Design of an efficient space constrained diffuser for supercritical CO₂ turbines

Joshua A. Keep 1, Adam J. Head 2 and Ingo H. Jahn 1
The University of Queensland, Brisbane, Australia 1, Delft University of Technology, Delft, The Netherlands 2
E-mail: j.keep@uq.edu.au, a.j.head@tudelft.nl, I.jahn@uq.edu.au

Abstract. Radial inflow turbines are an arguably relevant architecture for energy extraction from ORC and supercritical CO₂ power cycles. At small scale, design constraints can prescribe high exit velocities for such turbines, which lead to high kinetic energy in the turbine exhaust stream. The inclusion of a suitable diffuser in a radial turbine system allows some exhaust kinetic energy to be recovered as static pressure, thereby ensuring efficient operation of the overall turbine system. In supercritical CO₂ Brayton cycles, the high turbine inlet pressure can lead to a sealing challenge if the rotor is supported from the rotor rear side, due to the seal operating at rotor inlet pressure. An alternative to this is a cantilevered layout with the rotor exit facing the bearing system. While such a layout is attractive for the sealing system, it limits the axial space claim of any diffuser. Previous studies into conical diffuser geometries for supercritical CO₂ have shown that in order to achieve optimal static pressure recovery, longer geometries of a shallower cone angle are necessitated when compared to air. A diffuser with a combined annular-radial arrangement is investigated as a means to package the aforementioned geometric characteristics into a limited space claim for a 100kW radial inflow turbine. Simulation results show that a diffuser of this design can attain static pressure rise coefficients greater than 0.88. This confirms that annular-radial diffusers are a viable design solution for supercritical CO₂ radial inflow turbines, thus enabling an alternative cantilevered rotor layout.

1. Introduction
The inclusion of a suitable diffuser in a radial turbine system is essential for the efficient operation of the turbine system in order to recover exhaust kinetic energy as static pressure. Several studies into supercritical CO₂ power cycles show that cycle efficiency is maximised for turbine inlet pressures between 20 and 30 MPa and turbine exhaust pressures immediately above the critical pressure for CO₂ in the vicinity of 8 Mpa [1, 2]. For these pressure ratios, it is feasible to employ a single stage radial inflow turbine as the work extraction method. While this may be feasible from an aerodynamic standpoint, this can lead to a sealing challenge if the rotor is supported from the rear in a cantilevered arrangement as shown in Fig. 1a. If the rotor layout is inverted such that the rotor exit is facing the bearing system as in Fig. 1b, the conditions for any sealing system are improved with a lower static pressure and temperature.

While such a layout is favourable for a shaft sealing system, it axially limits the extent of any diffuser due to shaft bending and constrains its entrance to an annular geometry. In effect, the diffuser must adopt either a cropped annular geometry, or annular-radial geometry.

Within the power conversion and turbomachinery group at the University of Queensland a prototype design of a 100kW radial inflow supercritical CO₂ turbine has been completed.
including a shaft layout as described in Fig. 1b. The design of the diffuser to be incorporated in this turbine design will be the focus of this paper. The diffuser geometric constraints are listed in Table 1.

This paper presents a theoretical design of a swirl robust combined annular-radial diffuser. A parameter study on the influence of diffuser geometric features on aerodynamic performance for fixed flow conditions is detailed.

2. Theory and background
The aim of a diffuser is to convert kinetic energy into a pressure rise. Diffuser performance is typically described by the pressure rise coefficient, $C_p$, defined in Eqn. 1.

$$C_p = \frac{P_2 - P_1}{P_{1, \text{total}} - P_1}$$  \hspace{1cm} (1)

Where subscripts 1 and 2 refer to inlet and outlet respectively. For a diffuser of an arbitrary annular geometry as described in Fig. 2, the ideal pressure rise coefficient is defined according to Eqn. 2 [3].

$$C_{p,i} = 1 - \left( \frac{r_1}{r_2} \right)^2 \frac{\tan^2 \alpha_1 + \left( \frac{h_i^2}{r_2^4} \right)}{\tan^2 \alpha_1 + 1}$$  \hspace{1cm} (2)

Where $\alpha_1$ is the swirl angle. For a diffuser without swirl, this reduces to Eqn 3.

$$C_{p,i} = 1 - \frac{1}{Ar^2}$$  \hspace{1cm} (3)
Figure 2: A diffuser with arbitrary annular geometry.

Figure 3: Sectional view of proposed geometry for analysis. N.b. hub centre body curve approximated as Bezier curve.

Table 2: Aerodynamic parameters of influence.

| First order                                      | Second order                                      |
|--------------------------------------------------|---------------------------------------------------|
| Inlet Mach number \((M_1)\)                      | Turbulence length scale and intensity             |
| Inlet Reynolds Number \((Re_1)\)                 | Inlet mass flow distribution                      |
| Inlet total temperature and pressure \((T_1, total, P_1, total)\) |                                                   |
| Inlet aerodynamic blockage                       |                                                   |
| inlet swirl angle \((\alpha_1)\)                 |                                                   |

It is recognised by several authors that aerodynamic parameters have the most significant impact on diffuser performance and hence are classified as first order parameters [3, 4, 5]. These parameters are listed in Table 2 along with second order parameters.

Whilst these flow parameters are significant to the performance of any diffuser, the impact of their variation on performance is not explicitly considered in the current study, instead they are considered fixed and to be representative of nominal turbine outlet conditions.

Monje et.al. [4] completed a CFD study of conical diffuser geometries with both \(CO_2\) and air as the working fluids. The authors replicated some of the geometries and earlier experimental studies of Dolan and Runstadler [6] with air as the working fluid. This study indicated that the frictional effects for supercritical \(CO_2\) were less pronounced than for air, leading to optimal \(CO_2\) geometries possessing a narrower cone angle and longer passage length. Considering this, a combined annular-radial diffuser geometry is investigated (see Fig. 3) as a means to package a large mean length in a short axial space with annular inflow from the turbine. A key study into annular diffusers was that of Sovran and Klomp in the 1960’s [5] which identified annular diffusers as industrially significant for turbomachinery applications and considered performance impact of non-curved geometry variations.

Diffuser inlet swirl is anticipated in operation of the turbine due to required off-design operation and uncertainty in the rotor outlet velocity direction at the on-design condition and also during off-design operation. Hence any diffuser considered for this application should be swirl robust. From observing Eqn. 2, the influence of swirl on pressure recovery in an annular...
diffuser can be reduced by limiting change in mean radius \((r)\) or passage width \((h)\) over the diffuser. Considering this, there are two simple cases that are swirl robust. Firstly, purely axial flow \((r=\text{constant})\), which dictates no change in passage area, and hence limited diffusion, as diffusion is primarily area governed [3]. Secondly, purely radial flow \((h=\text{constant})\), where passage area increases with mean radius, leading to diffusion.

In considering the transition from an annular to a radial geometry, limited pressure recovery is possible due to the risk of flow separation [3]. A key study into a combined axial and radial diffuser by Moller [7] presented a design method for a bend with no change in passage area along the flow path.

Previous studies on axial-radial and radial-radial diffusers gave strong consideration to the impact of wall contouring and wall spacing on static pressure rise with respect to swirl. An experimental study into wall taper in a radial-radial diffuser by Yingkang and Sjolander [8] showed that parallel walled diffusers produced the most consistent pressure recovery coefficient with varying swirl angle. In considering the passage width of the parallel radial section of an axial-radial diffuser, Moller [7] tested geometries with non-dimensional width \(h/d\) between 0.1 and 0.2 with optimal determined analytically as 0.143. In a separate experimental study presented by Japikse [3], De Kranski and Sarpal determined the optimal \(h/d\) as 0.15.

3. Nominal design
The nominal diffuser geometry was created based on the geometric characteristics reviewed in the previous section for swirl robustness and pressure rise, and size constrained by factors listed in Table 1.

Considering the impact of swirl on ideal performance, the annular inlet section of diffuser under consideration is of fixed mean radius with no passage area change. In the transition from annular to radial a constant passage area is maintained. For the radial-radial section, walls remain parallel with spacing scaled from axial studies to an annular geometry. Graphically, this geometry is presented in Fig. 3.

The geometric relations for determining spacing \(h\) in Fig. 3 presented by Moller [7] are based on the diameter of a cylindrical inlet. For an annular inlet, these are scaled based on equivalent diameter \((d_{eq})\) defined in Eqn. 4 as opposed to the conventional definition of hydraulic diameter \((d_h)\) for an annular passage.

\[
d_{eq} = 2 \sqrt{r_s^2 - r_h^2} \quad (4)
\]

The passage width of the radial-radial section of the diffuser can be scaled to the equivalent inlet diameter according to Eqn. 5

\[
\frac{h}{d_{eq}} = a_1 \quad (5)
\]

For the transition from annular to radial, the shroud curve can be prescribed as a circular section with radius, \(r\). For bend inlet and outlet passage areas to be equivalent, \(r\) is mathematically described by Eqn. 6.

\[
r = \frac{d_{eq}}{8a} - r_s \quad (6)
\]

To define the hub curve, a similar method to Moller [7] was employed, where passage areas normal to the flow were fixed and passage width was numerically evaluated in order to prescribe coordinates of the hub centre body curve. For practical purposes it is sufficient to approximate this curve by a Bezier curve defined using 4 control points.

In considering the length, \(L\), of the annular inlet, it should be noted that the primary influence is that upon boundary layer growth and its impact on aerodynamic blockage [4]. For the nominal case, a larger value of \(L\) within the geometric constraints was chosen.
To define the maximum radial extent of the diffuser, relations presented in Moller were adapted to an annular geometry. The ideal radial extent of an axial-radial diffuser is defined according to Eqn. 7.

\[ r_o = 2.471 \frac{r_{in} Re_d^{\frac{1}{2}}}{Re_d} \]  

For the nominal design under consideration, diffuser inlet flow conditions are approximated as the rotor outlet conditions from a mean-line design. Values are listed in Table 3. Considering inlet conditions, this would lead to a prohibitively large outer radius. This correlates with the work of Monje [4] where it was noted that supercritical \( CO_2 \) diffuser geometries posses the optimal pressure recovery for area ratios in the region of 20. For analysis presented in this paper, outer radius is constrained to 75 mm, see Table 1, leading to a cropped diffuser. A larger area ratio could be mapped to the geometry through increasing \( h \), however this would dictate a large increase in passage area ratio on the bend, likely resulting in separation and consequently poor performance. Geometric parameters for the nominal diffuser design are presented in Table 4.

| Table 3: Nominal diffuser inlet conditions. |
|--------------------------------------------|
| Parameter | Value |
| \( \dot{m} \) | 1.0 kg/s |
| \( V_{1,n} \) | 50 m/s |
| \( \rho_1 \) | 65 kg/m\(^3\) |
| \( \alpha_1 \) | 0 |
| \( \mu_1 \) | 3.2\( \times 10^{-5} \) Pa.s |
| \( Re_d \) | 9.6\( \times 10^6 \) |
| \( T \) | 750 K |

| Table 4: Geometric parameters of nominal diffuser. |
|---------------------------------------------|
| Name | Symbol | Nominal value (mm) |
| Shroud radius | \( r_s \) | 12.3 |
| Hub radius | \( r_h \) | 7.6 |
| Inlet axial length | \( L \) | 15.0 |
| Transition radius | \( r \) | 4.6 |
| Outlet radius | \( r_o \) | 75.0 |
| Radial passage scaling factor | \( a_1 \) | 0.143 |
| Radial passage width | \( h \) | 2.8 |

4. Modelling
The diffuser was modelled as a quarter circumferential section of a 3D geometry. The ANSYS CFX 16.1 solver was used for the high-fidelity calculations [9]. The three-dimensional Reynolds Averaged Navier Stokes (RANS) equations were closed with the Shear Stress Transport \( k-\omega \) turbulence model.

The required thermodynamic and transport properties for the vapour phase are incorporated into the CFD solver through lookup tables. These properties were retrieved through an in-house software interface [10]. The generated Real Gas Property (RGP) tables allow for accurate property estimations which are bilinearly interpolated by the solver. The thermophysical library
used was RefProp, which implements NIST’s formulation of the Span and Wagner Equation of State (EoS) model [11]. The tables were systematically refined until a sufficient accuracy was obtained in the solution.

Boundary conditions for simulations are specified in Table 5. The turbulence intensity was set as 10% based on work by Monje [4]. In prior turbine simulations it was noted that turbine exit mass flow distribution possessed a strong skewness towards the shroud line. This was incorporated in the diffuser inlet boundary conditions in order to more physically represent the operating environment. The two dimensional distribution of inlet mass flow rate boundary condition is prescribed according to Fig. 4.

| Surface           | Type       | Value                                    |
|-------------------|------------|------------------------------------------|
| Inlet             | Mass flow  | Radial variation described Fig. 4, total 0.25 kg/s |
| Outlet            | Static pressure | 8.9 MPa                               |
| Shroud and hub    | No-slip wall  | N/A                                     |
| Edges             | Rotational periodicity | N/A                                  |

4.1. Mesh

A structured 3 dimensional grid of approximately $1.0 \times 10^6$ cells was generated using Pointwise. A suitable first layer height was determined as $1.0 \times 10^{-6}$ mm based on similarity of wall shear stress profiles to coarser and finer first layer heights, and maintaining $y+$ values in a range between 2 and 10.

In order to minimise skewness and area ratio, the nominal mesh was specified 140 nodes in the stream-wise direction,150 between the flow bounding surfaces, and 50 in the circumferential direction giving a total of $1.02 \times 10^6$ cells. Based on the nominal mesh a grid convergence study was conducted using a refinement ratio of 1.2. For this study both a fine and coarse mesh relative to the nominal were generated. Static entropy and total enthalpy were monitored as mass flow rate averages at inlet and exit of the geometry. Values for these are shown in Table 6. The nominal mesh sizing for this study was used for all geometry variants with the ability to coarsen if needed.
Table 6: Grid study of static entropy and total enthalpy, refinement ratio 1.2.

| Mesh   | $s_1$ (J/kg.K) | $s_2$ (J/kg.K) | $h_{1,\text{total}}$ (J/kg) | $h_{2,\text{total}}$ (J/kg) |
|--------|----------------|----------------|-----------------------------|-----------------------------|
| Coarse | 2797.63        | 2797.72        | 957635                      | 957635                      |
| Nominal| 2797.64        | 2797.72        | 957635                      | 957635                      |
| Fine   | 2797.64        | 2797.72        | 957635                      | 957635                      |

5. Results and discussion

5.1. Performance of the nominal diffuser

Pressure rise coefficient, $C_p$ was calculated according to Eqn. 1 as 0.89. In order to visualise sources of pressure losses and possible separation within the diffuser, pressure contours were plotted for a slice of the geometry in Fig. 5a. Further to this, pressure along shroud, hub and meridional (mean of shroud and hub) lines were also plotted in Fig. 5b. Meridional total pressure was also plotted. The meridional pressure lines give insight into the progression of inviscid flow within the diffuser.

![Pressure contour](image1.png)

(a) Pressure contour (MPa).

![Pressure along lines](image2.png)

(b) Pressure along hub, shroud and meridional lines.

Figure 5: Pressure distribution for nominal geometry.

From Fig. 5 it can be seen that there are high and low pressure regions on hub and shroud surfaces respectively, however there is no separation or re-circulation. There is no significant rise in meridional static pressure prior to the radial-radial section which matches design intent.

5.2. Effects of diffuser geometry

In order to understand the impact of geometry changes on performance and flow, a parameter study was conducted on the geometry whilst retaining boundary conditions and mesh density. Details of the geometries are summarised in Table 7.

Results of $C_p$ for each geometry are presented in the last column of Table 7. Pressure contours are plotted for a slice of the geometry in Figures 6a - 8a. Static pressure for shroud, hub and meridional lines as well as meridional total pressure are plotted in Figures 6b - 8b.
Figure 6: Pressure distribution for geometry variation 1.

(a) Pressure contour (MPa)  
(b) Pressure along hub, shroud and meridional lines.

c) Vector plot of recirculation region.

Figure 7: Pressure distribution for geometry variation 2.

(a) Pressure contour (MPa).  
(b) Pressure along hub, shroud and meridional lines.

Figure 8: Pressure distribution for geometry variation 3.

(a) Pressure contour (MPa).  
(b) Pressure along hub, shroud and meridional lines.
Table 7: Details of geometries for parameter study.

| I.D. | description                  | a₁  | r₀   | L    | hub centre body | Cₚ  |
|------|------------------------------|-----|------|------|----------------|-----|
| Nominal | nominal                      | 0.143 | 75mm   | 15mm | Y              | 0.89 |
| 1    | no hub centre body           | 0.143 | 75mm   | 15mm | N              | 0.88 |
| 2    | short inlet                  | 0.143 | 75mm   | 5mm  | Y              | 0.89 |
| 3    | narrow radial-radial         | 0.1  | 75mm   | 15mm | Y              | 0.82 |

5.3. Discussion

From observing Figures 5a-8a, all geometries show the majority of the static pressure rise in the radial-radial section, confirming design intent. This trend is also illustrated in meridional total pressure plots where decreases are present in all geometries towards the end of the radial-radial section. Flow separation is only observed in geometry variation 1, where a re-circulation region is present in place of the hub centre body. This re-circulation region is illustrated in Figure 6c. The nominal diffuser displays the highest pressure recovery, however removal of the hub centre body and shortening the inlet have only a small impact on the bulk pressure recovery.

Geometry variation 1 showed good attachment to the shroud surface, with a captive pocket in place of the hub centre body due to the discontinuous nature of the hub line. Overall pressure rise was not significantly influenced by this compared to the nominal case, however a spike in meridional static pressure is observed. This is due to the meridional line passing through the high pressure circulation region. Compared with the nominal case geometry variation 2 shows a smaller pressure drop along the shroud line. Bulk pressure recovery however is lower, but is not affected significantly. Geometry variation 3 presents a similar flow distribution on the bend to the nominal case and geometry variation 2, however with a small difference between shroud and hub pressures, however overall pressure recovery was adversely effected in comparison.

From comparing the 4 geometries, it appears that the largest parameter of influence on pressure rise is a₁ (passage height to equivalent diameter ratio). A value close to that presented in the literature (0.15) being in the optimal region. Lower values, as illustrated by geometry 3, showed lower pressure recovery. Higher values of a₁ constrain r to be unsuitably small, and geometries were non-physical. As a geometric alternative, lower values of a₁ may be suitable if the shroud surface of the bend in the diffuser is not constrained to a circular profile.

The impact of the diffuser inlet length appears to be minimal on overall pressure rise performance, however it remains uncertain if the inlet section needs to be sufficiently large to accommodate non-uniform mass flow rate from turbine exhaust. Further simulation work should be undertaken to verify this with shorter inlet lengths than simulated in geometry variation 2.

Overall, high values of pressure rise coefficient were observed when compared to radial-radial diffusers operating on air as presented by Japikse [3] in the range 0.7-0.8. A probable cause for this is the lower viscosity of CO₂, and hence lower viscous losses.

6. Conclusions and recommendations

Overall the design method adapted from Moller [7] appears to be suitable for the preliminary design of an annular-radial diffuser. Design intent is confirmed with pressure recovery focused in the radial-radial section of each geometry. Comparing values of pressure rise coefficient, the nominal design produced favourable results compared to geometry variations. Additionally, a similar trend was observed regarding the omission of the hub centre body to that presented in Moller [7].

Diffuser geometries with passage height to equivalent diameter ratios in the region of 0.15 showed the highest pressure recovery. Further nominal flow studies should be conducted with
passage height to equivalent diameter ratios in this region in order to determine the optimum. The presence or absence of a hub centre body showed a minimal impact on pressure recovery, and it’s impact on manufacturing and costs should be considered for future designs. Inlet length did not appear to be significant to pressure rise, and should be minimised in order to reduce shaft bending.

The key result from this work is that a short and simple diffuser design (nominal and geometry variations 1 and 2) can attain static pressure rise coefficients greater than 0.88. This confirms that annula-radial diffusers are a viable design solution for supercritical CO₂ radial inflow turbines, thus enabling an alternative cantilevered rotor layout.

In selection of a finalised design, robustness to off-design operating conditions should be quantified. Off-design diffuser inlet conditions should be considered with particular attention to swirl, and the impact on flow separation on the bend for such a design. To enhance the reliability of simulations, the hub line should be modelled as a no-slip wall rotating boundary in future work. Future work should focus on optimising the present nominal design with respect to off-design performance and manufacturing constraints.

References
[1] Dyreby, J., Klein, S., Nellis, G., and Reindl, D., 2014. “Design considerations for supercritical carbon dioxide brayton cycles with recompression”. Journal of Engineering for Gas Turbines and Power, 136(10), Jul, p. 101701.
[2] Dostal, V., 2004. “A supercritical carbon dioxide cycle for next generation nuclear reactors”. PhD thesis, Massachusetts Institute of Technology, http://hdl.handle.net/1721.1/17746.
[3] Japikse, D., and Baines, N. C., 1998. Turbomachinery Diffuser Design Technology. Concepts Eti.
[4] Monje, B., Sanchez, D., Chacartegui, R., Sanchez, T., Savill, M., and Pilidis, P., 2013. “Aerodynamic analysis of conical diffusers operating with air and supercritical carbon dioxide”. International Journal of Heat and Fluid Flow, 44, Sep, pp. 542 – 553.
[5] Sovran, G., and Klomp, E., 1965. “Experimentally determined optimum geometries for rectilinear diffusers with rectangular, conical or annular cross-section”. In Fluid mechanics of internal flow - Proceedings of the symposium of the fluid mechanics of internal flow, G. Sovran, ed.
[6] Dolan, F. X., and Runstadler, P. W., 1973. Pressure recovery performance of conical diffusers at high subsonic mach numbers. Technical Report NASA-CR-2299, NASA, Jul.
[7] Moller, P. S., 1966. “A radial diffuser using incompressible flow between narrowly spaced disks”. Journal of Basic Engineering, 88(1), Mar., pp. 155–162.
[8] Zhu, Y., and Sjolander, S. A., 1987. “Effect of geometry on the performance of radial vaneless diffusers”. Journal of Turbomachinery, 109(4), Oct, pp. 550–556.
[9] ANSYS Academic Research, 2015, ANSYS Release 16.1, Theory Manual, Inc., Canonsburg, P.A.
[10] Colonna, P., and Van der Stelt, T., 2016. Fluidprop: a program for the estimation of thermo physical properties of fluids.
[11] Lemmon, E. W., Huber, M. L., and McLinden, M. O., 2010. “Nist standard reference database 23: Reference fluid thermodynamic and transport properties - REFPROP”. Version 9.0. National Institute of Standards and Technology, Standard Reference Data Program, Gaithersburg.