

Autodesk Inventor®  
Tutorial Exercise

# SAE Car Wheel Center Exercise. Autodesk Inventor® Parametric Study



## **OBJECTIVE:**

To create simulations of various wheel center designs that focus on reducing the mass of the current design. The new designs include reducing the thickness of the legs, modifying the fillet radius at the intersection of the legs and the hub, and modifying the radius of the fillet on the back face near the top of each leg.

## Contents

OBJECTIVE .....	3
DESCRIPTION .....	3
DATASET .....	4
DESIGN CRITERIA.....	4
EXERCISE.....	5
DESIGN CRITERIA.....	5
Create a New Simulation .....	5
Review the Material .....	5
Add Constraints.....	6
Add Loads .....	6
Run a Simulation .....	7
Change the Display Settings .....	8
CONCLUSION: .....	8
Create a New Parameter .....	8
Create a Second Simulation .....	10
Create a Parametric Study .....	10
View the Configurations.....	11
Define Design Constraints .....	11
Visualize All Results .....	13
Create and View XY Plots .....	13
Promote the Improved Design .....	14
CONCLUSION: .....	14

**OBJECTIVE**

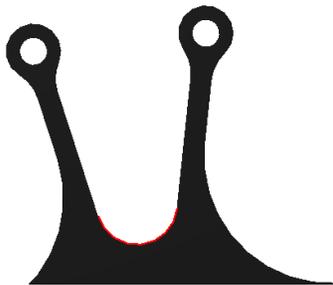
To create simulations of various wheel center designs that focus on reducing the mass of the current design. The new designs include reducing the depth of the radial arms and modifying the fillet radii at the intersection of the legs and the hub.

**DESCRIPTION**

This exercise is focused on reducing the mass of the part, while maintaining the design specifications.

Using Autodesk Inventor, a parametric study will be run using three parameters. They are:

1. The smaller radius between the radial arms connecting to the wheel rim to the axle.



The larger radius between the radial arms connecting to the wheel rim to the axle.



2. The depth of the radial arms..



**DATASET**



*Wheel.iam*

The part to be analyzed is *Wheel Center.ipt*.



**DESIGN CRITERIA**

<b>YIELD STRENGTH (MPa)</b>	<b>DEFLECTION (mm)</b>	<b>SAFETY FACTOR</b>
276 (AL 6061)	1.25 mm	2.0

## EXERCISE

In this exercise, you review a design for the wheel center in the Formula SAE race car built by the Oklahoma University Sooner Racing Team. The objective is to reduce the mass of the wheel center which is currently 1.143 kg.

Note that your results may vary slightly from the figures quoted here.

## DESIGN CRITERIA

The following table lists the maximum allowable stress and displacement values and the minimum allowable safety factor for this exercise.

YIELD STRENGTH	DISPLACEMENT	SAFETY FACTOR
276 MPa	1.25 mm	2.0

## Create a New Simulation

In this section of the exercise, you open the current version of the wheel assembly.

1. Make *Wheel Center.ipt* the active project.
2. Open *Wheel.iam*.

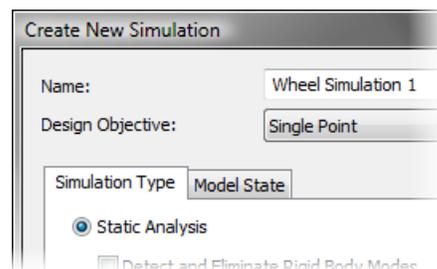


3. In the browser, right-click Wheel Center:1. Click Open.

4. If required, on the ViewCube, click Home.



5. On the Environments tab, Begin panel, click Stress Analysis.
6. On the Manage panel, click Create Simulation.
7. For Name, enter **Wheel Simulation 1**.



8. Click OK.

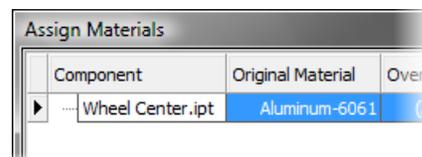
## Review the Material

In this section of the exercise, you review the currently assigned material.

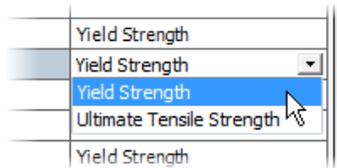
1. On the Material panel, click Assign.



2. Review the Assign Materials dialog box. For this simulation the Aluminum 6061 is correct.



- Under the Safety Factor column, click on the down-arrow beside Yield Strength for the part.



**Note:** The Safety Factor is calculated on the Yield Strength or Ultimate Tensile Strength of the material. For example, if the Yield Strength of the material is 276 MPa and the von-Mises equivalent stress is 138 MPa the safety factor is 2.0 ( $276/138 = 2.0$ ).

- Click Cancel to close the Assign Materials dialog box.

### Add Constraints

In this section of the exercise, you add pin constraints on the bolt holes and a frictionless constraint to the wheel center.

- On the Constraints panel, click Pin.



- Select the face of a hole as shown.



- Click Apply.
- Repeat this work flow for the other 9 holes.
- Click Cancel.
- In the browser, check to ensure that there are 10 pin constraints

- On the Constraints panel, click Frictionless.



- Select the face of the hole in the wheel center as shown.



- Click OK.

### Add Loads

In this section of the exercise, you add loads to the wheel.

- On the Loads panel, click Moment.



- Select the face of the hole in the wheel center as shown.



- In the Moment dialog box, for Magnitude, enter **130000 N mm**.

**Note:** This value is typical of the moment applied to this wheel under acceleration.

- Click OK. The moment is added and is displayed as a glyph on the face of the part.

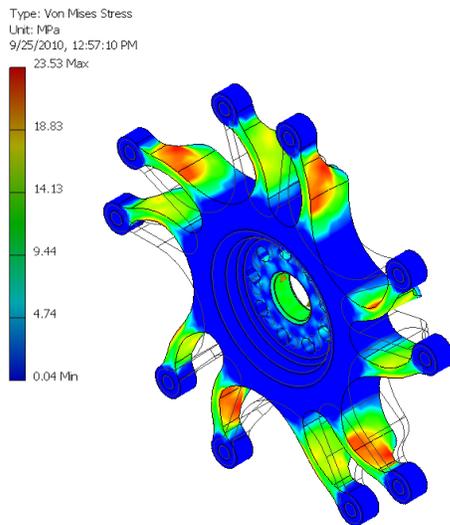
## Run a Simulation

In this section of the exercise, you run a simulation.

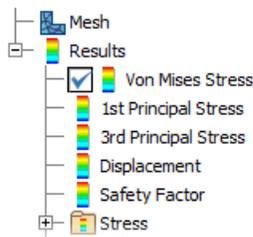
1. On the Solve panel, click Simulate.



2. Click Run.

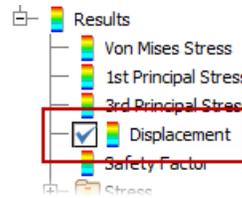


3. In the browser, review the Results folder. By default, Von Mises Stress is active.



4. Review the Maximum and Minimum values. The values are 23.54 MPa and 0 MPa respectively. Comparing these values to the supplied design criteria shows that the design is compliant.

5. In the browser, double-click Displacement.



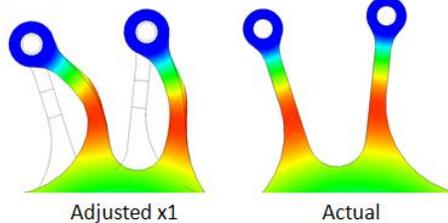
6. Review the Maximum and Minimum values. The values are 0.06325 mm and 0 mm respectively. Comparing these values to the supplied design criteria shows that the design is compliant.
7. In the browser, double-click Safety Factor.
8. Review the Minimum value. The value is 11.68. Comparing this value to the supplied design criteria shows that the design is compliant.
9. On the Exit menu, click Finish Stress Analysis.
10. In the browser, right-click Wheel Center.ipt. Click iProperties.
11. On the Physical tab, click Update. The mass of the brake pedal is 1.143 kg.
12. Click Close.
13. On the Environments tab, Begin panel, click Stress Analysis.
14. In the browser, double-click Von Mises Stress.
15. On the Result panel, click Animate.



16. On the Animate Results dialog box, click Play.
17. Click OK.
18. Repeat the animation for Displacement and Safety Factor.
19. Close the Animate Results dialog box.

## Change the Display Settings

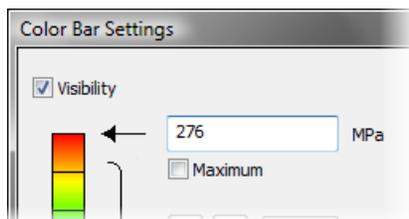
The displacement, shown by the bending radial arms, is exaggerated to make it easier to visualize. The actual displacement is very small as shown.



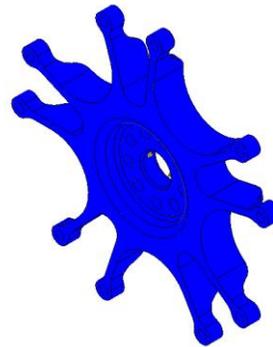
1. In the browser, double-click Von Mises Stress.
2. On the Display panel, click Color Bar.



3. In the Color Bar Settings dialog box, select the Maximum check box to remove the check mark.
4. For Maximum, enter the yield strength for the material, **276 MPa**.



5. Click Apply. The color of the wheel center changes and there is no red color visible. A review of the results with these display settings makes it clear that the design is well within the design criteria.



6. Select the Maximum check box
7. Click OK.

## CONCLUSION:

The design results indicate that modifications can be made to reduce the mass of the wheel center. These modifications include reducing the radius at the intersection of the arms and the hub and reducing the depth of the arms.

These values can be changed individually and the simulations run again. A more effective work flow is to create a parametric study. This allows you to nominate parameters for evaluation, define the parameter range, specify the design constraints, and analyze the results of each parameter.

## Create a New Parameter

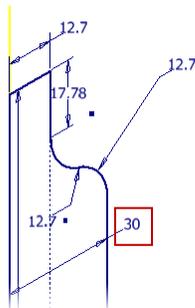
Parameters will allow us to change the geometry of the part automatically and easily analyze multiple design options. In this section of the exercise, you create a new parameter for the depth of the wheel center arm.

1. On the Exit menu, click Finish Stress Analysis.

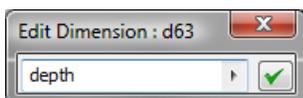
2. On the Manage tab, Parameters panel, click Parameters.



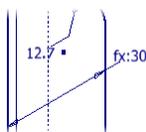
3. In the Parameters dialog box, click Add Numeric.
4. For User Parameter, enter **depth**.
5. In the Equation column, enter **30 mm**. Press ENTER.
6. Click Done.
7. In the browser, right-click Revolution1. Click Edit Sketch.
8. In the graphics window, double-click the 30 dimension.



9. In the Edit Dimension box, enter **depth**.



10. Click the check mark. The dimension display changes and the *fx* prefix indicates that the value is a function.

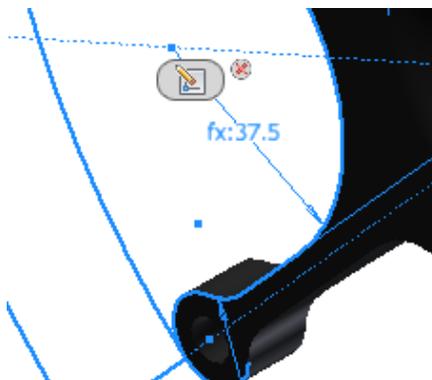
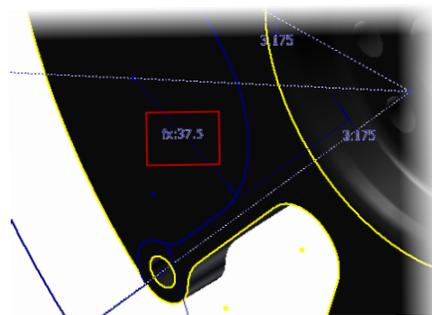
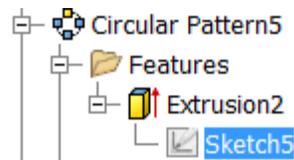


11. On the Exit panel, click Finish Sketch.

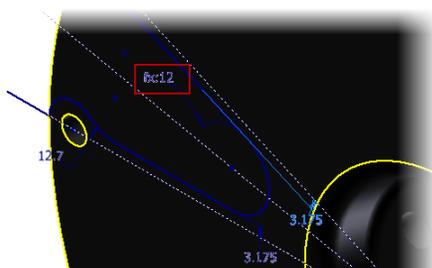
**Note:** 2 other user parameters are defined and already applied to the sketch.

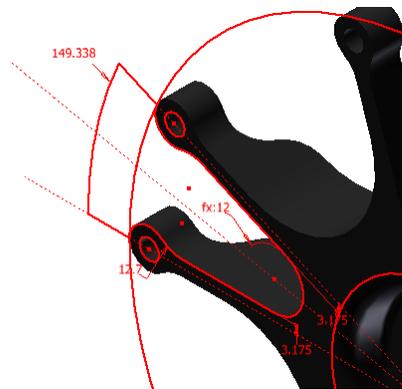
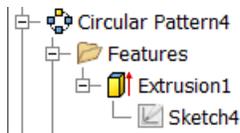
- **largerad** defines the larger radius at the bottom of the arm. The location can be viewed in the Browser

window under Sketch5 on Circular Pattern5. The current value is 37.5 mm.



- **smallrad** defines the smaller radius at the bottom of the arm. The location can be viewed in the Browser window under Sketch4 on Circular Pattern4. The current value is 12 mm.

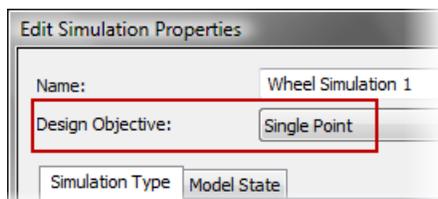




Because these radii parameter are defined to be tangent to the radial arms, making their values smaller makes the arms longer by removing material near the axle.

### Create a Second Simulation

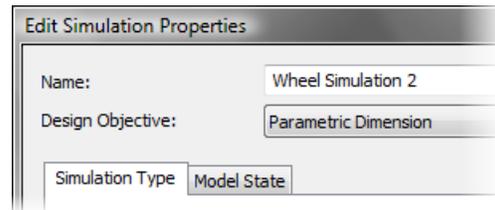
The first simulation was created with a Design Objective of Single Point.



To create a parametric study, a second simulation will be created with a Parametric Dimension objective.

1. On the Environments tab, Begin panel, click Stress Analysis.
2. In the browser, right-click Wheel Simulation 1. Click Copy Simulation.
3. Right-click Wheel Simulation 1:1. Click Edit Simulation Properties.
4. For Name, edit the name to **Wheel Simulation 2**.

5. From the Design Objective list, select Parametric Dimension.

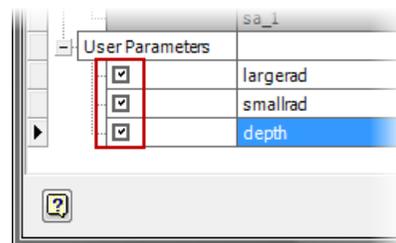


6. Click OK.

### Create a Parametric Study

To create the parametric study, the existing parameters are accessed and included in the simulation.

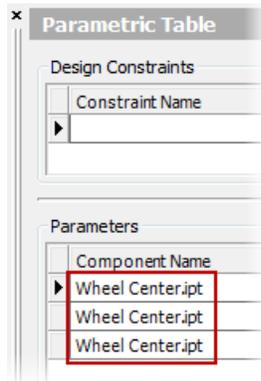
1. In the browser, right-click Wheel Center.ipt. Click Show Parameters.
2. In the Select Parameters dialog box, scroll down to the User Parameters section.
3. Under User Parameters, select the largerad, smallrad, and depth check boxes.



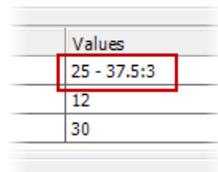
4. Click OK.
5. On the Manage tab, click Parametric Table.



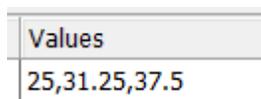
- If required, expand the table to display the 3 components under Parameters.



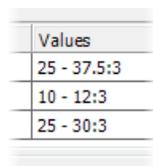
- In the largerad row, under the Values column, enter **25-37.5**. Press ENTER.



**Note:** The range is listed and the number 3 indicates three values will be used in the parametric study, 25, 31.25, and 37.5. If preferred, you can also enter these values manually, separated by commas.



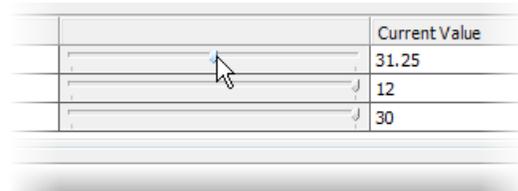
- Repeat the work flow for smallrad and depth.
  - For smallrad, enter **10-12**.
  - For depth, enter **25-30**.



### View the Configurations

In this section of the exercise, you generate all the configurations and view each configuration.

- On the ViewCube, click Home.
- Under Component Name, right-click the first Wheel Center.ipt. Click Generate All Configurations. This prepares all geometric configurations, as defined by the parameter values, for the analysis.
- Drag the Current View slider for a configuration to view the modified geometry. In this example, largerad is modified to 31.25.

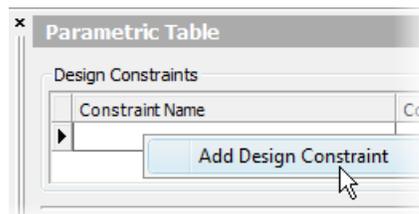


- Return all values to the maximum value by dragging the slider to the right.

### Define Design Constraints

In this section of the exercise, you enable access to the design criteria by adding design constraints. These will allow you to quickly see which configurations meet your design constraints, and which don't.

- Under Design Constraints, right-click in the cell under Constraint Name. Click Add Design Constraint.



- Under Results Component, select Von Mises Stress.
- Click OK.
- In the Parametric Table, under Constraint Type, select Upper Limit from the list.
- In the Limits cell, enter **276**.

- In the Safety Factor cell, enter **2**. This automatically calculates a constraint of 138 MPa for Von Mises stress (276 MPa/ 2).

Parametric Table			
Design Constraints			
Constraint Name	Constraint Type	Limit	Safety Factor
Max Von Mises Stress	Upper limit	276	2

- The results are displayed with the Current Value for each user parameter set to maximum.

Result Value
23.6644
0.0632313
1.14236

- Repeat the work flow to add 2 more design constraints with the values as shown.

Parametric Table			
Design Constraints			
Constraint Name	Constraint Type	Limit	Safety Factor
Max Von Mises Stress	Upper limit	276	2
Max Displacement	Upper limit	1.25	1
Mass	View the value		

- Drag the Current Value slider to modify the 3 parameters and review the results.
- Drag the slider to the left (minimum) for all values.
- Review the Result Value column (your results may vary slightly). Comparing these values to the supplied design criteria shows that the design is compliant. This is clearly shown with the green circle in the results column.

**Note:** You may have to drag the lower edge of the Design Constraints table down to view Mass.

- On the Solve panel, click Simulate.



**Note:** The default option is Smart Set of Configurations This solves the base configuration changes each parameter with respect to the base, and then interpolates the other results. In this simulation, 7 configurations will be run.

- Click Run. Depending on the speed of your computer, this may take a few minutes.

Result Value
31.3086
0.118145
0.656476

Current Value
25
10
25

**Note:** If a design configuration does not meet the design criteria – for example, if the Von Mises Stress safety factor requirement were much higher or a material was used with a lower yield strength – a red square would clearly call-out the non-compliant design.

Safety Factor	Result Value	Unit
10	31.7704	MPa
1	0.118237	mm
	0.656476	kg

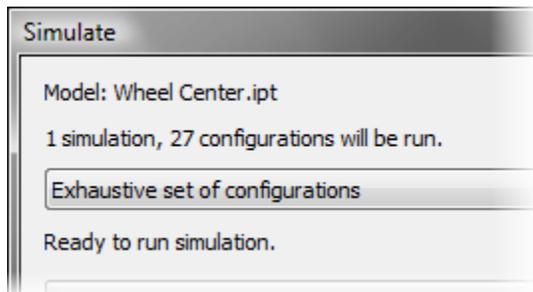
### Visualize All Results

You will note that the Color display does not update for all configurations. Those results were not calculated individually, they were interpolated because Smart Set of Configurations was chosen. To better view all results, you must run an Exhaustive Set of Configurations.

1. On the Solve panel, click Simulate.

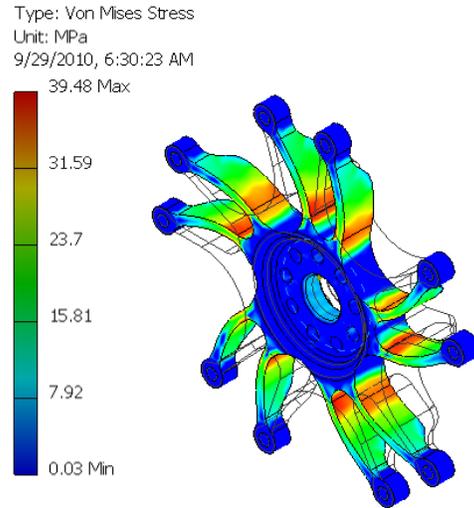


2. Select Exhaustive set of configurations from the list. 27 simulations will be run.



3. Click Run. This may take a few minutes to complete.

**Note:** The results are displayed for all configurations and the results have been updated for the configuration with the minimum values for the parameters (i.e. Von Mises stress is now 39.5 instead of 31.3). This is because Inventor is now calculating those results instead of interpolating them. The mass of the part is now 0.656 kilograms – a reduction 43% from the original 1.143kg.



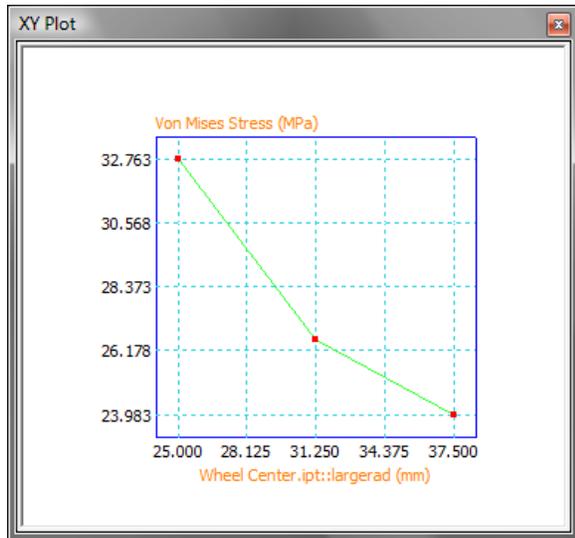
Result Value
39.484
0.157026
0.655981

### Create and View XY Plots

With parametric studies, you can view the results as a two-dimensional XY plot. In this section of the exercise, you view the XY plot for Von Mises Stress at the minimum value for all parameters.

1. In the browser, Von Mises Stress should be active. Right-click anywhere in the Parameters table and select XY Plot. This graph will plot whatever results are currently being displayed.

- Review the plot. These plots can be used to review a set of configurations and included in a report.

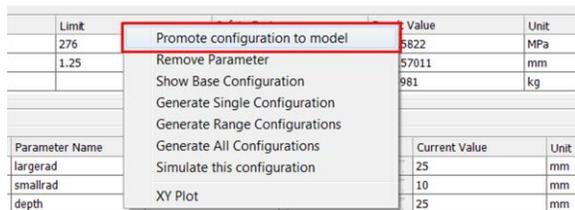


- Close the XY Plot window.

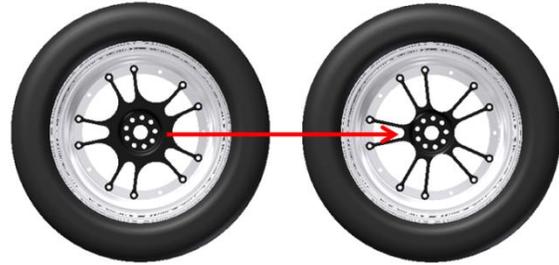
### Promote the Improved Design

Once you determine that a configuration satisfies your design needs, you are able to promote that configuration back to the model as a CAD edit.

- Ensure that all parameter value sliders are set to the minimum value.
- Right click on the Parameters table and select Promote Configuration to Model.



- Make Wheel.iam the active file. The part has been changed.



### CONCLUSION:

The objective was to reduce the mass of the wheel center and that has been achieved. The current design is 43% lighter than the original design. What more can be done to reduce the mass? In this assembly the overall dimensions are controlled by 2 design criteria.

- The radius of the arms.
- The bolt hole diameter around the drive shaft.

The stress, displacement, and safety factors are all well within the design criteria, but the design criteria prevent more meaningful reductions in the mass of the wheel center. It is now just 0.656 kg and further modifications would not represent a significant reduction for the cost and effort required.

Autodesk and Autodesk Inventor are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.