*)

Definition at line 81 of file RebarTools.py.

```
00081     def Activated(self):
00082         import LShapeRebar
00083         # Call to CommandUShaepRebar() function
00084         LShapeRebar.CommandLShapeRebar()
00085
```

Here is the call graph for this function:

7.10.2.2 def RebarTools.LShapeRebarTool.GetResources (*self*)

Definition at line 70 of file RebarTools.py.

```
00070     def GetResources(self):
00071         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/LShapeRebar.svg',
00072                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "L-Shape Rebar"),
00073                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a L-Shape bar reinforcement from the
       selected face of the Structural element.")}
00074
```

7.10.2.3 def RebarTools.LShapeRebarTool.IsActive (*self*)

Definition at line 75 of file RebarTools.py.

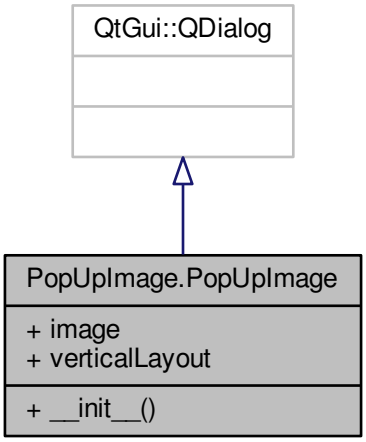
```
00075     def IsActive(self):
00076         if FreeCADGui.ActiveDocument:
00077             return True
00078         else:
00079             return False
00080
```

The documentation for this class was generated from the following file:

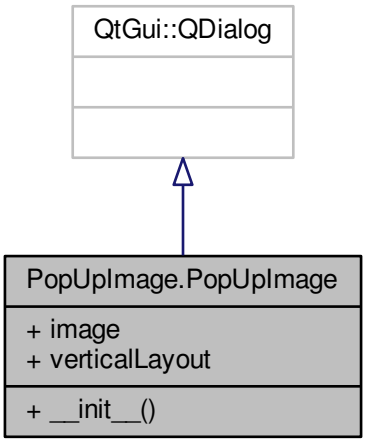
- RebarTools.py

7.11 PopUpImage.PopUpImage Class Reference

Inheritance diagram for PopUpImage.PopUpImage:



Collaboration diagram for PopUpImage.PopUpImage:



Public Member Functions

- def __init__ (self, img)

Public Attributes

- image
- verticalLayout

7.11.1 Detailed Description

Definition at line 35 of file PopUpImage.py.

7.11.2 Constructor & Destructor Documentation

7.11.2.1 def PopUpImage.PopUpImage.__init__ (*self, img*)

Definition at line 36 of file PopUpImage.py.

```
00036     def __init__(self, img):
00037         QtGui.QDialog.__init__(self)
00038         self.image = QtSvg.QSvgWidget(img)
00039         self.setWindowTitle(QtGui.QApplication.translate("RebarTool", "Detailed description", None))
00040         self.verticalLayout = QtGui.QVBoxLayout(self)
00041         self.verticalLayout.addWidget(self.image)
00042
```

7.11.3 Member Data Documentation

7.11.3.1 PopUpImage.PopUpImage.image

Definition at line 38 of file PopUpImage.py.

7.11.3.2 PopUpImage.PopUpImage.verticalLayout

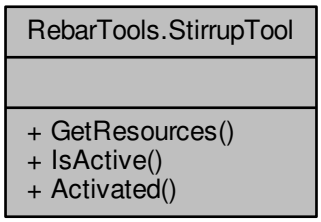
Definition at line 40 of file PopUpImage.py.

The documentation for this class was generated from the following file:

- PopUpImage.py

7.12 RebarTools.StirrupTool Class Reference

Collaboration diagram for RebarTools.StirrupTool:



Public Member Functions

- def GetResources (self)
- def IsActive (self)
- def Activated (self)

7.12.1 Detailed Description

Definition at line 86 of file RebarTools.py.

7.12.2 Member Function Documentation

7.12.2.1 def RebarTools.StirrupTool.Activated (*self*)

Definition at line 99 of file RebarTools.py.

```
00099     def Activated(self):
00100         import Stirrup
00101         # Call to CommandStirrup() function
00102         Stirrup.CommandStirrup()
00103
```

Here is the call graph for this function:

7.12.2.2 def RebarTools.StirrupTool.GetResources (*self*)

Definition at line 88 of file RebarTools.py.

```
00088     def GetResources(self):
00089         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/StirrupRebar.svg',
00090                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Stirrup"),
00091                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a Stirrup bar reinforcement from the
       selected face of the Structural element.")}
00092
```

7.12.2.3 def RebarTools.StirrupTool.IsActive (*self*)

Definition at line 93 of file RebarTools.py.

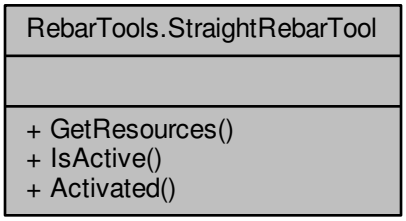
```
00093     def IsActive(self):
00094         if FreeCADGui.ActiveDocument:
00095             return True
00096         else:
00097             return False
00098
```

The documentation for this class was generated from the following file:

- RebarTools.py

7.13 RebarTools.StraightRebarTool Class Reference

Collaboration diagram for RebarTools.StraightRebarTool:



Public Member Functions

- def GetResources (self)
- def IsActive (self)
- def Activated (self)

7.13.1 Detailed Description

Definition at line 32 of file RebarTools.py.

7.13.2 Member Function Documentation

7.13.2.1 def RebarTools.StraightRebarTool.Activated (*self*)

Definition at line 45 of file RebarTools.py.

```
00045     def Activated(self):
00046         import StraightRebar
00047         # Call to CommandStraightRebar() function
00048         StraightRebar.CommandStraightRebar()
00049
```

Here is the call graph for this function:

7.13.2.2 def RebarTools.StraightRebarTool.GetResources (*self*)

Definition at line 34 of file RebarTools.py.

```
00034     def GetResources(self):
00035         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
    /icons/dropdown_list/StraightRebar.svg',
00036                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Straight Rebar"),
00037                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a Striaght bar reinforcement from the
     selected face of the Structural element.")}
00038
```

7.13.2.3 def RebarTools.StraightRebarTool.IsActive (*self*)

Definition at line 39 of file RebarTools.py.

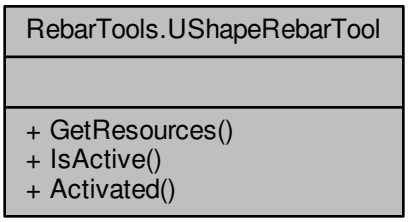
```
00039     def IsActive(self):
00040         if FreeCADGui.ActiveDocument:
00041             return True
00042         else:
00043             return False
00044
```

The documentation for this class was generated from the following file:

- RebarTools.py

7.14 RebarTools.UShapeRebarTool Class Reference

Collaboration diagram for RebarTools.UShapeRebarTool:



Public Member Functions

- def GetResources (self)
- def IsActive (self)
- def Activated (self)

7.14.1 Detailed Description

Definition at line 50 of file RebarTools.py.

7.14.2 Member Function Documentation

7.14.2.1 def RebarTools.UShapeRebarTool.Activated (*self*)

Definition at line 63 of file RebarTools.py.

```
00063     def Activated(self):
00064         import UShapeRebar
00065         # Call to CommandUShaepRebar() function
00066         UShapeRebar.CommandUShapeRebar()
00067
```

Here is the call graph for this function:

7.14.2.2 def RebarTools.UShapeRebarTool.GetResources (*self*)

Definition at line 52 of file RebarTools.py.

```
00052     def GetResources(self):
00053         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
    /icons/dropdown_list/UShapeRebar.svg',
00054                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "U-Shape Rebar"),
00055                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a U-Shape bar reinforcement from the
     selected face of the Structural element.")}
00056
```

7.14.2.3 def RebarTools.UShapeRebarTool.IsActive (*self*)

Definition at line 57 of file RebarTools.py.

```
00057     def IsActive(self):
00058         if FreeCADGui.ActiveDocument:
00059             return True
00060         else:
00061             return False
00062
```

The documentation for this class was generated from the following file:

- RebarTools.py

Chapter 8

File Documentation

8.1 BentShapeRebar.py File Reference

Classes

- class BentShapeRebar._BentShapeRebarTaskPanel

Namespaces

- BentShapeRebar

Functions

- def BentShapeRebar.getpointsOfBentShapeRebar (FacePRM, l_cover, r_cover, b_cover, t_cover, bentLength, bentAngle, orientation)
- def BentShapeRebar.makeBentShapeRebar (f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation="Bottom Left", structure=None, facename=None)
- def BentShapeRebar.editBentShapeRebar (Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def BentShapeRebar.editDialog (vobj)
- def BentShapeRebar.CommandBentShapeRebar ()

Variables

- string BentShapeRebar.__title__ = "BentShapeRebar"
- string BentShapeRebar.__author__ = "Amritpal Singh"
- string BentShapeRebar.__url__ = "https://www.freecadweb.org"

8.2 BentShapeRebar.py

```
00001 # -*- coding: utf-8 -*-
00002 # ****************************************************************************
00003 # *                                                                          *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>              *
00005 # *                                                                          *
00006 # *   This program is free software; you can redistribute it and/or modify   *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)     *
00008 # *   as published by the Free Software Foundation; either version 2 of      *
00009 # *   the License, or (at your option) any later version.                    *
00010 # *   for detail see the LICENCE text file.                                  *
00011 # *                                                                          *
00012 # *   This program is distributed in the hope that it will be useful,        *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of         *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the          *
00015 # *   GNU Library General Public License for more details.                   *
00016 # *                                                                          *
00017 # *   You should have received a copy of the GNU Library General Public      *
00018 # *   License along with this program; if not, write to the Free Software    *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307   *
00020 # *   USA                                                                    *
00021 # *                                                                          *
00022 # ****************************************************************************
00023
00024 __title__ = "BentShapeRebar"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 from RebarDistribution import runRebarDistribution, removeRebarDistribution
00032 from PopUpImage import showPopUpImageDialog
00033 import FreeCAD
00034 import FreeCADGui
00035 import ArchCommands
00036 import os
00037 import sys
00038 import math
00039
00040 def getpointsOfBentShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover,
      bentLength, bentAngle, orientation):
00041     """ getpointsOfBentShapeRebar(FacePRM, LeftCover, RightCover, BottomCover, TopCover, BentLength,
       BentAngle, Orientation):
00042     Return points of the LShape rebar in the form of array for sketch.
00043     It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
00044     """
00045     if orientation == "Bottom":
00046         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00047         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00048         x2 = x1 + bentLength
00049         y2 = y1
00050         dis = (FacePRM[0][1] - t_cover - b_cover) * math.tan(math.radians(bentAngle - 90))
00051         x3 = x2 + dis
00052         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00053         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover - bentLength - dis
00054         y4 = y3
00055         x5 = x4 + dis
00056         y5 = y2
00057         x6 = x5 + bentLength
00058         y6 = y5
00059     elif orientation == "Top":
00060         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00061         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00062         x2 = x1 + bentLength
00063         y2 = y1
00064         dis = (FacePRM[0][1] - t_cover - b_cover) * math.tan(math.radians(bentAngle - 90))
00065         x3 = x2 + dis
00066         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00067         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover - bentLength - dis
00068         y4 = y3
00069         x5 = x4 + dis
00070         y5 = y2
00071         x6 = x5 + bentLength
00072         y6 = y5
00073     elif orientation == "Left":
00074         x1 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00075         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00076         x2 = x1
00077         y2 = y1 - bentLength
00078         dis = (FacePRM[0][0] - r_cover - l_cover) * math.tan(math.radians(bentAngle - 90))
00079         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00080         y3 = y2 - dis
00081         x4 = x3
00082         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover + bentLength + dis
```

```
00083         x5 = x2
00084         y5 = y4 - dis
00085         x6 = x5
00086         y6 = y5 - bentLength
00087     elif orientation == "Right":
00088         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00089         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00090         x2 = x1
00091         y2 = y1 - bentLength
00092         dis = (FacePRM[0][0] - r_cover - l_cover) * math.tan(math.radians(bentAngle - 90))
00093         x3 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00094         y3 = y2 - dis
00095         x4 = x3
00096         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover + bentLength + dis
00097         x5 = x2
00098         y5 = y4 - dis
00099         x6 = x5
00100         y6 = y5 - bentLength
00101     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0),\
00102             FreeCAD.Vector(x3, y3, 0), FreeCAD.Vector(x4, y4, 0),\
00103             FreeCAD.Vector(x5, y5, 0), FreeCAD.Vector(x6, y6, 0)]
00104
00105 class _BentShapeRebarTaskPanel:
00106     def __init__(self, Rebar = None):
00107         if not Rebar:
00108             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00109             self.SelectedObj = selected_obj.Object
00110             self.FaceName = selected_obj.SubElementNames[0]
00111         else:
00112             self.FaceName = Rebar.Base.Support[0][1][0]
00113             self.SelectedObj = Rebar.Base.Support[0][0]
00114         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00115         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "Bent Shape Rebar", None))
00116         self.form.orientation.addItems(["Bottom", "Top", "Right", "Left"])
00117         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00118         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00119         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00120         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00121         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00122         self.form.orientation.currentIndexChanged.connect(self.getOrientation)
00123         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/BentShapeRebar.svg"))
00124         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00125         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/BentShapeRebarDetailed.svg"))
00126         self.Rebar = Rebar
00127
00128     def getOrientation(self):
00129         orientation = self.form.orientation.currentText()
00130         #if orientation == "Bottom":
00131         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
       "/icons/LShapeRebarBR.svg"))
00132         #elif orientation == "Top":
00133         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
       "/icons/LShapeRebarBL.svg"))
00134         #elif orientation == "Right":
00135         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
       "/icons/LShapeRebarTR.svg"))
00136         #else:
00137         #    self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] +
       "/icons/LShapeRebarTL.svg"))
00138
00139     def getStandardButtons(self):
00140         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00141
00142     def clicked(self, button):
00143         if button == int(QtGui.QDialogButtonBox.Apply):
00144             self.accept(button)
00145
00146     def accept(self, signal = None):
00147         f_cover = self.form.frontCover.text()
00148         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00149         b_cover = self.form.bottomCover.text()
00150         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00151         l_cover = self.form.l_sideCover.text()
00152         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00153         r_cover = self.form.r_sideCover.text()
00154         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00155         t_cover = self.form.topCover.text()
00156         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00157         bentLength = self.form.bentLength.text()
00158         bentLength = FreeCAD.Units.Quantity(bentLength).Value
00159         bentAngle = self.form.bentAngle.value()
00160         diameter = self.form.diameter.text()
```

```
00161         diameter = FreeCAD.Units.Quantity(diameter).Value
00162         rounding = self.form.rounding.value()
00163         orientation = self.form.orientation.currentText()
00164         amount_check = self.form.amount_radio.isChecked()
00165         spacing_check = self.form.spacing_radio.isChecked()
00166         if not self.Rebar:
00167             if amount_check:
00168                 amount = self.form.amount.value()
00169                 rebar = makeBentShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, bentLength, bentAngle, rounding, True, amount, orientation, self.
      SelectedObj, self.FaceName)
00170             elif spacing_check:
00171                 spacing = self.form.spacing.text()
00172                 spacing = FreeCAD.Units.Quantity(spacing).Value
00173                 rebar = makeBentShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, bentLength, bentAngle, rounding, False, spacing, orientation, self.
      SelectedObj, self.FaceName)
00174         else:
00175             if amount_check:
00176                 amount = self.form.amount.value()
00177                 rebar = editBentShapeRebar(self.Rebar, f_cover, b_cover, l_cover,
      r_cover, diameter, t_cover, bentLength, bentAngle, rounding, True, amount, orientation, self.
      SelectedObj, self.FaceName)
00178             elif spacing_check:
00179                 spacing = self.form.spacing.text()
00180                 spacing = FreeCAD.Units.Quantity(spacing).Value
00181                 rebar = editBentShapeRebar(self.Rebar, f_cover, b_cover, l_cover,
      r_cover, diameter, t_cover, bentLength, bentAngle, rounding, False, spacing, orientation, self.
      SelectedObj, self.FaceName)
00182         if self.CustomSpacing:
00183             rebar.CustomSpacing = self.CustomSpacing
00184             FreeCAD.ActiveDocument.recompute()
00185         self.Rebar = rebar
00186         if signal == int(QtGui.QDialogButtonBox.Apply):
00187             pass
00188         else:
00189             FreeCADGui.Control.closeDialog(self)
00190
00191     def amount_radio_clicked(self):
00192         self.form.spacing.setEnabled(False)
00193         self.form.amount.setEnabled(True)
00194
00195     def spacing_radio_clicked(self):
00196         self.form.amount.setEnabled(False)
00197         self.form.spacing.setEnabled(True)
00198
00199
00200 def makeBentShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter, t_cover, bentLength,
       bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation = "Bottom Left", structure =
      None, facename = None):
00201     """ makeBentShapeRebar(FrontCover, BottomCover, LeftCover, RightCover, Diameter, TopCover, BentLength,
       BentAngle, Rounding,
00202     AmountSpacingCheck, AmountSpacingValue, Orientation, Structure, Facename): Adds the Bent-Shape
       reinforcement bar to the
00203     selected structural object.
00204     It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
00205     """
00206     if not structure and not facename:
00207         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00208         structure = selected_obj.Object
00209         facename = selected_obj.SubElementNames[0]
00210     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00211     #StructurePRM = getTrueParametersOfStructure(structure)
00212     FacePRM = getParametersOfFace(structure, facename)
00213     if not FacePRM:
00214         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
       derived\n")
00215         return
00216     # Get points of L-Shape rebar
00217     points = getpointsOfBentShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover
      , bentLength, bentAngle, orientation)
00218     import Part
00219     import Arch
00220     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00221     sketch.MapMode = "FlatFace"
00222     sketch.Support = [(structure, facename)]
00223     FreeCAD.ActiveDocument.recompute()
00224     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00225     sketch.addGeometry(Part.LineSegment(points[1], points[2]), False)
00226     sketch.addGeometry(Part.LineSegment(points[2], points[3]), False)
00227     sketch.addGeometry(Part.LineSegment(points[3], points[4]), False)
00228     sketch.addGeometry(Part.LineSegment(points[4], points[5]), False)
00229     import Sketcher
00230     if amount_spacing_check:
00231         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00232         FreeCAD.ActiveDocument.recompute()
00233     else:
```

```
00234         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00235         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
      f_cover)
00236     rebar.Rounding = rounding
00237     # Adds properties to the rebar object
00238     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Shape of rebar")).RebarShape = "BentShapeRebar"
00239     rebar.ViewObject.setEditorMode("RebarShape", 2)
00240     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Front cover of rebar")).FrontCover = f_cover
00241     rebar.setEditorMode("FrontCover", 2)
00242     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Left Side cover of rebar")).LeftCover = l_cover
00243     rebar.setEditorMode("LeftCover", 2)
00244     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Right Side cover of rebar")).RightCover = r_cover
00245     rebar.setEditorMode("RightCover", 2)
00246     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00247     rebar.setEditorMode("BottomCover", 2)
00248     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
       "Amount radio button is checked")).AmountCheck
00249     rebar.setEditorMode("AmountCheck", 2)
00250     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
      , "Top cover of rebar")).TopCover = t_cover
00251     rebar.setEditorMode("TopCover", 2)
00252     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00253     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Shape of rebar")).Orientation = orientation
00254     rebar.setEditorMode("Orientation", 2)
00255     rebar.setEditorMode("TrueSpacing", 2)
00256     rebar.addProperty("App::PropertyDistance", "BentLength", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "BentLength cover of rebar")).BentLength = bentLength
00257     rebar.setEditorMode("BentLength", 2)
00258     rebar.addProperty("App::PropertyDistance", "BentAngle", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Bent Angle of rebar")).BentAngle = bentAngle
00259     rebar.setEditorMode("BentAngle", 2)
00260
00261     if amount_spacing_check:
00262         rebar.AmountCheck = True
00263     else:
00264         rebar.AmountCheck = False
00265         rebar.TrueSpacing = amount_spacing_value
00266     rebar.Label = "BentShapeRebar"
00267     FreeCAD.ActiveDocument.recompute()
00268     return rebar
00269
00270 def editBentShapeRebar(Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover,
      bentLength, bentAngle, rounding, amount_spacing_check, amount_spacing_value, orientation, structure = None,
      facename = None):
00271     sketch = Rebar.Base
00272     if structure and facename:
00273         sketch.Support = [(structure, facename)]
00274     # Check if sketch support is empty.
00275     if not sketch.Support:
00276         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00277         return
00278     # Assigned values
00279     facename = sketch.Support[0][1][0]
00280     structure = sketch.Support[0][0]
00281     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00282     #StructurePRM = getTrueParametersOfStructure(structure)
00283     # Get parameters of the face where sketch of rebar is drawn
00284     FacePRM = getParametersOfFace(structure, facename)
00285     # Get points of L-Shape rebar
00286     points = getpointsOfBentShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover
      , bentLength, bentAngle, orientation)
00287     sketch.movePoint(0, 1, points[0], 0)
00288     FreeCAD.ActiveDocument.recompute()
00289     sketch.movePoint(0, 2, points[1], 0)
00290     FreeCAD.ActiveDocument.recompute()
00291     sketch.movePoint(1, 1, points[1], 0)
00292     FreeCAD.ActiveDocument.recompute()
00293     sketch.movePoint(1, 2, points[2], 0)
00294     FreeCAD.ActiveDocument.recompute()
00295
00296     sketch.movePoint(2, 1, points[2], 0)
00297     FreeCAD.ActiveDocument.recompute()
00298     sketch.movePoint(2, 2, points[3], 0)
00299     FreeCAD.ActiveDocument.recompute()
00300     sketch.movePoint(3, 1, points[3], 0)
00301     FreeCAD.ActiveDocument.recompute()
00302     sketch.movePoint(3, 2, points[4], 0)
00303     FreeCAD.ActiveDocument.recompute()
00304
```

```
00305     sketch.movePoint(4, 1, points[4], 0)
00306     FreeCAD.ActiveDocument.recompute()
00307     sketch.movePoint(4, 2, points[5], 0)
00308     FreeCAD.ActiveDocument.recompute()
00309
00310     Rebar.OffsetStart = f_cover
00311     Rebar.OffsetEnd = f_cover
00312     if amount_spacing_check:
00313         Rebar.Amount = amount_spacing_value
00314         FreeCAD.ActiveDocument.recompute()
00315         Rebar.AmountCheck = True
00316     else:
00317         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00318         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00319         FreeCAD.ActiveDocument.recompute()
00320         Rebar.AmountCheck = False
00321     Rebar.Diameter = diameter
00322     Rebar.FrontCover = f_cover
00323     Rebar.LeftCover = l_cover
00324     Rebar.RightCover = r_cover
00325     Rebar.BottomCover = b_cover
00326     Rebar.TopCover = t_cover
00327     Rebar.BentLength = bentLength
00328     Rebar.BentAngle = bentAngle
00329     Rebar.Rounding = rounding
00330     Rebar.TrueSpacing = amount_spacing_value
00331     Rebar.Orientation = orientation
00332     FreeCAD.ActiveDocument.recompute()
00333     return Rebar
00334
00335 def editDialog(vobj):
00336     FreeCADGui.Control.closeDialog()
00337     obj = _BentShapeRebarTaskPanel(vobj.Object)
00338     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00339     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00340     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00341     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00342     obj.form.diameter.setText(str(vobj.Object.Diameter))
00343     obj.form.topCover.setText(str(vobj.Object.TopCover))
00344     obj.form.bentLength.setText(str(vobj.Object.BentLength))
00345     obj.form.bentAngle.setValue(vobj.Object.BentAngle)
00346     obj.form.rounding.setValue(vobj.Object.Rounding)
00347     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00348     if vobj.Object.AmountCheck:
00349         obj.form.amount.setValue(vobj.Object.Amount)
00350     else:
00351         obj.form.amount_radio.setChecked(False)
00352         obj.form.spacing_radio.setChecked(True)
00353         obj.form.amount.setDisabled(True)
00354         obj.form.spacing.setEnabled(True)
00355         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00356     #obj.form.PickSelectedFace.setVisible(False)
00357     FreeCADGui.Control.showDialog(obj)
00358
00359 def CommandBentShapeRebar():
00360     selected_obj = check_selected_face()
00361     if selected_obj:
00362         FreeCADGui.Control.showDialog(_BentShapeRebarTaskPanel())
```

8.3 HelicalRebar.py File Reference

Classes

- class HelicalRebar._HelicalRebarTaskPanel

Namespaces

- HelicalRebar

Functions

- def HelicalRebar.getpointsOfHelicalRebar (FacePRM, s_cover, b_cover, t_cover, pitch, edges, diameter, size, direction)
- def HelicalRebar.createHelicalWire (FacePRM, s_cover, b_cover, t_cover, pitch, size, direction, helix=None)
- def HelicalRebar.makeHelicalRebar (s_cover, b_cover, diameter, t_cover, pitch, structure=None, facename=None)
- def HelicalRebar.editHelicalRebar (Rebar, s_cover, b_cover, diameter, t_cover, pitch, structure=None, facename=None)
- def HelicalRebar.editDialog (vobj)
- def HelicalRebar.CommandHelicalRebar ()

Variables

- string HelicalRebar.__title__ = "HelicalRebar"
- string HelicalRebar.__author__ = "Amritpal Singh"
- string HelicalRebar.__url__ = "https://www.freecadweb.org"

8.4 HelicalRebar.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
00023
00024 __title__ = "HelicalRebar"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 from PopUpImage import showPopUpImageDialog
00032 import FreeCAD
00033 import FreeCADGui
00034 import ArchCommands
00035 import os
00036 import sys
00037 import math
00038
00039 def getpointsOfHelicalRebar(FacePRM, s_cover, b_cover, t_cover, pitch, edges,
      diameter, size, direction):
00040     """ getpointsOfHelicalRebar(FacePRM, s_cover, b_cover, t_cover):
00041     Return points of the LShape rebar in the form of array for sketch."""
00042     dx = s_cover + diameter / 2
00043     dz = float(pitch) / edges
00044     R = diameter / 2 - dx
00045     R = FacePRM[0][0] / 2 - s_cover
00046     points = []
00047     if direction[2] in {-1,1}:
00048         z = 0
00049         l = 0
00050         if direction[2] == 1:
```

```
00051             zz = FacePRM[1][2] - t_cover
00052         elif direction[2] == -1:
00053             zz = FacePRM[1][2] + b_cover
00054         count = 0
00055         flag = False
00056         while (round(z) < abs(size - b_cover - t_cover)):
00057             for i in range(0, int(edges) + 1):
00058                 if not i and flag:
00059                     continue
00060                 if not flag:
00061                     z -= dz
00062                     flag = True
00063                 iAngle = i * 360 / edges
00064                 x =  FacePRM[1][0] + R * math.cos(math.radians(iAngle))
00065                 y =  FacePRM[1][1] + R * math.sin(math.radians(iAngle))
00066                 points.append(FreeCAD.Vector(x, y, zz))
00067                 count += 1
00068                 if direction[2] == 1:
00069                     zz -= dz
00070                 elif direction[2] == -1:
00071                     zz += dz
00072                 z += dz
00073     return points
00074
00075 def createHelicalWire(FacePRM, s_cover, b_cover, t_cover, pitch, size, direction, helix =
    None):
00076     """ createHelicalWire(FacePRM, SideCover, BottomCover, TopCover, Pitch, Size, Direction, Helix = None):
00077     It creates a helical wire."""
00078     import Part
00079     if not helix:
00080         helix = FreeCAD.ActiveDocument.addObject("Part::Helix","Helix")
00081     helix.Pitch = pitch
00082     helix.Radius = FacePRM[0][0] / 2 - s_cover
00083     helix.Angle = 0
00084     helix.LocalCoord = 0
00085     helix.Height = size - b_cover - t_cover
00086     if round(direction.x) == 1:
00087         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0] - b_cover, FacePRM[1][1], FacePRM[1][2])
00088         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, -1, 0), 90)
00089     elif round(direction.x) == -1:
00090         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0] + t_cover, FacePRM[1][1], FacePRM[1][2])
00091         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, -1, 0), -90)
00092     elif round(direction.y) == 1:
00093         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1] - b_cover, FacePRM[1][2])
00094         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(1, 0, 0), 90)
00095     elif round(direction.y) == -1:
00096         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1] + t_cover, FacePRM[1][2])
00097         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(-1, 0, 0), 90)
00098     elif round(direction.z) == 1:
00099         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1], FacePRM[1][2] - size + b_cover)
00100         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, 0, 1), 0)
00101     elif round(direction.z) == -1:
00102         helix.Placement.Base = FreeCAD.Vector(FacePRM[1][0], FacePRM[1][1], FacePRM[1][2] + b_cover)
00103         helix.Placement.Rotation = FreeCAD.Rotation(FreeCAD.Vector(0, 0, -1), 0)
00104     FreeCAD.ActiveDocument.recompute()
00105     return helix
00106
00107 class _HelicalRebarTaskPanel:
00108     def __init__(self, Rebar = None):
00109         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00110         self.form.setWindowTitle(QtGui.QApplication.translate("Arch", "Helical Rebar", None))
00111         if not Rebar:
00112             normal = facenormalDirection()
00113         else:
00114             normal = facenormalDirection(Rebar.Base.Support[0][0], Rebar.Base.Support[0]
    [1][0])
00115         if not round(normal.z) in {1, -1}:
00116             self.form.topCoverLabel.setText(translate("RebarAddon", "Left Cover"))
00117             self.form.bottomCoverLabel.setText(translate("RebarAddon", "Right Cover"))
00118         self.form.PickSelectedFace.clicked.connect(self.getSelectedFace)
00119         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
    /icons/HelicalRebar.svg"))
00120         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
    (os.path.abspath(__file__))[0] + "/icons/HelicalRebarDetailed.svg"))
00121         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
    QtGui.QStyle.SP_DialogHelpButton))
00122         self.Rebar = Rebar
00123         self.SelectedObj = None
00124         self.FaceName = None
00125
00126     def getStandardButtons(self):
00127         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
    QtGui.QDialogButtonBox.Cancel)
00128
00129     def clicked(self, button):
00130         if button == int(QtGui.QDialogButtonBox.Apply):
00131             self.accept(button)
```

```
00132
00133     def getSelectedFace(self):
00134         getSelectedFace(self)
00135         normal = facenormalDirection()
00136         if not round(normal.z) in {1, -1}:
00137             self.form.topCoverLabel.setText(translate("RebarAddon", "Left Cover"))
00138             self.form.bottomCoverLabel.setText(translate("RebarAddon", "Right Cover"))
00139         else:
00140             self.form.topCoverLabel.setText(translate("RebarAddon", "Top Cover"))
00141             self.form.bottomCoverLabel.setText(translate("RebarAddon", "Bottom Cover"))
00142
00143
00144     def accept(self, signal = None):
00145         b_cover = self.form.bottomCover.text()
00146         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00147         s_cover = self.form.sideCover.text()
00148         s_cover = FreeCAD.Units.Quantity(s_cover).Value
00149         t_cover = self.form.topCover.text()
00150         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00151         pitch = self.form.pitch.text()
00152         pitch = FreeCAD.Units.Quantity(pitch).Value
00153         diameter = self.form.diameter.text()
00154         diameter = FreeCAD.Units.Quantity(diameter).Value
00155         if not self.Rebar:
00156             rebar = makeHelicalRebar(s_cover, b_cover, diameter, t_cover, pitch, self.
    SelectedObj, self.FaceName)
00157         else:
00158             rebar = editHelicalRebar(self.Rebar, s_cover, b_cover, diameter, t_cover,
    pitch, self.SelectedObj, self.FaceName)
00159         self.Rebar = rebar
00160         if signal == int(QtGui.QDialogButtonBox.Apply):
00161             pass
00162         else:
00163             FreeCADGui.Control.closeDialog(self)
00164
00165 def makeHelicalRebar(s_cover, b_cover, diameter, t_cover, pitch, structure = None, facename
     = None):
00166     """ makeHelicalRebar(SideCover, BottomCover, Diameter, TopCover, Pitch, Structure, Facename):
00167     Adds the Helical reinforcement bar to the selected structural object."""
00168     if not structure and not facename:
00169         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00170         structure = selected_obj.Object
00171         facename = selected_obj.SubElementNames[0]
00172     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00173     #StructurePRM = getTrueParametersOfStructure(structure)
00174     FacePRM = getParametersOfFace(structure, facename, False)
00175     if not FacePRM:
00176         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
     derived\n")
00177         return
00178     size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00179     normal = face.normalAt(0,0)
00180     #normal = face.Placement.Rotation.inverted().multVec(normal)
00181     import Arch
00182     helix = createHelicalWire(FacePRM, s_cover, b_cover, t_cover, pitch, size, normal)
00183     helix.Support = [(structure, facename)]
00184     rebar = Arch.makeRebar(structure, helix, diameter, 1, 0)
00185     rebar.OffsetStart = 0
00186     rebar.OffsetEnd = 0
00187     FreeCAD.ActiveDocument.recompute()
00188     # Adds properties to the rebar object
00189     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Shape of rebar")).RebarShape = "HelicalRebar"
00190     rebar.ViewObject.setEditorMode("RebarShape", 2)
00191     rebar.addProperty("App::PropertyDistance", "SideCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
    ", "Front cover of rebar")).SideCover = s_cover
00192     rebar.setEditorMode("SideCover", 2)
00193     rebar.addProperty("App::PropertyDistance", "Pitch", "RebarDialog", QT_TRANSLATE_NOOP("App::Property", "
    Left Side cover of rebar")).Pitch = pitch
00194     rebar.setEditorMode("Pitch", 2)
00195     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00196     rebar.setEditorMode("BottomCover", 2)
00197     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
    , "Top cover of rebar")).TopCover = t_cover
00198     rebar.setEditorMode("TopCover", 2)
00199     rebar.Label = "HelicalRebar"
00200     FreeCAD.ActiveDocument.recompute()
00201     return rebar
00202
00203 def editHelicalRebar(Rebar, s_cover, b_cover, diameter, t_cover, pitch, structure = None,
    facename = None):
00204     sketch = Rebar.Base
00205     if structure and facename:
00206         sketch.Support = [(structure, facename)]
00207     # Check if sketch support is empty.
00208     if not sketch.Support:
```

```
00209         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
      unchecked Edit->Preferences->Arch.")
00210         return
00211     # Assigned values
00212     facename = sketch.Support[0][1][0]
00213     structure = sketch.Support[0][0]
00214     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00215     #StructurePRM = getTrueParametersOfStructure(structure)
00216     # Get parameters of the face where sketch of rebar is drawn
00217     FacePRM = getParametersOfFace(structure, facename, False)
00218     size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00219     normal = face.normalAt(0,0)
00220     #normal = face.Placement.Rotation.inverted().multVec(normal)
00221     helix = createHelicalWire(FacePRM, s_cover, b_cover, t_cover, pitch, size, normal,
     Rebar.Base)
00222     FreeCAD.ActiveDocument.recompute()
00223     Rebar.Diameter = diameter
00224     Rebar.SideCover = s_cover
00225     Rebar.BottomCover = b_cover
00226     Rebar.TopCover = t_cover
00227     Rebar.Pitch = pitch
00228     FreeCAD.ActiveDocument.recompute()
00229     return Rebar
00230
00231 def editDialog(vobj):
00232     FreeCADGui.Control.closeDialog()
00233     obj = _HelicalRebarTaskPanel(vobj.Object)
00234     obj.form.sideCover.setText(str(vobj.Object.SideCover))
00235     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00236     obj.form.diameter.setText(str(vobj.Object.Diameter))
00237     obj.form.topCover.setText(str(vobj.Object.TopCover))
00238     obj.form.pitch.setText(str(vobj.Object.Pitch))
00239     FreeCADGui.Control.showDialog(obj)
00240
00241 def CommandHelicalRebar():
00242     selected_obj = check_selected_face()
00243     if selected_obj:
00244         FreeCADGui.Control.showDialog(_HelicalRebarTaskPanel())
```

8.5 LShapeRebar.py File Reference

Classes

- class LShapeRebar._LShapeRebarTaskPanel

Namespaces

- LShapeRebar

Functions

- def LShapeRebar.getpointsOfLShapeRebar (FacePRM, l_cover, r_cover, b_cover, t_cover, orientation)
- def LShapeRebar.makeLShapeRebar (f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation="Bottom Left", structure=None, facename=None)
- def LShapeRebar.editLShapeRebar (Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def LShapeRebar.editDialog (vobj)
- def LShapeRebar.CommandLShapeRebar ()

Variables

- string LShapeRebar.__title__ = "LShapeRebar"
- string LShapeRebar.__author__ = "Amritpal Singh"
- string LShapeRebar.__url__ = "https://www.freecadweb.org"

8.6 LShapeRebar.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
00023
00024 __title__ = "LShapeRebar"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 from RebarDistribution import runRebarDistribution, removeRebarDistribution
00032 from PopUpImage import showPopUpImageDialog
00033 import FreeCAD
00034 import FreeCADGui
00035 import ArchCommands
00036 import os
00037 import sys
00038 import math
00039
00040 def getpointsOfLShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover, orientation):
00041     """ getpointsOfLShapeRebar(FacePRM, LeftCover, RightCover, BottomCover, TopCover, Orientation):
00042     Return points of the LShape rebar in the form of array for sketch.
00043     It takes four different orientations input i.e. 'Bottom Left', 'Bottom Right ', 'Top Left', 'Top Right'
    .
00044     """
00045     if orientation == "Bottom Left":
00046         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00047         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00048         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00049         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00050         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00051         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00052     elif orientation == "Bottom Right":
00053         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00054         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00055         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00056         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00057         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00058         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00059     elif orientation == "Top Left":
00060         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00061         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00062         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00063         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00064         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00065         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00066     elif orientation == "Top Right":
00067         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00068         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00069         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00070         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00071         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00072         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00073     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0),\
00074             FreeCAD.Vector(x3, y3, 0)]
00075
00076 class _LShapeRebarTaskPanel:
00077     def __init__(self, Rebar = None):
00078         self.CustomSpacing = None
00079         if not Rebar:
00080             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00081             self.SelectedObj = selected_obj.Object
00082             self.FaceName = selected_obj.SubElementNames[0]
00083         else:
```

```
00084             self.FaceName = Rebar.Base.Support[0][1][0]
00085             self.SelectedObj = Rebar.Base.Support[0][0]
00086         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00087         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "L-Shape Rebar", None))
00088         self.form.orientation.addItems(["Bottom Right", "Bottom Left", "Top Right", "Top Left"])
00089         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00090         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00091         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00092         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00093         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00094         self.form.orientation.currentIndexChanged.connect(self.getOrientation)
00095         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarBR.svg"))
00096         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00097         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/LShapeRebarDetailed.svg"))
00098         self.Rebar = Rebar
00099
00100     def getOrientation(self):
00101         orientation = self.form.orientation.currentText()
00102         if orientation == "Bottom Right":
00103             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarBR.svg"))
00104         elif orientation == "Bottom Left":
00105             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarBL.svg"))
00106         elif orientation == "Top Right":
00107             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarTR.svg"))
00108         else:
00109             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/LShapeRebarTL.svg"))
00110
00111     def getStandardButtons(self):
00112         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00113
00114     def clicked(self, button):
00115         if button == int(QtGui.QDialogButtonBox.Apply):
00116             self.accept(button)
00117
00118     def accept(self, signal = None):
00119         f_cover = self.form.frontCover.text()
00120         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00121         b_cover = self.form.bottomCover.text()
00122         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00123         l_cover = self.form.l_sideCover.text()
00124         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00125         r_cover = self.form.r_sideCover.text()
00126         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00127         t_cover = self.form.topCover.text()
00128         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00129         diameter = self.form.diameter.text()
00130         diameter = FreeCAD.Units.Quantity(diameter).Value
00131         rounding = self.form.rounding.value()
00132         orientation = self.form.orientation.currentText()
00133         amount_check = self.form.amount_radio.isChecked()
00134         spacing_check = self.form.spacing_radio.isChecked()
00135         if not self.Rebar:
00136             if amount_check:
00137                 amount = self.form.amount.value()
00138                 rebar = makeLShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00139             elif spacing_check:
00140                 spacing = self.form.spacing.text()
00141                 spacing = FreeCAD.Units.Quantity(spacing).Value
00142                 rebar = makeLShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter,
      t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00143         else:
00144             if amount_check:
00145                 amount = self.form.amount.value()
00146                 rebar = editLShapeRebar(self.Rebar, f_cover, b_cover, l_cover, r_cover,
       diameter, t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00147             elif spacing_check:
00148                 spacing = self.form.spacing.text()
00149                 spacing = FreeCAD.Units.Quantity(spacing).Value
00150                 rebar = editLShapeRebar(self.Rebar, f_cover, b_cover, l_cover, r_cover,
       diameter, t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00151         if self.CustomSpacing:
00152             rebar.CustomSpacing = self.CustomSpacing
00153             FreeCAD.ActiveDocument.recompute()
```

```
00154         self.Rebar = rebar
00155         if signal == int(QtGui.QDialogButtonBox.Apply):
00156             pass
00157         else:
00158             FreeCADGui.Control.closeDialog(self)
00159 
00160     def amount_radio_clicked(self):
00161         self.form.spacing.setEnabled(False)
00162         self.form.amount.setEnabled(True)
00163 
00164     def spacing_radio_clicked(self):
00165         self.form.amount.setEnabled(False)
00166         self.form.spacing.setEnabled(True)
00167 
00168 
00169 def makeLShapeRebar(f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding,
    amount_spacing_check, amount_spacing_value, orientation = "Bottom Left", structure = None, facename = None):
00170     """ makeLShapeRebar(FrontCover, BottomCover, LeftCover, RightCover, Diameter, TopCover, Rounding,
     AmountSpacingCheck, AmountSpacingValue,
00171     Orientation, Structure, Facename): Adds the L-Shape reinforcement bar to the selected structural
     object.
00172     It takes four different orientations input i.e. 'Bottom Left', 'Bottom Right ', 'Top Left', 'Top Right'
    .
00173     """
00174     if not structure and not facename:
00175         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00176         structure = selected_obj.Object
00177         facename = selected_obj.SubElementNames[0]
00178     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00179     #StructurePRM = getTrueParametersOfStructure(structure)
00180     FacePRM = getParametersOfFace(structure, facename)
00181     if not FacePRM:
00182         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
     derived\n")
00183         return
00184     # Get points of L-Shape rebar
00185     points = getpointsOfLShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover,
    orientation)
00186     import Part
00187     import Arch
00188     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00189     sketch.MapMode = "FlatFace"
00190     sketch.Support = [(structure, facename)]
00191     FreeCAD.ActiveDocument.recompute()
00192     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00193     sketch.addGeometry(Part.LineSegment(points[1], points[2]), False)
00194     import Sketcher
00195     if amount_spacing_check:
00196         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00197         FreeCAD.ActiveDocument.recompute()
00198     else:
00199         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00200         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
    f_cover)
00201     rebar.Rounding = rounding
00202     # Adds properties to the rebar object
00203     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Shape of rebar")).RebarShape = "LShapeRebar"
00204     rebar.ViewObject.setEditorMode("RebarShape", 2)
00205     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Front cover of rebar")).FrontCover = f_cover
00206     rebar.setEditorMode("FrontCover", 2)
00207     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
    ", "Left Side cover of rebar")).LeftCover = l_cover
00208     rebar.setEditorMode("LeftCover", 2)
00209     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Right Side cover of rebar")).RightCover = r_cover
00210     rebar.setEditorMode("RightCover", 2)
00211     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00212     rebar.setEditorMode("BottomCover", 2)
00213     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
     "Amount radio button is checked")).AmountCheck
00214     rebar.setEditorMode("AmountCheck", 2)
00215     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
    , "Top cover of rebar")).TopCover = t_cover
00216     rebar.setEditorMode("TopCover", 2)
00217     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
    App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00218     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
    ", "Shape of rebar")).Orientation = orientation
00219     rebar.setEditorMode("Orientation", 2)
00220     rebar.setEditorMode("TrueSpacing", 2)
00221     if amount_spacing_check:
00222         rebar.AmountCheck = True
00223     else:
00224         rebar.AmountCheck = False
```

```
00225         rebar.TrueSpacing = amount_spacing_value
00226     rebar.Label = "LShapeRebar"
00227     FreeCAD.ActiveDocument.recompute()
00228     return rebar
00229
00230 def editLShapeRebar(Rebar, f_cover, b_cover, l_cover, r_cover, diameter, t_cover, rounding,
      amount_spacing_check, amount_spacing_value, orientation, structure = None, facename = None):
00231     sketch = Rebar.Base
00232     if structure and facename:
00233         sketch.Support = [(structure, facename)]
00234     # Check if sketch support is empty.
00235     if not sketch.Support:
00236         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00237         return
00238     # Assigned values
00239     facename = sketch.Support[0][1][0]
00240     structure = sketch.Support[0][0]
00241     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00242     #StructurePRM = getTrueParametersOfStructure(structure)
00243     # Get parameters of the face where sketch of rebar is drawn
00244     FacePRM = getParametersOfFace(structure, facename)
00245     # Get points of L-Shape rebar
00246     points = getpointsOfLShapeRebar(FacePRM, l_cover, r_cover, b_cover, t_cover,
      orientation)
00247     sketch.movePoint(0, 1, points[0], 0)
00248     FreeCAD.ActiveDocument.recompute()
00249     sketch.movePoint(0, 2, points[1], 0)
00250     FreeCAD.ActiveDocument.recompute()
00251     sketch.movePoint(1, 1, points[1], 0)
00252     FreeCAD.ActiveDocument.recompute()
00253     sketch.movePoint(1, 2, points[2], 0)
00254     FreeCAD.ActiveDocument.recompute()
00255     Rebar.OffsetStart = f_cover
00256     Rebar.OffsetEnd = f_cover
00257     if amount_spacing_check:
00258         Rebar.Amount = amount_spacing_value
00259         FreeCAD.ActiveDocument.recompute()
00260         Rebar.AmountCheck = True
00261     else:
00262         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00263         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00264         FreeCAD.ActiveDocument.recompute()
00265         Rebar.AmountCheck = False
00266     Rebar.Diameter = diameter
00267     Rebar.FrontCover = f_cover
00268     Rebar.LeftCover = l_cover
00269     Rebar.RightCover = r_cover
00270     Rebar.BottomCover = b_cover
00271     Rebar.TopCover = t_cover
00272     Rebar.Rounding = rounding
00273     Rebar.TrueSpacing = amount_spacing_value
00274     Rebar.Orientation = orientation
00275     FreeCAD.ActiveDocument.recompute()
00276     return Rebar
00277
00278 def editDialog(vobj):
00279     FreeCADGui.Control.closeDialog()
00280     obj = _LShapeRebarTaskPanel(vobj.Object)
00281     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00282     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00283     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00284     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00285     obj.form.diameter.setText(str(vobj.Object.Diameter))
00286     obj.form.topCover.setText(str(vobj.Object.TopCover))
00287     obj.form.rounding.setValue(vobj.Object.Rounding)
00288     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00289     if vobj.Object.AmountCheck:
00290         obj.form.amount.setValue(vobj.Object.Amount)
00291     else:
00292         obj.form.amount_radio.setChecked(False)
00293         obj.form.spacing_radio.setChecked(True)
00294         obj.form.amount.setDisabled(True)
00295         obj.form.spacing.setEnabled(True)
00296         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00297     #obj.form.PickSelectedFace.setVisible(False)
00298     FreeCADGui.Control.showDialog(obj)
00299
00300 def CommandLShapeRebar():
00301     selected_obj = check_selected_face()
00302     if selected_obj:
00303         FreeCADGui.Control.showDialog(_LShapeRebarTaskPanel())
```

8.7 PopUpImage.py File Reference

Classes

- class PopUpImage.PopUpImage

Namespaces

- PopUpImage

Functions

- def PopUpImage.showPopUpImageDialog (img)

Variables

- string PopUpImage.__title__ = "PopUpImage"
- string PopUpImage.__author__ = "Amritpal Singh"
- string PopUpImage.__url__ = "https://www.freecadweb.org"

8.8 PopUpImage.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
00023
00024 __title__ = "PopUpImage"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028
00029 from PySide import QtCore
00030 from PySide import QtGui
00031 from PySide import QtSvg
00032 import FreeCADGui
00033 import os
00034
00035 class PopUpImage(QtGui.QDialog):
00036     def __init__(self, img):
00037         QtGui.QDialog.__init__(self)
00038         self.image = QtSvg.QSvgWidget(img)
00039         self.setWindowTitle(QtGui.QApplication.translate("RebarTool", "Detailed description", None))
00040         self.verticalLayout = QtGui.QVBoxLayout(self)
00041         self.verticalLayout.addWidget(self.image)
00042
00043 def showPopUpImageDialog(img):
00044     """ showPopUpImageDialog(image): This function will show a given image in a pop-up
00045     dialog box."""
00046     dialog = PopUpImage(img)
00047     dialog.exec_()
```

8.9 README.md File Reference

8.10 README.md

```
00001 # Rebar Addon for FreeCAD
00002
00003 Started as a Google Summer of Code ([GSoC](https://en.wikipedia.org/wiki/Google_Summer_of_Code) 2017)
       [project](https://summerofcode.withgoogle.com/archive/2017/projects/6536382147198976).
00004
00005 ![screenshot](http://i.imgur.com/r9b517K.jpg)
00006
00007 ## Documentation
00008 This project is aimed at easing up the process of rebaring in [FreeCAD](https://www.freecadweb.org).
       In this project, list of rebars will be provided to user under Rebar tools in the form of dropdown. This
       project covers six different rebar shapes as given below:
00009
00010 -
       ![icon](https://www.freecadweb.org/wiki/images/thumb/6/69/Arch_Rebar_Straight.png/32px-Arch_Rebar_Straight.png) **Stra
00011 ![screenshot](https://www.freecadweb.org/wiki/images/f/fd/StraightRebar.png)
00012
00013 -
       ![icon](https://www.freecadweb.org/wiki/images/thumb/4/4d/Arch_Rebar_UShape.png/32px-Arch_Rebar_UShape.png) **UShape Re
00014 ![screenshot](https://www.freecadweb.org/wiki/images/3/35/Footing_UShapeRebar.png)
00015
00016 -
       ![icon](https://www.freecadweb.org/wiki/images/thumb/3/38/Arch_Rebar_LShape.png/32px-Arch_Rebar_LShape.png) **LShape Re
00017 ![screenshot](https://www.freecadweb.org/wiki/images/1/10/LShapeRebarNew.png)
00018
00019 -
       ![icon](https://www.freecadweb.org/wiki/images/thumb/0/0b/Arch_Rebar_BentShape.png/32px-Arch_Rebar_BentShape.png) **Ben
00020 ![screenshot](https://www.freecadweb.org/wiki/images/e/e3/BentShapeRebar.png)
00021
00022 -
       ![icon](https://www.freecadweb.org/wiki/images/thumb/e/ef/Arch_Rebar_Stirrup.png/32px-Arch_Rebar_Stirrup.png) **Stirrup
00023 ![screenshot](https://www.freecadweb.org/wiki/images/9/9b/Stirrup.png)
00024
00025 -
       ![icon](https://www.freecadweb.org/wiki/images/thumb/c/c9/Arch_Rebar_Helical.png/32px-Arch_Rebar_Helical.png) **Helical
00026 ![screenshot](https://www.freecadweb.org/wiki/images/2/2f/HelicalRebar.png)
00027
00028 ## Video Tutorial
00029 [![IMAGE ALT TEXT
       HERE](http://i.imgur.com/ZQGCQoe.png)](https://www.youtube.com/watch?v=BYQQjEKmx5E&t=1435s)
00030
00031
00032 ## Installation
00033
00034 ### Pre-requisites
00035 - FreeCAD (version >= 0.17): [Installation guide](https://www.freecadweb.org/wiki/Installing)
00036
00037 ### Steps to install Rebar Addon in FreeCAD
00038 1. Open the FreeCAD Addon Manager (```Tool -> Addon manager```).
00039 2. When an addon manager will open, select ```Reinforcement``` from a list of workbenches shown by an
       addon manager.
00040 3. After selecting, click on ```Install/Update``` button.
00041 4. Restart FreeCAD.
00042 5. Now you will see different rebars in a drop-down list of rebar tools (```Arch -> Rebar tools ->
       Different rebars```).
00043
00044 ## How it works
00045 Each rebar tool has two files, one is ```Python``` file and second is there respective name ```UI```
       file like ```StraightRebar.py``` and ```StraightRebar.ui``` file). Let's take a straight rebar tool. In
       ```StraightRebar.py``` file, there are two functions. One is ```makeStraightRebar()``` function. This function
       creates straight rebar and adds new properties to the default ```Rebar``` object. Second function is
       ```editStraightRebar```. This function is used when we want to change a new properties(which is created by
       ```makeStraightRebar``` function) of the rebar object and it will take ```Rebar``` object as input which is created by
       ```makeStraightRebar``` function. In ```StraightRebar.py```, ```_StraightRebarTaskPanel``` class is
       present. This class loads UI(present in ```StriaghtRebar.ui``` file) in the task panel of FreeCAD. First time when
       a user clicks on ```Apply``` or ```Ok``` button, then ```makeStraightRebar``` function is executed and after
       that when user want to change the properties of Straight rebar then ```editStraightRebar``` function is
       excuted.
00046
00047 ## Extras
00048 - [FreeCAD forum thread](https://forum.freecadweb.org/viewtopic.php?f=8&t=22760)
00049 - [GSoC proposal](https://brlcad.org/wiki/User:Amritpal_singh/gsoc_proposal)
00050 - [Development logs](https://brlcad.org/wiki/User:Amritpal_singh/GSoC17/logs)
```

8.11 RebarDistribution.py File Reference

Classes

- class RebarDistribution._RebarDistributionDialog

Namespaces

- RebarDistribution

Functions

- def RebarDistribution.getCustomSpacingString (amount1, spacing1, amount2, spacing2, amount3, spacing3, frontCover, size)
- def RebarDistribution.getupleOfCustomSpacing (span_string)
- def RebarDistribution.runRebarDistribution (self)
- def RebarDistribution.removeRebarDistribution (self)

Variables

- string RebarDistribution.__title__ = "DialogDistribution"
- string RebarDistribution.__author__ = "Amritpal Singh"
- string RebarDistribution.__url__ = "https://www.freecadweb.org"
- RebarDistribution.CustomSpacing

8.12 RebarDistribution.py

```
00001 # -*- coding: utf-8 -*-
00002 # ****************************************************************************
00003 # *                                                                          *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>              *
00005 # *                                                                          *
00006 # *   This program is free software; you can redistribute it and/or modify   *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)     *
00008 # *   as published by the Free Software Foundation; either version 2 of      *
00009 # *   the License, or (at your option) any later version.                    *
00010 # *   for detail see the LICENCE text file.                                  *
00011 # *                                                                          *
00012 # *   This program is distributed in the hope that it will be useful,        *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of         *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the          *
00015 # *   GNU Library General Public License for more details.                   *
00016 # *                                                                          *
00017 # *   You should have received a copy of the GNU Library General Public      *
00018 # *   License along with this program; if not, write to the Free Software    *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307   *
00020 # *   USA                                                                    *
00021 # *                                                                          *
00022 # ****************************************************************************
00023
00024 __title__ = "DialogDistribution"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 import FreeCAD
00032 import FreeCADGui
00033 import ArchCommands
00034 import os
00035 import sys
00036 import math
00037
00038 class _RebarDistributionDialog():
00039     def __init__(self, frontCover, size):
00040         self.FrontCover = frontCover
```

```
00041         self.ExpandingLength = size
00042         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00043         self.form.setWindowTitle(QtGui.QApplication.translate("Arch", "Rebar Distribution", None))
00044         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
    /icons/RebarDistribution.svg"))
00045
00046     def accept(self):
00047         amount1 = self.form.amount1.value()
00048         spacing1 = self.form.spacing1.text()
00049         spacing1 = FreeCAD.Units.Quantity(spacing1).Value
00050         amount2 = self.form.amount2.value()
00051         spacing2 = self.form.spacing2.text()
00052         spacing2 = FreeCAD.Units.Quantity(spacing2).Value
00053         amount3 = self.form.amount3.value()
00054         spacing3 = self.form.spacing3.text()
00055         spacing3 = FreeCAD.Units.Quantity(spacing3).Value
00056         self.CustomSpacing = getCustomSpacingString(amount1, spacing1,
    amount2, spacing2, amount3, spacing3, self.FrontCover, self.
    ExpandingLength)
00057
00058     def setupUi(self):
00059         # Connect Signals and Slots
00060         self.form.buttonBox.accepted.connect(self.accept)
00061         pass
00062
00063 def getCustomSpacingString(amount1, spacing1, amount2, spacing2, amount3, spacing3,
    frontCover, size):
00064     seg1_area = amount1 * spacing1 - spacing1 / 2
00065     seg3_area = amount3 * spacing3 - spacing3 / 2
00066     seg2_area = size - seg1_area - seg3_area - 2 * frontCover
00067     if seg2_area < 0:
00068         FreeCAD.Console.PrintError("Sum of length of segment 1 and segment 2 is greater than length of
     rebar expands.\n")
00069         return
00070     if spacing1 and spacing2 and spacing3 and amount1 and amount2 and amount3:
00071         pass
00072     else:
00073         if spacing1 and spacing2 and spacing3:
00074             amount2 = math.ceil(seg2_area / spacing2)
00075             spacing2 = seg2_area / amount2
00076         elif amount1 and amount2 and amount3:
00077             spacing2 = math.floor(seg2_area / amount2)
00078     CustomSpacing = str(amount1) + "@" + str(spacing1) + "+" + str(int(amount2)) + "@" + str(spacing2) + "+
    " + str(amount3) + "@" + str(spacing3)
00079     return CustomSpacing
00080
00081 def getupleOfCustomSpacing(span_string):
00082     """ gettupleOfCustomSpacing(span_string): This function take input
00083     in specific syntax and return output in the form of list. For eg.
00084     Input: "3@100+2@200+3@100"
00085     Output: [(3, 100), (2, 200), (3, 100)]"""
00086     import string
00087     span_st = string.strip(span_string)
00088     span_sp = string.split(span_st, '+')
00089     index = 0
00090     spacinglist = []
00091     while index < len(span_sp):
00092         # Find "@" recursively in span_sp array.
00093         in_sp = string.split(span_sp[index], '@')
00094         spacinglist.append((int(in_sp[0]),float(in_sp[1])))
00095         index += 1
00096     return spacinglist
00097
00098 def runRebarDistribution(self):
00099     frontCover = self.form.frontCover.text()
00100     frontCover = FreeCAD.Units.Quantity(frontCover).Value
00101     face = self.SelectedObj.Shape.Faces[getFaceNumber(self.FaceName) - 1]
00102     size = (ArchCommands.projectToVector(self.SelectedObj.Shape.copy(), face.normalAt(0, 0))).Length
00103     dialog = _RebarDistributionDialog(frontCover, size)
00104     dialog.setupUi()
00105     dialog.form.exec_()
00106     self.CustomSpacing = dialog.CustomSpacing
00107
00108 def removeRebarDistribution(self):
00109     self.CustomSpacing = ""
00110     self.Rebar.CustomSpacing = ""
00111     FreeCAD.ActiveDocument.recompute()
00112
00113 #runRebarDistribution(App.ActiveDocument.Rebar)
```

8.13 Rebarfunc.py File Reference

Namespaces

- Rebarfunc

Functions

- def Rebarfunc.getEdgesAngle (edge1, edge2)
- def Rebarfunc.checkRectangle (edges)
- def Rebarfunc.getBaseStructuralObject (obj)
- def Rebarfunc.getBaseObject (obj)
- def Rebarfunc.getFaceNumber (s)
- def Rebarfunc.facenormalDirection (structure=None, facename=None)
- def Rebarfunc.getTrueParametersOfStructure (obj)
- def Rebarfunc.getParametersOfFace (structure, facename, sketch=True)
- def Rebarfunc.extendedTangentPartLength (rounding, diameter, angle)
- def Rebarfunc.extendedTangentLength (rounding, diameter, angle)
- def Rebarfunc.check_selected_face ()
- def Rebarfunc.getSelectedFace (self)
- def Rebarfunc.showWarning (message)
- def Rebarfunc.translate (context, text, disambig=None)

Variables

- string Rebarfunc.__title__ = "GenericRebarFuctions"
- string Rebarfunc.__author__ = "Amritpal Singh"
- string Rebarfunc.__url__ = "https://www.freecadweb.org"
- Rebarfunc.SelectedObj
- Rebarfunc.FaceName

8.14 Rebarfunc.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
00023
00024 __title__ = "GenericRebarFuctions"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from DraftGeomUtils import vec, isCubic
00030 import FreeCAD
00031 import FreeCADGui
```

```
00032 import math
00033
00034 # ------------------------------------------------------------------------
00035 # Generic functions
00036 # ------------------------------------------------------------------------
00037
00038 def getEdgesAngle(edge1, edge2):
00039     """ getEdgesAngle(edge1, edge2): returns a angle between two edges."""
00040     vec1 = vec(edge1)
00041     vec2 = vec(edge2)
00042     angle = vec1.getAngle(vec2)
00043     angle = math.degrees(angle)
00044     return angle
00045
00046 def checkRectangle(edges):
00047     """ checkRectangle(edges=[]): This function checks whether the given form rectangle
00048         or not. It will return True when edges form rectangular shape or return False
00049         when edges not form a rectangular."""
00050     angles = [round(getEdgesAngle(edges[0], edges[1])), round(
      getEdgesAngle(edges[0], edges[2])),
00051               round(getEdgesAngle(edges[0], edges[3]))]
00052     if angles.count(90) == 2 and (angles.count(180) == 1 or angles.count(0) == 1):
00053         return True
00054     else:
00055         return False
00056
00057 def getBaseStructuralObject(obj):
00058     """ getBaseStructuralObject(obj): This function will return last base
00059         structural object."""
00060     if not obj.Base:
00061         return obj
00062     else:
00063         return getBaseStructuralObject(obj.Base)
00064
00065 def getBaseObject(obj):
00066     """ getBaseObject(obj): This function will return last base object."""
00067     if hasattr(obj, "Base"):
00068         return getBaseObject(obj.Base)
00069     else:
00070         return obj
00071
00072 def getFaceNumber(s):
00073     """ getFaceNumber(facename): This will return a face number from face name.
00074     For eg.:
00075         Input: "Face12"
00076         Output: 12"""
00077     head = s.rstrip('0123456789')
00078     tail = s[len(head):]
00079     return int(tail)
00080
00081 def facenormalDirection(structure = None, facename = None):
00082     if not structure and not facename:
00083         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00084         structure = selected_obj.Object
00085         facename = selected_obj.SubElementNames[0]
00086     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00087     normal = face.normalAt(0,0)
00088     normal = face.Placement.Rotation.inverted().multVec(normal)
00089     return normal
00090
00091 # ------------------------------------------------------------------------
00092 # Main functions which is use while creating any rebar.
00093 # ------------------------------------------------------------------------
00094
00095 def getTrueParametersOfStructure(obj):
00096     """ getTrueParametersOfStructure(obj): This function return actual length,
00097     width and height of the structural element in the form of array like
00098     [Length, Width, Height]"""
00099     baseObject = getBaseObject(obj)
00100     # If selected_obj is not derived from any base object
00101     if baseObject:
00102         # If selected_obj is derived from SketchObject
00103         if baseObject.isDerivedFrom("Sketcher::SketchObject"):
00104             edges = baseObject.Shape.Edges
00105             if checkRectangle(edges):
00106                 for edge in edges:
00107                     # Representation vector of edge
00108                     rep_vector = edge.Vertexes[1].Point.sub(edge.Vertexes[0].Point)
00109                     rep_vector_angle = round(math.degrees(rep_vector.getAngle(FreeCAD.Vector(1,0,0))))
00110                     if rep_vector_angle in {0, 180}:
00111                         length = edge.Length
00112                     else:
00113                         width = edge.Length
00114             else:
00115                 return None
00116         else:
00117             return None
```

```
00118         height = obj.Height.Value
00119     else:
00120         structuralBaseObject = getBaseStructuralObject(obj)
00121         length = structuralBaseObject.Length.Value
00122         width = structuralBaseObject.Width.Value
00123         height = structuralBaseObject.Height.Value
00124     return [length, width, height]
00125
00126 def getParametersOfFace(structure, facename, sketch = True):
00127     """ getParametersOfFace(structure, facename, sketch = True): This function will return
00128     length, width and points of center of mass of a given face according to the sketch
00129     value in the form of list.
00130
00131     For eg.:
00132     Case 1: When sketch is True: We use True when we want to create rebars from sketch
00133         (planar rebars) and the sketch is strictly based on 2D so we neglected the normal
00134         axis of the face.
00135         Output: [(FaceLength, FaceWidth), (CenterOfMassX, CenterOfMassY)]
00136
00137     Case 2: When sketch is False: When we want to create non-planar rebars(like stirrup)
00138         or we want to create rebar from a wire. Also for creating rebar from wire
00139         we will require three coordinates (x, y, z).
00140         Output: [(FaceLength, FaceWidth), (CenterOfMassX, CenterOfMassY, CenterOfMassZ)]"""
00141     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00142     center_of_mass = face.CenterOfMass
00143     #center_of_mass = center_of_mass.sub(getBaseStructuralObject(structure).Placement.Base)
00144     center_of_mass = center_of_mass.sub(structure.Placement.Base)
00145     Edges = []
00146     facePRM = []
00147     # When structure is cubic. It support all structure is derived from
00148     # any other object (like a sketch, wire etc).
00149     if isCubic(structure.Shape):
00150         print 423
00151         for edge in face.Edges:
00152             if not Edges:
00153                 Edges.append(edge)
00154             else:
00155                 # Checks whether similar edges is already present in Edges list
00156                 # or not.
00157                 if round((vec(edge)).Length) not in [round((vec(x)).Length) for x in Edges]:
00158                     Edges.append(edge)
00159         if len(Edges) == 1:
00160             Edges.append(edge)
00161         # facePRM holds length of a edges.
00162         facePRM = [(vec(edge)).Length for edge in Edges]
00163         # Find the orientation of the face. Also eliminating normal axes
00164         # to the edge/face.
00165         # When edge is parallel to x-axis
00166         if round(Edges[0].tangentAt(0)[0]) in {1,-1}:
00167             x = center_of_mass[0]
00168             if round(Edges[1].tangentAt(0)[1]) in {1, -1}:
00169                 y = center_of_mass[1]
00170             else:
00171                 y = center_of_mass[2]
00172         # When edge is parallel to y-axis
00173         elif round(Edges[0].tangentAt(0)[1]) in {1,-1}:
00174             x = center_of_mass[1]
00175             if round(Edges[1].tangentAt(0)[0]) in {1, -1}:
00176                 # Change order when edge along x-axis is at second place.
00177                 facePRM.reverse()
00178                 y = center_of_mass[1]
00179             else:
00180                 y = center_of_mass[2]
00181         elif round(Edges[0].tangentAt(0)[2]) in {1,-1}:
00182             y = center_of_mass[2]
00183             if round(Edges[1].tangentAt(0)[0]) in {1, -1}:
00184                 x = center_of_mass[0]
00185             else:
00186                 x = center_of_mass[1]
00187             facePRM.reverse()
00188         facelength = facePRM[0]
00189         facewidth = facePRM[1]
00190     # When structure is not cubic. For founding parameters of given face
00191     # I have used bounding box.
00192     else:
00193         boundbox = face.BoundBox
00194         # Check that one length of bounding box is zero. Here bounding box
00195         # looks like a plane.
00196         if 0 in {round(boundbox.XLength), round(boundbox.YLength), round(boundbox.ZLength)}:
00197             normal = face.normalAt(0,0)
00198             normal = face.Placement.Rotation.inverted().multVec(normal)
00199             #print "x: ", boundbox.XLength
00200             #print "y: ", boundbox.YLength
00201             #print "z: ", boundbox.ZLength
00202             # Set length and width of user selected face of structural element
00203             flag = True
00204             # FIXME: Improve below logic.
```

```
00205             for i in range(len(normal)):
00206                 if round(normal[i]) == 0:
00207                     if flag and i == 0:
00208                         x = center_of_mass[i]
00209                         facelength =  boundbox.XLength
00210                         flag = False
00211                     elif flag and i == 1:
00212                         x = center_of_mass[i]
00213                         facelength = boundbox.YLength
00214                         flag = False
00215                     if i == 1:
00216                         y = center_of_mass[i]
00217                         facewidth = boundbox.YLength
00218                     elif i == 2:
00219                         y = center_of_mass[i]
00220                         facewidth = boundbox.ZLength
00221             #print [(facelength, facewidth), (x, y)]
00222     # Return parameter of the face when rebar is not created from the sketch.
00223     # For eg. non-planar rebars like stirrup etc.
00224     if not sketch:
00225         center_of_mass = face.CenterOfMass
00226         return [(facelength, facewidth), center_of_mass]
00227     #TODO: Add support when bounding box have depth. Here bounding box looks
00228     # like cuboid. If we given curved face.
00229     return [(facelength, facewidth), (x, y)]
00230
00231 # ------------------------------------------------------------------------
00232 # Functions which is mainly used while creating stirrup.
00233 # ------------------------------------------------------------------------
00234
00235 def extendedTangentPartLength(rounding, diameter, angle):
00236     """ extendedTangentPartLength(rounding, diameter, angle): Get a extended
00237     length of rounding on corners."""
00238     radius = rounding * diameter
00239     x1 = radius / math.tan(math.radians(angle))
00240     x2 = radius / math.cos(math.radians(90 - angle)) - radius
00241     return x1 + x2
00242
00243 def extendedTangentLength(rounding, diameter, angle):
00244     """ extendedTangentLength(rounding, diameter, angle): Get a extended
00245     length of rounding at the end of Stirrup for bent."""
00246     radius = rounding * diameter
00247     x1 = radius / math.sin(math.radians(angle))
00248     x2 = radius * math.tan(math.radians(90 - angle))
00249     return x1 + x2
00250
00251 # ------------------------------------------------------------------------
00252 # Warning / Alert functions when user do something wrong.
00253 #-------------------------------------------------------------------------
00254
00255 def check_selected_face():
00256     """ check_selected_face(): This function checks whether user have selected
00257         any face or not."""
00258     selected_objs = FreeCADGui.Selection.getSelectionEx()
00259     if not selected_objs:
00260         showWarning("Select any face of the structural element.")
00261         selected_obj = None
00262     else:
00263         selected_face_names = selected_objs[0].SubElementNames
00264         if not selected_face_names:
00265             selected_obj = None
00266             showWarning("Select any face of the structural element.")
00267         elif "Face" in selected_face_names[0]:
00268             if len(selected_face_names) > 1:
00269                 showWarning("You have selected more than one face of the structural element.")
00270                 selected_obj = None
00271             elif len(selected_face_names) == 1:
00272                 selected_obj = selected_objs[0]
00273         else:
00274             showWarning("Select any face of the selected the face.")
00275             selected_obj = None
00276     return selected_obj
00277
00278 def getSelectedFace(self):
00279     selected_objs = FreeCADGui.Selection.getSelectionEx()
00280     if selected_objs:
00281         if len(selected_objs[0].SubObjects) == 1:
00282             if "Face" in selected_objs[0].SubElementNames[0]:
00283                 self.SelectedObj = selected_objs[0].Object
00284                 self.FaceName = selected_objs[0].SubElementNames[0]
00285                 self.form.PickSelectedFaceLabel.setText("Selected face is " + self.FaceName)
00286             else:
00287                 showWarning("Select any face of the structural element.")
00288         else:
00289             showWarning("Select only one face of the structural element.")
00290     else:
00291         showWarning("Select any face of the structural element.")
```

```
00292
00293 def showWarning(message):
00294     """ showWarning(message): This function is used to produce warning
00295     message for the user."""
00296     msg = QtGui.QMessageBox()
00297     msg.setIcon(QtGui.QMessageBox.Warning)
00298     msg.setText(translate("RebarAddon", message))
00299     msg.setStandardButtons(QtGui.QMessageBox.Ok)
00300     msg.exec_()
00301
00302 # Qt tanslation handling
00303 def translate(context, text, disambig=None):
00304     return QtCore.QCoreApplication.translate(context, text, disambig)
```

8.15 RebarTools.py File Reference

Classes

- class RebarTools.StraightRebarTool
- class RebarTools.UShapeRebarTool
- class RebarTools.LShapeRebarTool
- class RebarTools.StirrupTool
- class RebarTools.BentShapeRebarTool
- class RebarTools.HelicalRebarTool

Namespaces

- RebarTools

Variables

- string RebarTools.__title__ = "RebarCommands"
- string RebarTools.__author__ = "Amritpal Singh"
- string RebarTools.__url__ = "https://www.freecadweb.org"
- list RebarTools.RebarCommands = ["Arch_Rebar_Straight", "Arch_Rebar_UShape", "Arch_Rebar_LShape", "Arch_Rebar_Stirrup", "Arch_Rebar_BentShape", "Arch_Rebar_Helical"]

8.16 RebarTools.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
```

```
00023
00024 __title__ = "RebarCommands"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide.QtCore import QT_TRANSLATE_NOOP
00029 import FreeCADGui
00030 import os
00031
00032 class StraightRebarTool:
00033
00034     def GetResources(self):
00035         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/StraightRebar.svg',
00036                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Straight Rebar"),
00037                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a Striaght bar reinforcement from the
       selected face of the Structural element.")}
00038
00039     def IsActive(self):
00040         if FreeCADGui.ActiveDocument:
00041             return True
00042         else:
00043             return False
00044
00045     def Activated(self):
00046         import StraightRebar
00047         # Call to CommandStraightRebar() function
00048         StraightRebar.CommandStraightRebar()
00049
00050 class UShapeRebarTool:
00051
00052     def GetResources(self):
00053         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/UShapeRebar.svg',
00054                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "U-Shape Rebar"),
00055                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a U-Shape bar reinforcement from the
       selected face of the Structural element.")}
00056
00057     def IsActive(self):
00058         if FreeCADGui.ActiveDocument:
00059             return True
00060         else:
00061             return False
00062
00063     def Activated(self):
00064         import UShapeRebar
00065         # Call to CommandUShaepRebar() function
00066         UShapeRebar.CommandUShapeRebar()
00067
00068 class LShapeRebarTool:
00069
00070     def GetResources(self):
00071         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/LShapeRebar.svg',
00072                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "L-Shape Rebar"),
00073                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a L-Shape bar reinforcement from the
       selected face of the Structural element.")}
00074
00075     def IsActive(self):
00076         if FreeCADGui.ActiveDocument:
00077             return True
00078         else:
00079             return False
00080
00081     def Activated(self):
00082         import LShapeRebar
00083         # Call to CommandUShaepRebar() function
00084         LShapeRebar.CommandLShapeRebar()
00085
00086 class StirrupTool:
00087
00088     def GetResources(self):
00089         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/StirrupRebar.svg',
00090                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Stirrup"),
00091                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a Stirrup bar reinforcement from the
       selected face of the Structural element.")}
00092
00093     def IsActive(self):
00094         if FreeCADGui.ActiveDocument:
00095             return True
00096         else:
00097             return False
00098
00099     def Activated(self):
00100         import Stirrup
00101         # Call to CommandStirrup() function
```

```
00102         Stirrup.CommandStirrup()
00103
00104 class BentShapeRebarTool:
00105
00106     def GetResources(self):
00107         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/BentShapeRebar.svg',
00108                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Bent-Shape Rebar"),
00109                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a BentShape bar reinforcement from the
       selected face of the Structural element.")}
00110
00111     def IsActive(self):
00112         if FreeCADGui.ActiveDocument:
00113             return True
00114         else:
00115             return False
00116
00117     def Activated(self):
00118         import BentShapeRebar
00119         # Call to CommandBentShaepRebar() function
00120         BentShapeRebar.CommandBentShapeRebar()
00121
00122 class HelicalRebarTool:
00123
00124     def GetResources(self):
00125         return {'Pixmap'  : os.path.split(os.path.abspath(__file__))[0]+'
      /icons/dropdown_list/HelixShapeRebar.svg',
00126                 'MenuText': QT_TRANSLATE_NOOP("RebarAddon", "Helical Rebar"),
00127                 'ToolTip' : QT_TRANSLATE_NOOP("RebarAddon", "Creates a Helical bar reinforcement from the
       selected face of the Structural element.")}
00128
00129     def IsActive(self):
00130         if FreeCADGui.ActiveDocument:
00131             return True
00132         else:
00133             return False
00134
00135     def Activated(self):
00136         import HelicalRebar
00137         # Call to CommandHelicalRebar() function
00138         HelicalRebar.CommandHelicalRebar()
00139
00140 FreeCADGui.addCommand('Arch_Rebar_Straight', StraightRebarTool())
00141 FreeCADGui.addCommand('Arch_Rebar_UShape', UShapeRebarTool())
00142 FreeCADGui.addCommand('Arch_Rebar_LShape', LShapeRebarTool())
00143 FreeCADGui.addCommand('Arch_Rebar_Stirrup', StirrupTool())
00144 FreeCADGui.addCommand('Arch_Rebar_BentShape', BentShapeRebarTool())
00145 FreeCADGui.addCommand('Arch_Rebar_Helical', HelicalRebarTool())
00146
00147 # List of all rebar commands
00148 RebarCommands = ["Arch_Rebar_Straight", "Arch_Rebar_UShape", "Arch_Rebar_LShape", "Arch_Rebar_Stirrup", "
      Arch_Rebar_BentShape", "Arch_Rebar_Helical"]
```

8.17 Stirrup.py File Reference

Classes

- class Stirrup._StirrupTaskPanel

Namespaces

- Stirrup

Functions

- def Stirrup.getpointsOfStirrup (FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle, bentFactor, diameter, rounding, facenormal)
- def Stirrup.makeStirrup (l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter, rounding, amount_spacing_check, amount_spacing_value, structure=None, facename=None)
- def Stirrup.editStirrup (Rebar, l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter, rounding, amount_spacing_check, amount_spacing_value, structure=None, facename=None)
- def Stirrup.editDialog (vobj)
- def Stirrup.CommandStirrup ()

Variables

- string Stirrup.__title__ = "StirrupRebar"
- string Stirrup.__author__ = "Amritpal Singh"
- string Stirrup.__url__ = "https://www.freecadweb.org"

8.18 Stirrup.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
00023
00024 __title__ = "StirrupRebar"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 from RebarDistribution import runRebarDistribution, removeRebarDistribution
00032 from PopUpImage import showPopUpImageDialog
00033 import FreeCAD
00034 import FreeCADGui
00035 import ArchCommands
00036 import os
00037 import sys
00038 import math
00039
00040 def getpointsOfStirrup(FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle,
      bentFactor, diameter, rounding, facenormal):
00041     """ getpointsOfStirrup(FacePRM, LeftCover, RightCover, TopCover, BottomCover, BentAngle, BentFactor,
       Diameter, Rounding, FaceNormal):
00042     Return the coordinates points of the Stirrup in the form of array."""
00043     angle = 180 - bentAngle
00044     tangent_part_length = extendedTangentPartLength(rounding, diameter, angle)
00045     tangent_length = extendedTangentLength(rounding, diameter, angle)
00046     if round(facenormal[0]) in {1,-1}:
00047         x1 = FacePRM[1][0]
00048         y1 = FacePRM[1][1] - FacePRM[0][0] / 2 + l_cover
00049         z1 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover + tangent_part_length
00050         y2 = FacePRM[1][1] - FacePRM[0][0] / 2 + l_cover
00051         z2 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00052         y3 = FacePRM[1][1] + FacePRM[0][0] / 2 - r_cover
00053         z3 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00054         y4 = FacePRM[1][1] + FacePRM[0][0] / 2 - r_cover
00055         z4 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00056         y5 = FacePRM[1][1] - FacePRM[0][0] / 2 + l_cover - tangent_part_length
00057         z5 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00058         side_length = abs(y5 - y4) - tangent_part_length
00059         normal_dis = (diameter * (side_length + tangent_part_length)) / side_length
00060         x2 = x1 - normal_dis / 4
00061         x3 = x2 - normal_dis / 4
00062         x4 = x3 - normal_dis / 4
00063         x5 = x4 - normal_dis / 4
00064         x0 = x1 + normal_dis / 4
00065         y0 = y1 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(angle))
00066         z0 = z1 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(angle))
00067         x6 = x5 - normal_dis / 4
00068         y6 = y5 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(90 - angle))
00069         z6 = z5 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(90 - angle))
```

```
00070     elif round(facenormal[1]) in {1,-1}:
00071         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00072         y1 = FacePRM[1][1]
00073         z1 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover + tangent_part_length
00074         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00075         z2 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00076         x3 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00077         z3 = FacePRM[1][2] - FacePRM[0][1] / 2 + b_cover
00078         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00079         z4 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00080         x5 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover - tangent_part_length
00081         z5 = FacePRM[1][2] + FacePRM[0][1] / 2 - t_cover
00082         side_length = abs(x5 - x4) - tangent_part_length
00083         normal_dis = (diameter * (side_length + tangent_part_length)) / side_length
00084         y2 = y1 - normal_dis / 4
00085         y3 = y2 - normal_dis / 4
00086         y4 = y3 - normal_dis / 4
00087         y5 = y4 - normal_dis / 4
00088         y0 = y1 + normal_dis / 4
00089         x0 = x1 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(angle))
00090         z0 = z1 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(angle))
00091         x6 = x5 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(90 - angle))
00092         y6 = y5 - normal_dis / 4
00093         z6 = z5 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(90 - angle))
00094     elif round(facenormal[2]) in {1,-1}:
00095         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00096         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover + tangent_part_length
00097         z1 = FacePRM[1][2]
00098         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00099         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00100         x3 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00101         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00102         x4 = FacePRM[1][0] + FacePRM[0][0] / 2 - r_cover
00103         y4 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00104         x5 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover - tangent_part_length
00105         y5 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00106         side_length = abs(x5 - x4) - tangent_part_length
00107         normal_dis = (diameter * (side_length + tangent_part_length)) / side_length
00108         z2 = z1 - normal_dis / 4
00109         z3 = z2 - normal_dis / 4
00110         z4 = z3 - normal_dis / 4
00111         z5 = z4 - normal_dis / 4
00112         z0 = z1 + normal_dis / 4
00113         x0 = x1 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(angle))
00114         y0 = y1 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(angle))
00115         x6 = x5 + (tangent_length + bentFactor * diameter) * math.sin(math.radians(90 - angle))
00116         y6 = y5 - (tangent_length + bentFactor * diameter) * math.cos(math.radians(90 - angle))
00117         z6 = z5 - normal_dis / 4
00118     return [FreeCAD.Vector(x0, y0, z0), FreeCAD.Vector(x1, y1, z1),\
00119             FreeCAD.Vector(x2, y2, z2), FreeCAD.Vector(x3, y3, z3),\
00120             FreeCAD.Vector(x4, y4, z4), FreeCAD.Vector(x5, y5, z5),\
00121             FreeCAD.Vector(x6, y6, z6)]
00122
00123 class _StirrupTaskPanel:
00124     def __init__(self, Rebar = None):
00125         self.CustomSpacing = None
00126         if not Rebar:
00127             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00128             self.SelectedObj = selected_obj.Object
00129             self.FaceName = selected_obj.SubElementNames[0]
00130         else:
00131             self.FaceName = Rebar.Base.Support[0][1][0]
00132             self.SelectedObj = Rebar.Base.Support[0][0]
00133         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00134         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "Stirrup Rebar", None))
00135         self.form.bentAngle.addItems(["135", "90"])
00136         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00137         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00138         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0]+"
      /icons/Stirrup.svg"))
00139         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00140         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00141         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00142         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00143         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/StirrupDetailed.svg"))
00144         self.Rebar = Rebar
00145
00146     def getStandardButtons(self):
00147         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00148
00149     def clicked(self, button):
00150         if button == int(QtGui.QDialogButtonBox.Apply):
00151             self.accept(button)
```

```
00152
00153     def accept(self, signal = None):
00154         l_cover = self.form.l_sideCover.text()
00155         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00156         r_cover = self.form.r_sideCover.text()
00157         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00158         t_cover = self.form.t_sideCover.text()
00159         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00160         b_cover = self.form.b_sideCover.text()
00161         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00162         f_cover = self.form.frontCover.text()
00163         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00164         diameter = self.form.diameter.text()
00165         diameter = FreeCAD.Units.Quantity(diameter).Value
00166         bentAngle = int(self.form.bentAngle.currentText())
00167         bentFactor = self.form.bentFactor.value()
00168         rounding = self.form.rounding.value()
00169         amount_check = self.form.amount_radio.isChecked()
00170         spacing_check = self.form.spacing_radio.isChecked()
00171         if not self.Rebar:
00172             if amount_check:
00173                 amount = self.form.amount.value()
00174                 rebar = makeStirrup(l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle,
    bentFactor, diameter,\
00175                     rounding, True, amount, self.SelectedObj, self.
    FaceName)
00176             elif spacing_check:
00177                 spacing = self.form.spacing.text()
00178                 spacing = FreeCAD.Units.Quantity(spacing).Value
00179                 rebar = makeStirrup(l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle,
    bentFactor, diameter,\
00180                     rounding, False, spacing, self.SelectedObj, self.
    FaceName)
00181         else:
00182             if amount_check:
00183                 amount = self.form.amount.value()
00184                 rebar = editStirrup(self.Rebar, l_cover, r_cover, t_cover, b_cover, f_cover
    , bentAngle, bentFactor,\
00185                     diameter, rounding, True, amount, self.SelectedObj, self.
    FaceName)
00186             elif spacing_check:
00187                 spacing = self.form.spacing.text()
00188                 spacing = FreeCAD.Units.Quantity(spacing).Value
00189                 rebar = editStirrup(self.Rebar, l_cover, r_cover, t_cover, b_cover, f_cover
    , bentAngle, bentFactor,\
00190                     diameter, rounding, False, spacing, self.SelectedObj, self.
    FaceName)
00191         if self.CustomSpacing:
00192             rebar.CustomSpacing = self.CustomSpacing
00193             FreeCAD.ActiveDocument.recompute()
00194         self.Rebar = rebar
00195         if signal == int(QtGui.QDialogButtonBox.Apply):
00196             pass
00197         else:
00198             FreeCADGui.Control.closeDialog(self)
00199
00200     def amount_radio_clicked(self):
00201         self.form.spacing.setEnabled(False)
00202         self.form.amount.setEnabled(True)
00203
00204     def spacing_radio_clicked(self):
00205         self.form.amount.setEnabled(False)
00206         self.form.spacing.setEnabled(True)
00207
00208
00209 def makeStirrup(l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor, diameter,
    rounding,\
00210         amount_spacing_check, amount_spacing_value, structure = None, facename = None):
00211     """ makeStirrup(LeftCover, RightCover, TopCover, BottomCover, FrontCover, BentAngle,
00212     BentFactor, Diameter, Rounding, AmountSpacingCheck, AmountSpacingValue, Structure, Facename):
00213     Adds the Stirrup reinforcement bar to the selected structural object."""
00214     if not structure and not facename:
00215         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00216         structure = selected_obj.Object
00217         facename = selected_obj.SubElementNames[0]
00218     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00219     #StructurePRM = getTrueParametersOfStructure(structure)
00220     FacePRM = getParametersOfFace(structure, facename, False)
00221     FaceNormal = face.normalAt(0,0)
00222     #FaceNormal = face.Placement.Rotation.inverted().multVec(FaceNormal)
00223     if not FacePRM:
00224         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
     derived\n")
00225         return
00226     # Calculate the coordinate values of Stirrup
00227     points = getpointsOfStirrup(FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle,
    bentFactor, diameter, rounding, FaceNormal)
```

```
00228     import Draft
00229     line = Draft.makeWire(points, closed = False, face = True, support = None)
00230     import Arch
00231     line.Support = [(structure, facename)]
00232     if amount_spacing_check:
00233         rebar = Arch.makeRebar(structure, line, diameter, amount_spacing_value, f_cover)
00234     else:
00235         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00236         rebar = Arch.makeRebar(structure, line, diameter,\
00237             int((size - diameter) / amount_spacing_value), f_cover)
00238     rebar.Direction = FaceNormal.negative()
00239     rebar.Rounding = rounding
00240     # Adds properties to the rebar object
00241     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog",\
00242         QT_TRANSLATE_NOOP("App::Property","Shape of rebar")).RebarShape = "Stirrup"
00243     rebar.ViewObject.setEditorMode("RebarShape", 2)
00244     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog",\
00245         QT_TRANSLATE_NOOP("App::Property", "Left Side cover of rebar")).LeftCover = l_cover
00246     rebar.setEditorMode("LeftCover", 2)
00247     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog",\
00248         QT_TRANSLATE_NOOP("App::Property", "Right Side cover of rebar")).RightCover = r_cover
00249     rebar.setEditorMode("RightCover", 2)
00250     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog",\
00251         QT_TRANSLATE_NOOP("App::Property", "Top Side cover of rebar")).TopCover = t_cover
00252     rebar.setEditorMode("TopCover", 2)
00253     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog",\
00254         QT_TRANSLATE_NOOP("App::Property", "Bottom Side cover of rebar")).BottomCover = b_cover
00255     rebar.setEditorMode("BottomCover", 2)
00256     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog",\
00257         QT_TRANSLATE_NOOP("App::Property", "Top cover of rebar")).FrontCover = f_cover
00258     rebar.setEditorMode("FrontCover", 2)
00259     rebar.addProperty("App::PropertyInteger", "BentAngle", "RebarDialog",\
00260         QT_TRANSLATE_NOOP("App::Property", "Bent angle between at the end of rebar")).BentAngle = bentAngle
00261     rebar.setEditorMode("BentAngle", 2)
00262     rebar.addProperty("App::PropertyInteger", "BentFactor", "RebarDialog",\
00263         QT_TRANSLATE_NOOP("App::Property", "Bent Length is the equal to BentFactor * Diameter")).BentFactor
      = bentFactor
00264     rebar.setEditorMode("BentFactor", 2)
00265     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog",\
00266         QT_TRANSLATE_NOOP("App::Property", "Amount radio button is checked")).AmountCheck
00267     rebar.setEditorMode("AmountCheck", 2)
00268     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog",\
00269         QT_TRANSLATE_NOOP("App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00270     rebar.setEditorMode("TrueSpacing", 2)
00271     if amount_spacing_check:
00272         rebar.AmountCheck = True
00273     else:
00274         rebar.AmountCheck = False
00275         rebar.TrueSpacing = amount_spacing_value
00276     rebar.Label = "Stirrup"
00277     FreeCAD.ActiveDocument.recompute()
00278     return rebar
00279
00280 def editStirrup(Rebar, l_cover, r_cover, t_cover, b_cover, f_cover, bentAngle, bentFactor,
    diameter, rounding,\
00281         amount_spacing_check, amount_spacing_value, structure = None, facename = None):
00282     sketch = Rebar.Base
00283     if structure and facename:
00284         sketch.Support = [(structure, facename)]
00285     # Check if sketch support is empty.
00286     if not sketch.Support:
00287         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
      unchecked Edit->Preferences->Arch.")
00288         return
00289     # Assigned values
00290     facename = sketch.Support[0][1][0]
00291     structure = sketch.Support[0][0]
00292     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00293     #StructurePRM = getTrueParametersOfStructure(structure)
00294     # Get parameters of the face where sketch of rebar is drawn
00295     FacePRM = getParametersOfFace(structure, facename, False)
00296     FaceNormal = face.normalAt(0, 0)
00297     #FaceNormal = face.Placement.Rotation.inverted().multVec(FaceNormal)
00298     # Calculate the coordinates value of Stirrup rebar
00299     points = getpointsOfStirrup(FacePRM, l_cover, r_cover, t_cover, b_cover, bentAngle,
    bentFactor, diameter, rounding, FaceNormal)
00300     Rebar.Base.Points = points
00301     FreeCAD.ActiveDocument.recompute()
00302     Rebar.Direction = FaceNormal.negative()
00303     Rebar.OffsetStart = f_cover
00304     Rebar.OffsetEnd = f_cover
00305     Rebar.BentAngle = bentAngle
00306     Rebar.BentFactor = bentFactor
00307     Rebar.Rounding = rounding
00308     Rebar.Diameter = diameter
00309     if amount_spacing_check:
00310         Rebar.Amount = amount_spacing_value
```

```
00311         FreeCAD.ActiveDocument.recompute()
00312         Rebar.AmountCheck = True
00313     else:
00314         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00315         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00316         FreeCAD.ActiveDocument.recompute()
00317         Rebar.AmountCheck = False
00318     Rebar.FrontCover = f_cover
00319     Rebar.LeftCover = l_cover
00320     Rebar.RightCover = r_cover
00321     Rebar.TopCover = t_cover
00322     Rebar.BottomCover = b_cover
00323     Rebar.TrueSpacing = amount_spacing_value
00324     FreeCAD.ActiveDocument.recompute()
00325     return Rebar
00326
00327 def editDialog(vobj):
00328     FreeCADGui.Control.closeDialog()
00329     obj = _StirrupTaskPanel(vobj.Object)
00330     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00331     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00332     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00333     obj.form.t_sideCover.setText(str(vobj.Object.TopCover))
00334     obj.form.b_sideCover.setText(str(vobj.Object.BottomCover))
00335     obj.form.diameter.setText(str(vobj.Object.Diameter))
00336     obj.form.bentAngle.setCurrentIndex(obj.form.bentAngle.findText(str(vobj.Object.BentAngle)))
00337     obj.form.bentFactor.setValue(vobj.Object.BentFactor)
00338     obj.form.rounding.setValue(vobj.Object.Rounding)
00339     if vobj.Object.AmountCheck:
00340         obj.form.amount.setValue(vobj.Object.Amount)
00341     else:
00342         obj.form.amount_radio.setChecked(False)
00343         obj.form.spacing_radio.setChecked(True)
00344         obj.form.amount.setDisabled(True)
00345         obj.form.spacing.setEnabled(True)
00346         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00347     #obj.form.PickSelectedFace.setVisible(False)
00348     FreeCADGui.Control.showDialog(obj)
00349
00350 def CommandStirrup():
00351     selected_obj = check_selected_face()
00352     if selected_obj:
00353         FreeCADGui.Control.showDialog(_StirrupTaskPanel())
```

8.19 StraightRebar.py File Reference

Classes

- class StraightRebar._StraightRebarTaskPanel

Namespaces

- StraightRebar

Functions

- def StraightRebar.getpointsOfStraightRebar (FacePRM, rt_cover, lb_cover, coverAlong, orientation)
- def StraightRebar.makeStraightRebar (f_cover, coverAlong, rt_cover, lb_cover, diameter, amount_spacing↩ _check, amount_spacing_value, orientation="Horizontal", structure=None, facename=None)
- def StraightRebar.editStraightRebar (Rebar, f_cover, coverAlong, rt_cover, lb_cover, diameter, amount_↩ spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def StraightRebar.editDialog (vobj)
- def StraightRebar.CommandStraightRebar ()

Variables

- string StraightRebar.__title__ = "StraightRebar"
- string StraightRebar.__author__ = "Amritpal Singh"
- string StraightRebar.__url__ = "https://www.freecadweb.org"

8.20 StraightRebar.py

```
00001 # -*- coding: utf-8 -*-
00002 # ***************************************************************************
00003 # *                                                                         *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>             *
00005 # *                                                                         *
00006 # *   This program is free software; you can redistribute it and/or modify  *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)    *
00008 # *   as published by the Free Software Foundation; either version 2 of     *
00009 # *   the License, or (at your option) any later version.                   *
00010 # *   for detail see the LICENCE text file.                                 *
00011 # *                                                                         *
00012 # *   This program is distributed in the hope that it will be useful,       *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of        *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the         *
00015 # *   GNU Library General Public License for more details.                  *
00016 # *                                                                         *
00017 # *   You should have received a copy of the GNU Library General Public     *
00018 # *   License along with this program; if not, write to the Free Software   *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307  *
00020 # *   USA                                                                   *
00021 # *                                                                         *
00022 # ***************************************************************************
00023
00024 __title__ = "StraightRebar"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 from RebarDistribution import runRebarDistribution, removeRebarDistribution
00032 from PopUpImage import showPopUpImageDialog
00033 import FreeCAD
00034 import FreeCADGui
00035 import ArchCommands
00036 import os
00037 import sys
00038 import math
00039
00040 def getpointsOfStraightRebar(FacePRM, rt_cover, lb_cover, coverAlong, orientation):
00041     """ getpointsOfStraightRebar(FacePRM, RightTopcover, LeftBottomcover, CoverAlong, Orientation):
00042     Return points of the Straight rebar in the form of array for sketch.
00043
00044     Case I: When Orientation is 'Horizontal':
00045         We have two option in CoverAlong i.e. 'Bottom Side' or 'Top Side'
00046     Case II: When Orientation is 'Vertical':
00047         We have two option in CoverAlong i.e. 'Left Side' or 'Right Side'
00048     """
00049     if orientation == "Horizontal":
00050         if coverAlong[0] == "Bottom Side":
00051             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + lb_cover
00052             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + coverAlong[1]
00053             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - rt_cover
00054             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + coverAlong[1]
00055         elif coverAlong[0] == "Top Side":
00056             cover = FacePRM[0][1] - coverAlong[1]
00057             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + lb_cover
00058             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + cover
00059             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - rt_cover
00060             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + cover
00061     elif orientation == "Vertical":
00062         if coverAlong[0] == "Left Side":
00063             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + coverAlong[1]
00064             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + lb_cover
00065             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + coverAlong[1]
00066             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + FacePRM[0][1] - rt_cover
00067         elif coverAlong[0] == "Right Side":
00068             cover = FacePRM[0][0] - coverAlong[1]
00069             x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + cover
00070             y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + lb_cover
00071             x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + cover
```

```
00072             y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + FacePRM[0][1] - rt_cover
00073     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0)]
00074
00075 class _StraightRebarTaskPanel:
00076     def __init__(self, Rebar = None):
00077         self.CustomSpacing = None
00078         if not Rebar:
00079             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00080             self.SelectedObj = selected_obj.Object
00081             self.FaceName = selected_obj.SubElementNames[0]
00082         else:
00083             self.FaceName = Rebar.Base.Support[0][1][0]
00084             self.SelectedObj = Rebar.Base.Support[0][0]
00085         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00086         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "Straight Rebar", None))
00087         self.form.orientation.addItems(["Horizontal", "Vertical"])
00088         self.form.coverAlong.addItems(["Bottom Side", "Top Side"])
00089         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00090         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00091         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00092         self.form.removeCustomSpacing.clicked.connect(lambda:
    removeRebarDistribution(self))
00093         self.form.PickSelectedFace.setCheckable(True)
00094         self.form.PickSelectedFace.toggle()
00095         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00096         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
    /icons/StraightRebarH.svg"))
00097         self.form.orientation.currentIndexChanged.connect(self.changeOrientation)
00098         self.form.coverAlong.currentIndexChanged.connect(self.changeCoverAlong)
00099         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
    QtGui.QStyle.SP_DialogHelpButton))
00100         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
    (os.path.abspath(__file__))[0] + "/icons/StraightRebarDetailed.svg"))
00101         self.Rebar = Rebar
00102
00103     def changeOrientation(self):
00104         orientation = self.form.orientation.currentText()
00105         if orientation == "Horizontal":
00106             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
    /icons/StraightRebarH.svg"))
00107             self.form.r_sideCoverLabel.setText("Right Side Cover")
00108             self.form.l_sideCoverLabel.setText("Left Side Cover")
00109             self.form.coverAlong.clear()
00110             self.form.coverAlong.addItems(["Bottom Side", "Top Side"])
00111         else:
00112             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
    /icons/StraightRebarV.svg"))
00113             self.form.r_sideCoverLabel.setText("Top Side Cover")
00114             self.form.l_sideCoverLabel.setText("Bottom Side Cover")
00115             self.form.coverAlong.clear()
00116             self.form.coverAlong.addItems(["Right Side", "Left Side"])
00117
00118     def changeCoverAlong(self):
00119         coverAlong = self.form.coverAlong.currentText()
00120         if coverAlong == "Bottom Side":
00121             self.form.bottomCoverLabel.setText("Bottom Cover")
00122         elif coverAlong == "Top Side":
00123             self.form.bottomCoverLabel.setText("Top Cover")
00124         elif coverAlong == "Right Side":
00125             self.form.bottomCoverLabel.setText("Right Cover")
00126         else:
00127             self.form.bottomCoverLabel.setText("Left Cover")
00128
00129     def getStandardButtons(self):
00130         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
    QtGui.QDialogButtonBox.Cancel)
00131
00132     def clicked(self, button):
00133         if button == int(QtGui.QDialogButtonBox.Apply):
00134             self.accept(button)
00135
00136     def accept(self, signal = None):
00137         f_cover = self.form.frontCover.text()
00138         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00139         cover = self.form.bottomCover.text()
00140         cover = FreeCAD.Units.Quantity(cover).Value
00141         lb_cover = self.form.l_sideCover.text()
00142         lb_cover = FreeCAD.Units.Quantity(lb_cover).Value
00143         rt_cover = self.form.r_sideCover.text()
00144         rt_cover = FreeCAD.Units.Quantity(rt_cover).Value
00145         orientation = self.form.orientation.currentText()
00146         coverAlong = self.form.coverAlong.currentText()
00147         diameter = self.form.diameter.text()
00148         diameter = FreeCAD.Units.Quantity(diameter).Value
00149         amount_check = self.form.amount_radio.isChecked()
00150         spacing_check = self.form.spacing_radio.isChecked()
00151         if not self.Rebar:
```

```
00152             if amount_check:
00153                 amount = self.form.amount.value()
00154                 rebar = makeStraightRebar(f_cover, (coverAlong, cover), rt_cover, lb_cover
      , diameter, True, amount, orientation, self.SelectedObj, self.FaceName)
00155             elif spacing_check:
00156                 spacing = self.form.spacing.text()
00157                 spacing = FreeCAD.Units.Quantity(spacing).Value
00158                 rebar = makeStraightRebar(f_cover, (coverAlong, cover), rt_cover, lb_cover
      , diameter, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00159         else:
00160             if amount_check:
00161                 amount = self.form.amount.value()
00162                 rebar = editStraightRebar(self.Rebar, f_cover, (coverAlong, cover),
      rt_cover, lb_cover, diameter, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00163             elif spacing_check:
00164                 spacing = self.form.spacing.text()
00165                 spacing = FreeCAD.Units.Quantity(spacing).Value
00166                 rebar = editStraightRebar(self.Rebar, f_cover, (coverAlong, cover),
      rt_cover, lb_cover, diameter, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00167         if self.CustomSpacing:
00168             rebar.CustomSpacing = self.CustomSpacing
00169             FreeCAD.ActiveDocument.recompute()
00170         self.Rebar = rebar
00171         if signal == int(QtGui.QDialogButtonBox.Apply):
00172             pass
00173         else:
00174             FreeCADGui.Control.closeDialog(self)
00175
00176     def amount_radio_clicked(self):
00177         self.form.spacing.setEnabled(False)
00178         self.form.amount.setEnabled(True)
00179
00180     def spacing_radio_clicked(self):
00181         self.form.amount.setEnabled(False)
00182         self.form.spacing.setEnabled(True)
00183
00184
00185 def makeStraightRebar(f_cover, coverAlong, rt_cover, lb_cover, diameter,
      amount_spacing_check, amount_spacing_value, orientation = "Horizontal", structure = None, facename = None):
00186     """ Adds the straight reinforcement bar to the selected structural object.
00187
00188     Case I: When orientation of straight rebar is 'Horizontal':
00189         makeStraightRebar(FrontCover, CoverAlong, RightCover, LeftCover, Diameter, AmountSpacingCheck,
       AmountSpacingValue, Orientation = "Horizontal",
00190         Structure, Facename)
00191         Note: Type of CoverAlong argument is a tuple. Syntax: (<Along>, <Value>). Here we have horizontal
       orientation so we can pass Top Side
00192         and Bottom Side to <Along> arguments.
00193         For eg. ("Top Side", 20) and ("Bottom Side", 20)
00194
00195     Case II: When orientation of straight rebar is 'Vertical':
00196         makeStraightRebar(FrontCover, CoverAlong, TopCover, BottomCover, Diameter, AmountSpacingCheck,
       AmountSpacingValue, Orientation = "Horizontal",
00197         Structure, Facename)
00198         Note: Type of CoverAlong argument is a tuple. Syntax: (<Along>, <Value>). Here we have vertical
       orientation so we can pass Left Side
00199         and Right Side to <Along> arguments.
00200         For eg. ("Left Side", 20) and ("Right Side", 20)
00201     """
00202     if not structure and not facename:
00203         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00204         structure = selected_obj.Object
00205         facename = selected_obj.SubElementNames[0]
00206     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00207     #StructurePRM = getTrueParametersOfStructure(structure)
00208     FacePRM = getParametersOfFace(structure, facename)
00209     if not FacePRM:
00210         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
       derived\n")
00211         return
00212     # Get points of Striaght rebar
00213     points = getpointsOfStraightRebar(FacePRM, rt_cover, lb_cover, coverAlong,
      orientation)
00214     import Part
00215     import Arch
00216     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00217     sketch.MapMode = "FlatFace"
00218     sketch.Support = [(structure, facename)]
00219     FreeCAD.ActiveDocument.recompute()
00220     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00221     if amount_spacing_check:
00222         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00223         FreeCAD.ActiveDocument.recompute()
00224     else:
```

```
00225         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00226         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
      f_cover)
00227     # Adds properties to the rebar object
00228     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Shape of rebar")).RebarShape = "StraightRebar"
00229     rebar.ViewObject.setEditorMode("RebarShape", 2)
00230     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Front cover of rebar")).FrontCover = f_cover
00231     rebar.setEditorMode("FrontCover", 2)
00232     rebar.addProperty("App::PropertyDistance", "RightTopCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Right/Top Side cover of rebar")).RightTopCover = rt_cover
00233     rebar.setEditorMode("RightTopCover", 2)
00234     rebar.addProperty("App::PropertyDistance", "LeftBottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Left/Bottom Side cover of rebar")).LeftBottomCover = lb_cover
00235     rebar.setEditorMode("LeftBottomCover", 2)
00236     rebar.addProperty("App::PropertyString", "CoverAlong", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
      , "Cover along")).CoverAlong = coverAlong[0]
00237     rebar.setEditorMode("CoverAlong", 2)
00238     rebar.addProperty("App::PropertyDistance", "Cover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property", "
      Cover of rebar along user selected side")).Cover = coverAlong[1]
00239     rebar.setEditorMode("Cover", 2)
00240     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
       "Amount radio button is checked")).AmountCheck
00241     rebar.setEditorMode("AmountCheck", 2)
00242     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
      App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00243     rebar.setEditorMode("TrueSpacing", 2)
00244     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Shape of rebar")).Orientation = orientation
00245     rebar.setEditorMode("Orientation", 2)
00246     if amount_spacing_check:
00247         rebar.AmountCheck = True
00248     else:
00249         rebar.AmountCheck = False
00250         rebar.TrueSpacing = amount_spacing_value
00251     rebar.Label = "StraightRebar"
00252     FreeCAD.ActiveDocument.recompute()
00253     return rebar
00254
00255 def editStraightRebar(Rebar, f_cover, coverAlong, rt_cover, lb_cover, diameter,
      amount_spacing_check, amount_spacing_value, orientation, structure = None, facename = None):
00256     sketch = Rebar.Base
00257     if structure and facename:
00258         sketch.Support = [(structure, facename)]
00259         FreeCAD.ActiveDocument.recompute()
00260     # Check if sketch support is empty.
00261     if not sketch.Support:
00262         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00263         return
00264     # Assigned values
00265     facename = sketch.Support[0][1][0]
00266     structure = sketch.Support[0][0]
00267     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00268     #StructurePRM = getTrueParametersOfStructure(structure)
00269     # Get parameters of the face where sketch of rebar is drawn
00270     FacePRM = getParametersOfFace(structure, facename)
00271     # Get points of Striaght rebar
00272     points = getpointsOfStraightRebar(FacePRM, rt_cover, lb_cover, coverAlong,
      orientation)
00273     sketch.movePoint(0, 1, points[0], 0)
00274     FreeCAD.ActiveDocument.recompute()
00275     sketch.movePoint(0, 2, points[1], 0)
00276     FreeCAD.ActiveDocument.recompute()
00277     Rebar.OffsetStart = f_cover
00278     Rebar.OffsetEnd = f_cover
00279     if amount_spacing_check:
00280         Rebar.Amount = amount_spacing_value
00281         FreeCAD.ActiveDocument.recompute()
00282         Rebar.AmountCheck = True
00283     else:
00284         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00285         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00286         FreeCAD.ActiveDocument.recompute()
00287         Rebar.AmountCheck = False
00288     Rebar.FrontCover = f_cover
00289     Rebar.RightTopCover = rt_cover
00290     Rebar.LeftBottomCover = lb_cover
00291     Rebar.CoverAlong = coverAlong[0]
00292     Rebar.Cover = coverAlong[1]
00293     Rebar.TrueSpacing = amount_spacing_value
00294     Rebar.Diameter = diameter
00295     Rebar.Orientation = orientation
00296     FreeCAD.ActiveDocument.recompute()
00297     return Rebar
00298
```

```
00299 def editDialog(vobj):
00300     FreeCADGui.Control.closeDialog()
00301     obj = _StraightRebarTaskPanel(vobj.Object)
00302     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00303     obj.form.r_sideCover.setText(str(vobj.Object.RightTopCover))
00304     obj.form.l_sideCover.setText(str(vobj.Object.LeftBottomCover))
00305     obj.form.bottomCover.setText(str(vobj.Object.Cover))
00306     obj.form.diameter.setText(str(vobj.Object.Diameter))
00307     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00308     obj.form.coverAlong.setCurrentIndex(obj.form.coverAlong.findText(str(vobj.Object.CoverAlong)))
00309     if vobj.Object.AmountCheck:
00310         obj.form.amount.setValue(vobj.Object.Amount)
00311     else:
00312         obj.form.amount_radio.setChecked(False)
00313         obj.form.spacing_radio.setChecked(True)
00314         obj.form.amount.setDisabled(True)
00315         obj.form.spacing.setEnabled(True)
00316         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00317     #obj.form.PickSelectedFace.setVisible(False)
00318     FreeCADGui.Control.showDialog(obj)
00319
00320 def CommandStraightRebar():
00321     selected_obj = check_selected_face()
00322     if selected_obj:
00323         FreeCADGui.Control.showDialog(_StraightRebarTaskPanel())
```

8.21 UShapeRebar.py File Reference

Classes

- class UShapeRebar._UShapeRebarTaskPanel

Namespaces

- UShapeRebar

Functions

- def UShapeRebar.getpointsOfUShapeRebar (FacePRM, r_cover, l_cover, b_cover, t_cover, orientation)
- def UShapeRebar.makeUShapeRebar (f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation="Bottom", structure=None, facename=None)
- def UShapeRebar.editUShapeRebar (Rebar, f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding, amount_spacing_check, amount_spacing_value, orientation, structure=None, facename=None)
- def UShapeRebar.editDialog (vobj)
- def UShapeRebar.CommandUShapeRebar ()

Variables

- string UShapeRebar.__title__ = "UShapeRebar"
- string UShapeRebar.__author__ = "Amritpal Singh"
- string UShapeRebar.__url__ = "https://www.freecadweb.org"

8.22 UShapeRebar.py

```
00001 # -*- coding: utf-8 -*-
00002 # ****************************************************************************
00003 # *                                                                          *
00004 # *   Copyright (c) 2017 - Amritpal Singh <amrit3701@gmail.com>              *
00005 # *                                                                          *
00006 # *   This program is free software; you can redistribute it and/or modify   *
00007 # *   it under the terms of the GNU Lesser General Public License (LGPL)     *
00008 # *   as published by the Free Software Foundation; either version 2 of      *
00009 # *   the License, or (at your option) any later version.                    *
00010 # *   for detail see the LICENCE text file.                                  *
00011 # *                                                                          *
00012 # *   This program is distributed in the hope that it will be useful,        *
00013 # *   but WITHOUT ANY WARRANTY; without even the implied warranty of         *
00014 # *   MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the          *
00015 # *   GNU Library General Public License for more details.                   *
00016 # *                                                                          *
00017 # *   You should have received a copy of the GNU Library General Public      *
00018 # *   License along with this program; if not, write to the Free Software    *
00019 # *   Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA  02111-1307   *
00020 # *   USA                                                                    *
00021 # *                                                                          *
00022 # ****************************************************************************
00023
00024 __title__ = "UShapeRebar"
00025 __author__ = "Amritpal Singh"
00026 __url__ = "https://www.freecadweb.org"
00027
00028 from PySide import QtCore, QtGui
00029 from Rebarfunc import *
00030 from PySide.QtCore import QT_TRANSLATE_NOOP
00031 from RebarDistribution import runRebarDistribution, removeRebarDistribution
00032 from PopUpImage import showPopUpImageDialog
00033 import FreeCAD
00034 import FreeCADGui
00035 import ArchCommands
00036 import os
00037 import sys
00038 import math
00039
00040 def getpointsOfUShapeRebar(FacePRM, r_cover, l_cover, b_cover, t_cover, orientation):
00041     """ getpointsOfUShapeRebar(FacePRM, RightCover, LeftCover, BottomCover, TopCover, Orientation):
00042     Return points of the UShape rebar in the form of array for sketch.
00043     It takes four different orientations input i.e. 'Bottom', 'Top', 'Left', 'Right'.
00044     """
00045     if orientation == "Bottom":
00046         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00047         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00048         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00049         y2 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00050         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00051         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00052         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00053         y4 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00054     elif orientation == "Top":
00055         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00056         y1 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00057         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00058         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00059         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00060         y3 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00061         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00062         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00063     elif orientation == "Left":
00064         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00065         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00066         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00067         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00068         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00069         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00070         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00071         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00072     elif orientation == "Right":
00073         x1 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00074         y1 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00075         x2 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00076         y2 = FacePRM[1][1] + FacePRM[0][1] / 2 - t_cover
00077         x3 = FacePRM[1][0] - FacePRM[0][0] / 2 + FacePRM[0][0] - r_cover
00078         y3 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00079         x4 = FacePRM[1][0] - FacePRM[0][0] / 2 + l_cover
00080         y4 = FacePRM[1][1] - FacePRM[0][1] / 2 + b_cover
00081     return [FreeCAD.Vector(x1, y1, 0), FreeCAD.Vector(x2, y2, 0),\
00082             FreeCAD.Vector(x3, y3, 0), FreeCAD.Vector(x4, y4, 0)]
00083
00084 class _UShapeRebarTaskPanel:
```

```
00085     def __init__(self, Rebar = None):
00086         self.CustomSpacing = None
00087         if not Rebar:
00088             selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00089             self.SelectedObj = selected_obj.Object
00090             self.FaceName = selected_obj.SubElementNames[0]
00091         else:
00092             self.FaceName = Rebar.Base.Support[0][1][0]
00093             self.SelectedObj = Rebar.Base.Support[0][0]
00094         self.form = FreeCADGui.PySideUic.loadUi(os.path.splitext(__file__)[0] + ".ui")
00095         self.form.setWindowTitle(QtGui.QApplication.translate("RebarAddon", "U-Shape Rebar", None))
00096         self.form.orientation.addItems(["Bottom", "Top", "Right", "Left"])
00097         self.form.amount_radio.clicked.connect(self.amount_radio_clicked)
00098         self.form.spacing_radio.clicked.connect(self.spacing_radio_clicked)
00099         self.form.customSpacing.clicked.connect(lambda: runRebarDistribution(self))
00100         self.form.removeCustomSpacing.clicked.connect(lambda:
      removeRebarDistribution(self))
00101         self.form.PickSelectedFace.clicked.connect(lambda: getSelectedFace(self))
00102         self.form.orientation.currentIndexChanged.connect(self.getOrientation)
00103         self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarBottom.svg"))
00104         self.form.toolButton.setIcon(self.form.toolButton.style().standardIcon(
      QtGui.QStyle.SP_DialogHelpButton))
00105         self.form.toolButton.clicked.connect(lambda: showPopUpImageDialog(os.path.split
      (os.path.abspath(__file__))[0] + "/icons/UShapeRebarDetailed.svg"))
00106         self.Rebar = Rebar
00107
00108     def getOrientation(self):
00109         orientation = self.form.orientation.currentText()
00110         if orientation == "Bottom":
00111             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarBottom.svg"))
00112         elif orientation == "Top":
00113             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarTop.svg"))
00114         elif orientation == "Right":
00115             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarRight.svg"))
00116         else:
00117             self.form.image.setPixmap(QtGui.QPixmap(os.path.split(os.path.abspath(__file__))[0] + "
      /icons/UShapeRebarLeft.svg"))
00118
00119     def getStandardButtons(self):
00120         return int(QtGui.QDialogButtonBox.Ok) | int(QtGui.QDialogButtonBox.Apply) | int(
      QtGui.QDialogButtonBox.Cancel)
00121
00122     def clicked(self, button):
00123         if button == int(QtGui.QDialogButtonBox.Apply):
00124             self.accept(button)
00125
00126     def accept(self, signal = None):
00127         f_cover = self.form.frontCover.text()
00128         f_cover = FreeCAD.Units.Quantity(f_cover).Value
00129         b_cover = self.form.bottomCover.text()
00130         b_cover = FreeCAD.Units.Quantity(b_cover).Value
00131         r_cover = self.form.r_sideCover.text()
00132         r_cover = FreeCAD.Units.Quantity(r_cover).Value
00133         l_cover = self.form.l_sideCover.text()
00134         l_cover = FreeCAD.Units.Quantity(l_cover).Value
00135         t_cover = self.form.topCover.text()
00136         t_cover = FreeCAD.Units.Quantity(t_cover).Value
00137         diameter = self.form.diameter.text()
00138         diameter = FreeCAD.Units.Quantity(diameter).Value
00139         rounding = self.form.rounding.value()
00140         orientation = self.form.orientation.currentText()
00141         amount_check = self.form.amount_radio.isChecked()
00142         spacing_check = self.form.spacing_radio.isChecked()
00143         if not self.Rebar:
00144             if amount_check:
00145                 amount = self.form.amount.value()
00146                 rebar = makeUShapeRebar(f_cover, b_cover, r_cover, l_cover, diameter,
      t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00147             elif spacing_check:
00148                 spacing = self.form.spacing.text()
00149                 spacing = FreeCAD.Units.Quantity(spacing).Value
00150                 rebar = makeUShapeRebar(f_cover, b_cover, r_cover, l_cover, diameter,
      t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
      FaceName)
00151         else:
00152             if amount_check:
00153                 amount = self.form.amount.value()
00154                 rebar = editUShapeRebar(self.Rebar, f_cover, b_cover, r_cover, l_cover,
       diameter, t_cover, rounding, True, amount, orientation, self.SelectedObj, self.
      FaceName)
00155             elif spacing_check:
00156                 spacing = self.form.spacing.text()
```

```
00157                 spacing = FreeCAD.Units.Quantity(spacing).Value
00158                 rebar = editUShapeRebar(self.Rebar, f_cover, b_cover, r_cover, l_cover,
 diameter, t_cover, rounding, False, spacing, orientation, self.SelectedObj, self.
FaceName)
00159         if self.CustomSpacing:
00160             rebar.CustomSpacing = self.CustomSpacing
00161             FreeCAD.ActiveDocument.recompute()
00162         self.Rebar = rebar
00163         if signal == int(QtGui.QDialogButtonBox.Apply):
00164             pass
00165         else:
00166             FreeCADGui.Control.closeDialog(self)
00167 
00168     def amount_radio_clicked(self):
00169         self.form.spacing.setEnabled(False)
00170         self.form.amount.setEnabled(True)
00171 
00172     def spacing_radio_clicked(self):
00173         self.form.amount.setEnabled(False)
00174         self.form.spacing.setEnabled(True)
00175 
00176 
00177 def makeUShapeRebar(f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding,
amount_spacing_check, amount_spacing_value, orientation = "Bottom", structure = None, facename = None):
00178     """ makeUShapeRebar(FrontCover, BottomCover, RightCover, LeftCover, Diameter, Topcover, Rounding,
 AmountSpacingCheck, AmountSpacingValue,
00179     Orientation, Structure, Facename): Adds the U-Shape reinforcement bar to the selected structural
 object.
00180     It takes four different types of orientations as input i.e 'Bottom', 'Top', 'Right', 'Left'.
00181     """
00182     if not structure and not facename:
00183         selected_obj = FreeCADGui.Selection.getSelectionEx()[0]
00184         structure = selected_obj.Object
00185         facename = selected_obj.SubElementNames[0]
00186     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00187     #StructurePRM = getTrueParametersOfStructure(structure)
00188     FacePRM = getParametersOfFace(structure, facename)
00189     if not FacePRM:
00190         FreeCAD.Console.PrintError("Cannot identified shape or from which base object sturctural element is
 derived\n")
00191         return
00192     # Get points of U-Shape rebar
00193     points = getpointsOfUShapeRebar(FacePRM, r_cover, l_cover, b_cover, t_cover,
orientation)
00194     import Part
00195     import Arch
00196     sketch = FreeCAD.activeDocument().addObject('Sketcher::SketchObject', 'Sketch')
00197     sketch.MapMode = "FlatFace"
00198     sketch.Support = [(structure, facename)]
00199     FreeCAD.ActiveDocument.recompute()
00200     sketch.addGeometry(Part.LineSegment(points[0], points[1]), False)
00201     sketch.addGeometry(Part.LineSegment(points[1], points[2]), False)
00202     import Sketcher
00203     sketch.addGeometry(Part.LineSegment(points[2], points[3]), False)
00204     if amount_spacing_check:
00205         rebar = Arch.makeRebar(structure, sketch, diameter, amount_spacing_value, f_cover)
00206         FreeCAD.ActiveDocument.recompute()
00207     else:
00208         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00209         rebar = Arch.makeRebar(structure, sketch, diameter, int((size - diameter) / amount_spacing_value),
f_cover)
00210     rebar.Rounding = rounding
00211     # Adds properties to the rebar object
00212     rebar.ViewObject.addProperty("App::PropertyString", "RebarShape", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Shape of rebar")).RebarShape = "UShapeRebar"
00213     rebar.ViewObject.setEditorMode("RebarShape", 2)
00214     rebar.addProperty("App::PropertyDistance", "FrontCover", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Front cover of rebar")).FrontCover = f_cover
00215     rebar.setEditorMode("FrontCover", 2)
00216     rebar.addProperty("App::PropertyDistance", "RightCover", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Right Side cover of rebar")).RightCover = r_cover
00217     rebar.setEditorMode("RightCover", 2)
00218     rebar.addProperty("App::PropertyDistance", "LeftCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
", "Left Side cover of rebar")).LeftCover = l_cover
00219     rebar.setEditorMode("LeftCover", 2)
00220     rebar.addProperty("App::PropertyDistance", "BottomCover", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Bottom cover of rebar")).BottomCover = b_cover
00221     rebar.setEditorMode("BottomCover", 2)
00222     rebar.addProperty("App::PropertyBool", "AmountCheck", "RebarDialog", QT_TRANSLATE_NOOP("App::Property",
 "Amount radio button is checked")).AmountCheck
00223     rebar.setEditorMode("AmountCheck", 2)
00224     rebar.addProperty("App::PropertyDistance", "TopCover", "RebarDialog", QT_TRANSLATE_NOOP("App::Property"
, "Top cover of rebar")).TopCover = t_cover
00225     rebar.setEditorMode("TopCover", 2)
00226     rebar.addProperty("App::PropertyDistance", "TrueSpacing", "RebarDialog", QT_TRANSLATE_NOOP("
App::Property", "Spacing between of rebars")).TrueSpacing = amount_spacing_value
00227     rebar.setEditorMode("TrueSpacing", 2)
```

```
00228     rebar.addProperty("App::PropertyString", "Orientation", "RebarDialog", QT_TRANSLATE_NOOP("App::Property
      ", "Shape of rebar")).Orientation = orientation
00229     rebar.setEditorMode("Orientation", 2)
00230     if amount_spacing_check:
00231         rebar.AmountCheck = True
00232     else:
00233         rebar.AmountCheck = False
00234         rebar.TrueSpacing = amount_spacing_value
00235     rebar.Label = "UShapeRebar"
00236     FreeCAD.ActiveDocument.recompute()
00237     return rebar
00238
00239 def editUShapeRebar(Rebar, f_cover, b_cover, r_cover, l_cover, diameter, t_cover, rounding,
      amount_spacing_check, amount_spacing_value, orientation, structure = None, facename = None):
00240     sketch = Rebar.Base
00241     if structure and facename:
00242         sketch.Support = [(structure, facename)]
00243     # Check if sketch support is empty.
00244     if not sketch.Support:
00245         showWarning("You have checked remove external geometry of base sketchs when needed.\nTo
       unchecked Edit->Preferences->Arch.")
00246         return
00247     # Assigned values
00248     facename = sketch.Support[0][1][0]
00249     structure = sketch.Support[0][0]
00250     face = structure.Shape.Faces[getFaceNumber(facename) - 1]
00251     #StructurePRM = getTrueParametersOfStructure(structure)
00252     # Get parameters of the face where sketch of rebar is drawn
00253     FacePRM = getParametersOfFace(structure, facename)
00254     # Get points of U-Shape rebar
00255     points = getpointsOfUShapeRebar(FacePRM, r_cover, l_cover, b_cover, t_cover,
      orientation)
00256     sketch.movePoint(0, 1, points[0], 0)
00257     FreeCAD.ActiveDocument.recompute()
00258     sketch.movePoint(0, 2, points[1], 0)
00259     FreeCAD.ActiveDocument.recompute()
00260     sketch.movePoint(1, 1, points[1], 0)
00261     FreeCAD.ActiveDocument.recompute()
00262     sketch.movePoint(1, 2, points[2], 0)
00263     FreeCAD.ActiveDocument.recompute()
00264     sketch.movePoint(2, 1, points[2], 0)
00265     FreeCAD.ActiveDocument.recompute()
00266     sketch.movePoint(2, 2, points[3], 0)
00267     FreeCAD.ActiveDocument.recompute()
00268     Rebar.OffsetStart = f_cover
00269     Rebar.OffsetEnd = f_cover
00270     if amount_spacing_check:
00271         Rebar.Amount = amount_spacing_value
00272         FreeCAD.ActiveDocument.recompute()
00273         Rebar.AmountCheck = True
00274     else:
00275         size = (ArchCommands.projectToVector(structure.Shape.copy(), face.normalAt(0, 0))).Length
00276         Rebar.Amount = int((size - diameter) / amount_spacing_value)
00277         FreeCAD.ActiveDocument.recompute()
00278         Rebar.AmountCheck = False
00279     Rebar.Diameter = diameter
00280     Rebar.FrontCover = f_cover
00281     Rebar.RightCover = r_cover
00282     Rebar.LeftCover = l_cover
00283     Rebar.BottomCover = b_cover
00284     Rebar.TopCover = t_cover
00285     Rebar.Rounding = rounding
00286     Rebar.TrueSpacing = amount_spacing_value
00287     Rebar.Orientation = orientation
00288     FreeCAD.ActiveDocument.recompute()
00289     return Rebar
00290
00291 def editDialog(vobj):
00292     FreeCADGui.Control.closeDialog()
00293     obj = _UShapeRebarTaskPanel(vobj.Object)
00294     obj.form.frontCover.setText(str(vobj.Object.FrontCover))
00295     obj.form.r_sideCover.setText(str(vobj.Object.RightCover))
00296     obj.form.l_sideCover.setText(str(vobj.Object.LeftCover))
00297     obj.form.bottomCover.setText(str(vobj.Object.BottomCover))
00298     obj.form.diameter.setText(str(vobj.Object.Diameter))
00299     obj.form.topCover.setText(str(vobj.Object.TopCover))
00300     obj.form.rounding.setValue(vobj.Object.Rounding)
00301     obj.form.orientation.setCurrentIndex(obj.form.orientation.findText(str(vobj.Object.Orientation)))
00302     if vobj.Object.AmountCheck:
00303         obj.form.amount.setValue(vobj.Object.Amount)
00304     else:
00305         obj.form.amount_radio.setChecked(False)
00306         obj.form.spacing_radio.setChecked(True)
00307         obj.form.amount.setDisabled(True)
00308         obj.form.spacing.setEnabled(True)
00309         obj.form.spacing.setText(str(vobj.Object.TrueSpacing))
00310     #obj.form.PickSelectedFace.setVisible(False)
```

```
00311     FreeCADGui.Control.showDialog(obj)
00312
00313 def CommandUShapeRebar():
00314     selected_obj = check_selected_face()
00315     if selected_obj:
00316         FreeCADGui.Control.showDialog(_UShapeRebarTaskPanel())
```